ANSYS Mechanical APDL Element Reference

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 15.0
November 2013

ANSYS, Inc. is
certified to ISO
9001:2008.

Copyright and Trademark Information

© 2013 SAS IP, Inc. All rights reserved. Unauthorized use, distribution or duplication is prohibited.

ANSYS, ANSYS Workbench, Ansoft, AUTODYN, EKM, Engineering Knowledge Manager, CFX, FLUENT, HFSS and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. is certified to ISO 9001:2008.

U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the legal information in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, please contact ANSYS, Inc.

Published in the U.S.A.

Table of Contents

1. About This Reference ... 1
1.1. Overview ... 1
1.2. Conventions Used in This Reference ... 1
1.2.1. Applicable Products ... 2
1.2.2. Product Codes ... 3
1.3. Product Capabilities ... 3
2. Element Classifications ... 5
2.1. Overview of Element Characteristics ... 5
2.1.1. Identifying Element Types ... 5
2.1.2. 2-D vs. 3-D Elements ... 5
2.1.3. Element Shape Characteristics ... 6
2.1.4. Element Degrees of Freedom and Discipline ... 6
2.1.5. User-Defined Elements ... 6
2.2. Summary of Element Types ... 6
2.3. Selecting Elements for Your Analysis ... 22
2.4. Current-Technology Elements ... 37
2.4.1. Legacy vs. Current Element Technologies ... 38
2.5. Degenerated Shape Elements ... 39
2.5.1. Obtaining the Best Results from Degenerated Shape Elements ... 39
2.5.2. Degenerated Triangular 2-D Solid and Shell Elements ... 39
2.5.2.1. Triangular Shell Elements Are Preferred in Some Cases ... 39
2.5.3. Degenerated Triangular Prism Elements ... 40
2.5.4. Degenerated Tetrahedral Elements ... 40
2.5.5. Surface Stresses on Condensed Element Faces ... 40
2.6. Special-Purpose Elements ... 40
2.6.1. Superelements ... 41
2.6.2. User-Defined Elements ... 41
2.7. GUI-Inaccessible Elements ... 41
3. General Element Features ... 43
3.1. Element Input ... 43
3.1.1. Element Name ... 43
3.1.2. Nodes ... 43
3.1.2.1. Internal nodes ... 44
3.1.3. Degrees of Freedom ... 44
3.1.3.1. Internal Degrees of Freedom ... 44
3.1.4. Material Properties ... 44
3.1.5. Special Element Features ... 45
3.1.6. Sections ... 45
3.1.7. Real Constants ... 46
3.1.8. Element Loading ... 46
3.1.8.1. Surface Loads ... 47
3.1.8.1.1. Applying Surface Loads ... 47
3.1.8.1.2. Multiple Surface Loads ... 47
3.1.8.1.3. Surface Load Labels and Keys ... 48
3.1.8.1.4. Tapered Surface Load ... 48
3.1.8.2. Body Loads ... 48
3.1.8.2.1. Applying Body loads ... 48
3.1.8.2.2. Temperature Body Loads ... 49
3.1.8.3. Inertial Loads ... 49
3.1.8.4. Initial Stresses ... 49

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.1.8.5. Ocean Loads ... 49
3.1.9. Nodal Loading ... 49
3.1.10. Element Properties and Controls (KEYOPTs) ... 50
3.1.10.1. Element KEYOPT Defaults ... 50
3.1.10.1.1. Consider Automatic Element Selection ... 50
3.1.10.1.2. KEYOPT Settings May Vary Between Products ... 50
3.2. Solution Output ... 50
3.2.1. Output File Contents ... 51
3.2.2. Results File and Database Contents ... 51
3.2.3. Element Solution ... 51
3.2.3.1. The Element Output Definitions Table ... 52
3.2.3.2. The Item and Sequence Number Table ... 52
3.2.3.3. Surface Loads ... 53
3.2.3.4. Integration Point Solution (Output Listing Only) ... 53
3.2.3.5. Centroidal Solution (Output Listing Only) ... 53
3.2.3.6. Element Nodal Solution ... 53
3.2.3.7. Element Nodal Loads ... 54
3.2.3.8. Nonlinear Solutions ... 54
3.2.3.9. 2-D and Axisymmetric Solutions ... 54
3.2.3.10. Member Force Solution ... 54
3.2.3.11. Failure Criteria ... 54
3.2.4. Nodal Solution ... 55
3.3. Coordinate Systems ... 55
3.3.1. Understanding the Element Coordinate System ... 56
3.3.1.1. Modifying the Element Orientation ... 56
3.3.2. Elements That Operate in the Nodal Coordinate System ... 57
3.3.3. Solution Coordinate Systems ... 58
4. Element Support for Material Models ... 59
5. Elements for Stress Analysis ... 67
5.1. Continuum Stress Elements ... 67
5.1.1. Stress States in 2-D Continuum Elements ... 67
5.1.1.1. Plane Stress ... 68
5.1.1.2. Plane Strain ... 68
5.1.1.3. Generalized Plane Strain ... 68
5.1.1.4. Axisymmetric ... 68
5.1.1.5. General Axisymmetric ... 68
5.1.1.5.1. Alternate General Axisymmetric Method for Linear Problems ... 69
5.1.2. Element Technologies ... 69
5.1.2.1. Technologies for Lower-Order Quad and Brick Elements ... 69
5.1.2.1.1. B-bar Method (Selective Reduced Integration) ... 70
5.1.2.1.2. Enhanced Strain Formulation ... 70
5.1.2.1.3. Simplified Enhanced Strain Formulation ... 70
5.1.2.1.4. Uniform Reduced Integration ... 70
5.1.2.2. Technologies for Lower-Order Tetrahedron Elements ... 71
5.1.2.3. Technologies for Higher-Order Quad or Brick Elements ... 71
5.1.2.4. Technologies for Higher-Order Triangle or Tetrahedron Elements ... 71
5.1.3. Mixed u-P Formulation ... 71
5.1.3.1. Overview of Mixed u-P Element Technologies ... 71
5.1.3.2. Mixed u-P Element Summary ... 72
5.1.3.3. Applications of Mixed u-P Formulations ... 73
5.1.3.4. Mixed u-P Models and Overconstraint or No Unique Solution ... 74
5.1.4. Generalized Plane Strain Options ... 74

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.1.5. Axisymmetric Elements with Nonaxisymmetric Loads ... 76
5.1.5.1. General Axisymmetric Elements ... 76
5.1.5.1.1. Example: 3-D General Axisymmetric Model ... 79
5.1.5.2. Harmonic Axisymmetric Elements ... 83
5.1.5.2.1. Harmonic Axisymmetric Elements with Nonaxisymmetric Loads ... 84
5.2. Shell Elements ... 88
5.2.1. Axisymmetric Shells ... 88
5.2.2. Shell Stress and Stiffness ... 88
5.2.3. Membrane and Surface Stress Options ... 88
5.2.4. Shell Shear and Warping ... 89
5.2.5. Curved Shell Elements ... 89
5.2.6. Shell Element Coordinate System ... 89
5.2.7. Shell Element Nodes ... 89
5.3. Beam and Link (Truss) Elements ... 90
5.4. Pipe Elements ... 90
5.5. Reinforcing Elements ... 91
5.5.1. Discrete Reinforcing ... 91
5.5.2. Smeared Reinforcing ... 91
5.6. Surface Elements ... 91
5.7. Automatic Selection of Element Technologies and Formulations ... 92
5.8. Elements Under Linear Perturbation ... 95
5.8.1. Material Properties of Structural Elements in Linear Perturbation ... 96
5.8.1.1. Specifying Material Behavior in Linear Perturbation ... 96
5.8.2. Interpretation of Structural Element Results After a Linear Perturbation Analysis ... 97
5.8.3. Loads, Initial Conditions, and Other Limitations in Linear Perturbation ... 97
6. Elements for Multiphysics Analysis ... 99
6.1. Elements for Coupled-Physics Analysis ... 99
6.1.1. Coupled-Field Element Classifications ... 99
6.1.2. Selecting Elements for Coupled-Physics Analysis ... 100
6.2. Elements for Thermal Analysis ... 102
6.2.1. Thermal Elements for the Structural Region ... 102
6.2.2. Thermal Elements for the Fluid Region ... 103
6.2.3. Thermal Loading Elements ... 103
6.2.3.1. Using Thermal Surface-Loading Elements ... 103
6.2.3.2. Using Contact Elements to Load Thermal Models ... 104
6.2.4. Thermal Control Element ... 104
6.2.5. Other Elements Useful in a Thermal Analysis ... 104
6.3. Elements for Acoustic Analysis ... 104
6.3.1. Acoustic Element Types ... 105
6.3.2. 3-D Acoustic Element Usage ... 105
6.3.3. 3-D Acoustic Element Input ... 105
6.3.4. 3-D Acoustic Element Input Summary ... 108
6.3.5. 3-D Acoustic Element Item Output Data ... 109
6.3.6. 3-D Acoustic Element Assumptions and Restrictions ... 111
6.3.7. 3-D Acoustic Element Product Restrictions ... 112
6.4. Elements for Electric and Magnetic Analysis ... 112
6.5. Elements for Diffusion Analysis ... 114
7. Element Library ... 115
I. Element Library ... 117
5. SOLID5: 3-D Coupled-Field Solid ... 119
9. INFIN9: 2-D Infinite Boundary ... 129
11. LINK11: Linear Actuator ... 133

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13. PLANE13: 2-D Coupled-Field Solid 137
14. COMBIN14: Spring-Damper 147
21. MASS21: Structural Mass 155
25. PLANE25: Axisymmetric-Harmonic 4-Node Structural Solid 159
27. MATRIX27: Stiffness, Damping, or Mass Matrix 167
28. SHELL28: Shear/Twist Panel 173
29. FLUID29: 2-D Axisymmetric Harmonic Acoustic Fluid 179
30. FLUID30: 3-D Acoustic Fluid 185
31. LINK31: Radiation Link 187
33. LINK33: 3-D Conduction Bar 191
34. LINK34: Convection Link 195
35. PLANE35: 2-D 6-Node Triangular Thermal Solid 201
36. SOURC36: Current Source 207
37. COMBIN37: Control 211
38. FLUID38: Dynamic Fluid Coupling 221
39. COMBIN39: Nonlinear Spring 225
40. COMBIN40: Combination 233
41. SHELL41: Membrane Shell 241
47. INFIN47: 3-D Infinite Boundary 249
50. MATRIX50: Superelement (or Substructure) 253
53. PLANE53: 2-D 8-Node Magnetic Solid 257
55. PLANE55: 2-D Thermal Solid 267
61. SHELL61: Axisymmetric-Harmonic Structural Shell 273
65. SOLID65: 3-D Reinforced Concrete Solid 289
68. LINK68: Coupled Thermal-Electric Line 299
70. SOLID70: 3-D Thermal Solid 305
71. MASS71: Thermal Mass 311
75. PLANE75: Axisymmetric-Harmonic 4-Node Thermal Solid 315
77. PLANE77: 2-D 8-Node Thermal Solid 321
78. PLANE78: Axisymmetric-Harmonic 8-Node Thermal Solid 327
83. PLANE83: Axisymmetric-Harmonic 8-Node Structural Solid 333
87. SOLID87: 3-D 10-Node Tetrahedral Thermal Solid 341
90. SOLID90: 3-D 20-Node Thermal Solid 347
94. CIRCU94: Piezoelectric Circuit 353
96. SOLID96: 3-D Magnetic Scalar Solid 361
97. SOLID97: 3-D Magnetic Solid 367
98. SOLID98: Tetrahedral Coupled-Field Solid 379
110. INFIN110: 2-D Infinite Solid 389
111. INFIN111: 3-D Infinite Solid 395
116. FLUID116: Coupled Thermal-Fluid Pipe 401
121. PLANE121: 2-D 8-Node Electrostatic Solid 413
122. SOLID122: 3-D 20-Node Electrostatic Solid 419
123. SOLID123: 3-D 10-Node Tetrahedral Electrostatic Solid 425
124. CIRCU124: Electric Circuit 431
125. CIRCU125: Diode 443
126. TRANS126: Electromechanical Transducer 449
129. FLUID129: 2-D Infinite Acoustic 459
130. FLUID130: 3-D Infinite Acoustic 463
131. SHELL131: 4-Node Thermal Shell 467
132. SHELL132: 8-Node Thermal Shell 475
136. FLUID136: 3-D Squeeze Film Fluid Element 483
138. FLUID138: 3-D Viscous Fluid Link Element 495

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

139. FLUID139: 3-D Slide Film Fluid Element 499
144. ROM144: Reduced Order Electrostatic-Structural 505
151. SURF151: 2-D Thermal Surface Effect 509
152. SURF152: 3-D Thermal Surface Effect 519
153. SURF153: 2-D Structural Surface Effect 529
154. SURF154: 3-D Structural Surface Effect 537
156. SURF156: 3-D Structural Surface Line Load Effect 547
157. SHELL157: Thermal-Electric Shell 553
159. SURF159: General Axisymmetric Surface 559
160. LINK160: Explicit 3-D Spar (or Truss) 569
161. BEAM161: Explicit 3-D Beam 573
162. PLANE162: Explicit 2-D Structural Solid 587
163. SHELL163: Explicit Thin Structural Shell 593
164. SOLID164: Explicit 3-D Structural Solid 605
165. COMBI165: Explicit Spring-Damper 611
166. MASS166: Explicit 3-D Structural Mass 615
167. LINK167: Explicit Tension-Only Spar 617
168. SOLID168: Explicit 3-D 10-Node Tetrahedral Structural Solid 621
169. TARGE169: 2-D Target Segment 627
170. TARGE170: 3-D Target Segment 635
171. CONTA171: 2-D 2-Node Surface-to-Surface Contact 647
172. CONTA172: 2-D 3-Node Surface-to-Surface Contact 665
173. CONTA173: 3-D 4-Node Surface-to-Surface Contact 683
174. CONTA174: 3-D 8-Node Surface-to-Surface Contact 701
175. CONTA175: 2-D/3-D Node-to-Surface Contact 719
176. CONTA176: 3-D Line-to-Line Contact 739
177. CONTA177: 3-D Line-to-Surface Contact 755
178. CONTA178: 3-D Node-to-Node Contact 769
179. PRETS179: Pretension 793
180. LINK180: 3-D Spar (or Truss) 797
181. SHELL181: 4-Node Structural Shell 805
182. PLANE182: 2-D 4-Node Structural Solid 821
183. PLANE183: 2-D 8-Node or 6-Node Structural Solid 829
184. MPC184: Multipoint Constraint Element 837
184link. MPC184-Link/Beam: Multipoint Constraint Element: Rigid Link or Rigid Beam 843
184slid. MPC184-Slider: Multipoint Constraint Element: Slider 849
184revo. MPC184-Revolute: Multipoint Constraint Element: Revolute Joint 853
184univ. MPC184-Universal: Multipoint Constraint Element: Universal Joint 863
184slot. MPC184-Slot: Multipoint Constraint Element: Slot Joint 873
184poin. MPC184-Point: Multipoint Constraint Element: Point-in-plane Joint 881
184tran. MPC184-Trans: Multipoint Constraint Element: Translational Joint 889
184cyl. MPC184-Cylin: Multipoint Constraint Element: Cylindrical Joint 897
184plan. MPC184-Planar: Multipoint Constraint Element: Planar Joint 909
184weld. MPC184-Weld: Multipoint Constraint Element: Weld Joint 921
184orie. MPC184-Orient: Multipoint Constraint Element: Orient Joint 927
184sphe. MPC184-Spherical: Multipoint Constraint Element: Spherical Joint 933
184gen. MPC184-General: Multipoint Constraint Element: General Joint 939
184scr. MPC184-Screw: Multipoint Constraint Element: Screw Joint 949
185. SOLID185: 3-D 8-Node Structural Solid 959
186. SOLID186: 3-D 20-Node Structural Solid 975
187. SOLID187: 3-D 10-Node Tetrahedral Structural Solid 991
188. BEAM188: 3-D 2-Node Beam 997

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

189. BEAM189: 3-D 3-Node Beam 1017
190. SOLSH190: 3-D 8-Node Structural Solid Shell 1037
192. INTER192: 2-D 4-Node Gasket 1047
193. INTER193: 2-D 6-Node Gasket 1051
194. INTER194: 3-D 16-Node Gasket 1055
195. INTER195: 3-D 8-Node Gasket 1061
200. MESH200: Meshing Facet 1065
201. FOLLW201: Follower Load 1071
202. INTER202: 2-D 4-Node Cohesive 1075
203. INTER203: 2-D 6-Node Cohesive 1079
204. INTER204: 3-D 16-Node Cohesive 1083
205. INTER205: 3-D 8-Node Cohesive 1087
208. SHELL208: 2-Node Axisymmetric Shell 1091
209. SHELL209: 3-Node Axisymmetric Shell 1101
212. CPT212: 2-D 4-Node Coupled Pore-Pressure Mechanical Solid 1111
213. CPT213: 2-D 8-Node Coupled Pore-Pressure Mechanical Solid 1117
214. COMBI214: 2-D Spring-Damper Bearing 1123
215. CPT215: 3-D 8-Node Coupled Pore-Pressure Mechanical Solid 1131
216. CPT216: 3-D 20-Node Coupled Pore-Pressure Mechanical Solid 1137
217. CPT217: 3-D 10-Node Coupled Pore-Pressure Mechanical Solid 1143
220. FLUID220: 3-D Acoustic Fluid 20-Node Solid Element 1149
221. FLUID221: 3-D Acoustic Fluid 10-Node Solid Element 1151
223. PLANE223: 2-D 8-Node Coupled-Field Solid 1153
226. SOLID226: 3-D 20-Node Coupled-Field Solid 1169
227. SOLID227: 3-D 10-Node Coupled-Field Solid 1185
230. PLANE230: 2-D 8-Node Electric Solid 1201
231. SOLID231: 3-D 20-Node Electric Solid 1207
232. SOLID232: 3-D 10-Node Tetrahedral Electric Solid 1213
233. PLANE233: 2-D 8-Node Electromagnetic Solid 1217
236. SOLID236: 3-D 20-Node Electromagnetic Solid 1227
237. SOLID237: 3-D 10-Node Electromagnetic Solid 1237
238. PLANE238: 2-D 8-Node Diffusion Solid 1247
239. SOLID239: 3-D 20-Node Diffusion Solid 1251
240. SOLID240: 3-D 10-Node Tetrahedral Diffusion Solid 1255
241. HSFLD241: 2-D Hydrostatic Fluid 1259
242. HSFLD242: 3-D Hydrostatic Fluid 1267
251. SURF251: 2-D Radiosity Surface 1273
252. SURF252: 3-D Radiosity Surface 1277
263. REINF263: 2-D Smeared Reinforcing 1281
264. REINF264: 3-D Discrete Reinforcing 1289
265. REINF265: 3-D Smeared Reinforcing 1297
272. SOLID272: General Axisymmetric Solid with 4 Base Nodes 1305
273. SOLID273: General Axisymmetric Solid with 8 Base Nodes 1313
278. SOLID278: 3-D 8-Node Thermal Solid 1321
279. SOLID279: 3-D 20-Node Thermal Solid 1333
281. SHELL281: 8-Node Structural Shell 1345
285. SOLID285: 3-D 4-Node Tetrahedral Structural Solid with Nodal Pressures 1359
288. PIPE288: 3-D 2-Node Pipe 1365
289. PIPE289: 3-D 3-Node Pipe 1383
290. ELBOW290: 3-D 3-Node Elbow 1401
300. USER300: User-Defined Element 1417
Bibliography 1419

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Index 1421

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

List of Figures

5.1. Generalized Plane Strain Deformation 75
5.2. General Axisymmetric Elements and Their Coordinate Systems (KEYOPT(2) = 3) 77
5.3. Axisymmetric Radial, Axial, Torsion and Moment Loadings 85
5.4. Bending and Shear Loading (ISYM = 1) 86
5.5. Uniform Lateral Loadings 87
5.6. Bending and Shear Loading (*ISYM* = -1) 87
5.7. Displacement and Force Loading Associated with *MODE* = 2 and *ISYM* = 1 87
5.1. SOLID5 Geometry 119
5.2. SOLID5 Element Output 123
9.1. INFIN9 Geometry 129
9.2. INFIN9 Element Usage 131
11.1. LINK11 Geometry 133
13.1. PLANE13 Geometry 137
13.2. PLANE13 Element Output 141
14.1. COMBIN14 Geometry 147
14.2. COMBIN14 Stress Output 150
21.1. MASS21 Geometry 155
25.1. PLANE25 Geometry 159
25.2. PLANE25 Stress Output 162
27.1. MATRIX27 Schematic 167
28.1. SHELL28 Geometry 173
28.2. SHELL28 Stress Output 175
29.1. FLUID29 Geometry 179
30.1. FLUID30 Geometry 185
31.1. LINK31 Geometry 187
33.1. LINK33 Geometry 191
34.1. LINK34 Geometry 195
35.1. PLANE35 Geometry 201
36.1. SOURC36 Geometry 207
37.1. COMBIN37 Geometry 211
37.2. COMBIN37 Behavior as a Function of Control Parameter 216
38.1. FLUID38 Geometry 221
39.1. COMBIN39 Geometry 225
39.2. COMBIN39 Force-Deflection Curves 229
40.1. COMBIN40 Geometry 233
40.2. COMBIN40 Behavior 236
41.1. SHELL41 Geometry 241
41.2. SHELL41 Stress Output 245
47.1. INFIN47 Geometry 249
50.1. MATRIX50 Schematic 253
53.1. PLANE53 Geometry 257
53.2. PLANE53 Magnetic Element Output 262
55.1. PLANE55 Geometry 267
61.1. SHELL61 Geometry 273
61.2. SHELL61 Stress Output 276
65.1. SOLID65 Geometry 289
65.2. SOLID65 Stress Output 294
68.1. LINK68 Geometry 299
70.1. SOLID70 Geometry 305
71.1. MASS71 Geometry 311

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

75.1. PLANE75 Geometry 315
77.1. PLANE77 Geometry 321
78.1. PLANE78 Geometry 327
83.1. PLANE83 Geometry 333
83.2. PLANE83 Stress Output 336
87.1. SOLID87 Geometry 341
90.1. SOLID90 Geometry 347
94.1. CIRCU94 Circuit Options 354
94.2. Load Functions and Corresponding Real Constants for Independent Current and Voltage Sources 357
96.1. SOLID96 Geometry 361
97.1. SOLID97 Geometry 368
98.1. SOLID98 Geometry 379
98.2. SOLID98 Element Output 382
110.1. INFIN110 Geometry 389
110.2. INFIN110 Element Output 392
111.1. INFIN111 Geometry 395
116.1. FLUID116 Geometry 401
121.1. PLANE121 Geometry 413
121.2. PLANE121 Output 415
122.1. SOLID122 Geometry 419
123.1. SOLID123 Geometry 425
124.1. CIRCU124 Circuit Element Options 432
124.2. CIRCU124 Circuit Source Options 432
124.3. CIRCU124 Coupled Circuit Source Options 433
124.4. Load Functions and Corresponding Real Constants for Independent Current and Voltage Sources 436
125.1. CIRCU125 Element Options 443
125.2. CIRCU125 I-U Characteristics 444
126.1. TRANS126 Geometry 449
126.2. TRANS126 Capacitance Relationship 450
126.3. TRANS126 Force Relationship 451
126.4. TRANS126 Valid/Invalid Orientations 457
129.1. FLUID129 Geometry 459
130.1. FLUID130 Geometry 463
131.1. SHELL131 Geometry 467
132.1. SHELL132 Geometry 475
136.1. FLUID136 Geometry 484
136.2. Moving Structure with Thin Film of FLUID136 Elements 488
136.3. Minimum Fluid and Minimum Mechanical Gaps 489
136.4. Mechanical Penetration 489
136.5. Fluid Penetration 489
138.1. FLUID138 Geometry 495
139.1. FLUID139 Geometry 500
144.1. ROM144 Schematic 505
151.1. SURF151 Geometry 509
152.1. SURF152 Geometry 519
153.1. SURF153 Geometry 529
153.2. Pressures 530
154.1. SURF154 Geometry 537
154.2. Pressures 538
156.1. SURF156 Geometry 547
156.2. Pressures 548
157.1. SHELL157 Geometry 553

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

159.1. SURF159 Geometry (KEYOPT(2) = 3) 559
159.2. SURF159 Pressures 561
160.1. LINK160 Geometry 569
161.1. BEAM161 Geometry 573
161.2. Standard Beam Cross Sections 575
161.3. Standard Beam Cross Sections (continued) 576
161.4. Definition of Relative Area for User Defined Integration Rule 584
161.5. Definition of Integration Points for User Defined Integration Rule 584
162.1. PLANE162 Geometry 587
162.2. PLANE162 Stress Output 591
163.1. SHELL163 Geometry 593
163.2. Arbitrary Ordering of Integration Points for User Defined Shell Integration Rule 594
163.3. Nodal Numbering for Pressure Loads (Positive Pressure Acts in Negative Z Direction) 595
164.1. SOLID164 Geometry 605
165.1. COMBI165 Geometry 611
166.1. MASS166 Geometry 615
167.1. LINK167 Geometry 617
168.1. SOLID168 Geometry 621
169.1. TARGE169 Geometry 627
169.2. TARGE169 2-D Segment Types 629
170.1. TARGE170 Geometry 636
170.2. TARGE170 Segment Types 639
171.1. CONTA171 Geometry 647
172.1. CONTA172 Geometry 665
173.1. CONTA173 Geometry 683
174.1. CONTA174 Geometry 701
175.1. CONTA175 Geometry 719
176.1. CONTA176 Geometry 739
176.2. Beam Sliding Inside a Hollow Beam 740
176.3. Parallel Beams in Contact 740
176.4. Crossing Beams in Contact 741
176.5. Equivalent Circular Cross Section 741
177.1. CONTA177 Geometry 755
178.1. CONTA178 Geometry 769
178.2. CONTA178 - Cylindrical Gap 775
178.3. CONTA178 - Spherical Gap 776
179.1. PRETS179 Geometry 793
180.1. LINK180 Geometry 797
180.2. LINK180 Stress Output 799
181.1. SHELL181 Geometry 806
181.2. SHELL181 Typical Bending Applications 809
181.3. SHELL181 Stress Output 814
182.1. PLANE182 Geometry 821
182.2. PLANE182 Stress Output 824
183.1. PLANE183 Geometry 829
183.2. PLANE183 Stress Output 832
184link.1. MPC184 Rigid Link/Beam Geometry 843
184slid.1. MPC184 Slider Geometry 849
184revo.1. MPC184 Revolute Joint Geometry 853
184univ.1. MPC184 Universal Joint Geometry 863
184slot.1. MPC184 Slot Joint Geometry 873
184poin.1. MPC184 Point-in-plane Joint Geometry 881

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

184tran.1. MPC184 Translational Joint Geometry 889
184cyl.1. MPC184 Cylindrical Joint Geometry 897
184plan.1. MPC184 Planar Joint Geometry 909
184weld.1. MPC184 Weld Joint Geometry 921
184orie.1. MPC184 Orient Joint Geometry 927
184sphe.1. MPC184 Spherical Joint Geometry 933
184gen.1. MPC184 General Joint Geometry 939
184scr.1. MPC184 Screw Joint Geometry 949
185.1. SOLID185 Homogeneous Structural Solid Geometry 960
185.2. SOLID185 Homogeneous Structural Solid Stress Output 963
185.3. SOLID185 Layered Structural Solid Geometry 966
185.4. SOLID185 Layered Structural Solid Stress Output 970
186.1. SOLID186 Homogeneous Structural Solid Geometry 976
186.2. SOLID186 Homogeneous Structural Solid Stress Output 979
186.3. SOLID186 Layered Structural Solid Geometry 982
186.4. SOLID186 Layered Structural Solid Stress Output 986
187.1. SOLID187 Geometry 991
187.2. SOLID187 Stress Output 993
188.1. BEAM188 Geometry 997
188.2. Transverse-Shear Deformation Estimation 998
188.3. Cross-Section Cells 1001
189.1. BEAM189 Geometry 1017
189.2. Transverse-Shear Deformation Estimation 1018
189.3. Cross-Section Cells 1021
190.1. SOLSH190 Geometry 1037
190.2. SOLSH190 Stress Output 1042
192.1. INTER192 Geometry 1047
192.2. INTER192 Stress Output 1049
193.1. INTER193 Geometry 1051
193.2. INTER193 Stress Output 1053
194.1. INTER194 Geometry 1055
194.2. INTER194 3-D 16-Node Degenerated Quadratic Interface 1056
194.3. INTER194 Stress Output 1058
195.1. INTER195 Geometry 1061
195.2. INTER195 Stress Output 1063
200.1. MESH200 Geometry 1065
201.1. FOLLW201 Geometry 1071
202.1. INTER202 Geometry 1075
202.2. INTER202 Stress Output 1077
203.1. INTER203 Geometry 1079
203.2. INTER203 Stress Output 1081
204.1. INTER204 Geometry 1083
204.2. INTER204 3-D 16-Node Degenerated Quadratic Interface 1084
204.3. INTER204 Stress Output 1085
205.1. INTER205 Geometry 1087
205.2. INTER205 Stress Output 1089
208.1. SHELL208 Geometry 1091
208.2. SHELL208 Element Stress Output 1096
209.1. SHELL209 Geometry 1101
209.2. SHELL209 Element Stress Output 1105
212.1. CPT212 Geometry 1111
212.2. CPT212 Stress Output 1113

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

213.1. CPT213 Geometry 1117
213.2. CPT213 Stress Output 1119
214.1. COMBI214 Geometry 1124
215.1. CPT215 Structural Solid Geometry 1131
215.2. CPT215 Stress Output 1133
216.1. CPT216 Geometry 1137
216.2. CPT216 Stress Output 1140
217.1. CPT217 Geometry 1143
217.2. CPT217 Stress Output 1145
220.1. FLUID220 Geometry 1149
221.1. FLUID221 Geometry 1151
223.1. PLANE223 Geometry 1153
226.1. SOLID226 Geometry 1170
227.1. SOLID227 Geometry 1186
230.1. PLANE230 Geometry 1201
230.2. PLANE230 Output 1203
231.1. SOLID231 Geometry 1207
232.1. SOLID232 Geometry 1213
233.1. PLANE233 Geometry 1217
236.1. SOLID236 Geometry 1228
237.1. SOLID237 Geometry 1238
238.1. PLANE238 Geometry 1247
238.2. PLANE238 Output 1249
239.1. SOLID239 Geometry 1251
240.1. SOLID240 Geometry 1255
241.1. HSFLD241 Geometry 1259
242.1. HSFLD242 Geometry 1267
251.1. SURF251 Geometry 1273
251.2. SURF251 Elements Without Coincident Nodes 1274
251.3. Net Radiation Heat Flux 1276
252.1. SURF252 Geometry 1277
252.2. SURF252 Elements Without Coincident Nodes 1278
252.3. Net Radiation Heat Flux 1280
263.1. REINF263 Geometry 1281
263.2. REINF263 Coordinate System 1282
263.3. REINF263 Stress Output 1284
264.1. REINF264 Coordinate System 1291
264.2. REINF264 Stress Output 1293
265.1. REINF265 Geometry 1297
265.2. REINF265 Coordinate System 1299
265.3. REINF265 Stress Output 1302
272.1. SOLID272 Geometry (KEYOPT(2) = 3) 1305
272.2. SOLID272 Stress Output 1308
273.1. SOLID273 Geometry (KEYOPT(2) = 3) 1313
273.2. SOLID273 Stress Output 1316
278.1. SOLID278 Homogeneous Thermal Solid Geometry 1322
278.2. SOLID278 Layered Thermal Solid Geometry 1326
278.3. SOLID278 Layered Thermal Solid Heat Flux/Temperature Gradient Output 1329
279.1. SOLID279 homogeneous Thermal Solid Geometry 1334
279.2. SOLID279 homogeneous Thermal Solid Temperature Gradient/Heat Flux Output 1336
279.3. SOLID279 Layered Thermal Solid Geometry 1339
279.4. SOLID279 Layered Thermal Solid Temperature Gradient/Heat Flux Output 1342

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

281.1. SHELL281 Geometry 1346
281.2. SHELL281 Stress Output 1352
285.1. SOLID285 Geometry 1359
285.2. SOLID285 Stress Output 1361
288.1. PIPE288 Geometry 1365
288.2. Transverse-Shear-Deformation Estimation 1366
288.3. Typical Cross-Section Cell 1369
289.1. PIPE289 Geometry 1383
289.2. Transverse-Shear-Deformation Estimation 1384
289.3. Typical Cross-Section Cell 1386
290.1. ELBOW290 Geometry 1401
290.2. ELBOW290 Element Integration Stations 1409
290.3. ELBOW290 Stress Output 1410

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

List of Tables

3.1. Surface Loads Available in Each Discipline 47
3.2. Body Loads Available in Each Discipline 48
5.1. Number of Independent Pressure Degrees of Freedom in One Element 72
5.2. Fourier Terms for General Axisymmetric Elements 78
5.3. Recommendation Criteria for Element Technology (Linear Material) 93
5.4. Recommendation Criteria for Element Technology (Nonlinear Materials) 94
5.5. Elements Supporting Linear Perturbation Analysis 95
6.1. 3-D Acoustic Element Output Definitions 110
6.2. 3-D Acoustic Element Item and Sequence Numbers 111
5.1. SOLID5 Element Output Definitions 123
5.2. SOLID5 Item and Sequence Numbers 125
11.1. LINK11 Element Output Definitions 134
11.2. LINK11 Item and Sequence Numbers 135
13.1. PLANE13 Element Output Definitions 141
13.2. PLANE13 Miscellaneous Element Output 143
13.3. PLANE13 Item and Sequence Numbers 144
14.1. COMBIN14 Real Constants 150
14.2. COMBIN14 Element Output Definitions 151
14.3. COMBIN14 Item and Sequence Numbers 151
25.1. PLANE25 Element Output Definitions 163
25.2. PLANE25 Item and Sequence Numbers 164
28.1. SHELL28 Element Output Definitions 175
28.2. SHELL28 Item and Sequence Numbers 176
29.1. FLUID29 Element Output Definitions 182
29.2. FLUID29 Item and Sequence Numbers 183
31.1. LINK31 Element Output Definitions 189
31.2. LINK31 Item and Sequence Numbers 190
33.1. LINK33 Element Output Definitions 192
33.2. LINK33 Item and Sequence Numbers 193
34.1. LINK34 Element Output Definitions 197
34.2. LINK34 Item and Sequence Numbers 198
35.1. PLANE35 Element Output Definitions 203
35.2. PLANE35 Item and Sequence Numbers 204
36.1. SOURC36 Real Constants 209
37.1. COMBIN37 Real Constants 215
37.2. COMBIN37 Element Output Definitions 217
37.3. COMBIN37 Item and Sequence Numbers 218
38.1. FLUID38 Real Constants 223
39.1. COMBIN39 Real Constants 228
39.2. COMBIN39 Element Output Definitions 230
39.3. COMBIN39 Item and Sequence Numbers 231
40.1. COMBIN40 Element Output Definitions 237
40.2. COMBIN40 Item and Sequence Numbers 238
41.1. SHELL41 Real Constants 244
41.2. SHELL41 Element Output Definitions 245
41.3. SHELL41 Miscellaneous Element Output 246
41.4. SHELL41 Item and Sequence Numbers 247
53.1. DOF and Force Labels 257
53.2. PLANE53 Real Constants 261
53.3. PLANE53 Element Output Definitions 262

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

53.4. PLANE53 Miscellaneous Element Output 263
53.5. PLANE53 Item and Sequence Numbers 264
55.1. PLANE55 Element Output Definitions 270
55.2. PLANE55 Item and Sequence Numbers 271
61.1. SHELL61 Element Output Definitions 276
61.2. SHELL61 Item and Sequence Numbers (KEYOPT(6) = 0 or 1) 278
61.3. SHELL61 Item and Sequence Numbers (KEYOPT(6) = 3) 279
61.4. SHELL61 Item and Sequence Numbers (KEYOPT(6) = 5) 281
61.5. SHELL61 Item and Sequence Numbers (KEYOPT(6) = 7) 283
61.6. SHELL61 Item and Sequence Numbers (KEYOPT(6) = 9) 285
65.1. SOLID65 Concrete Material Data 293
65.2. SOLID65 Element Output Definitions 294
65.3. SOLID65 Miscellaneous Element Output 296
65.4. SOLID65 Element Status Table 296
65.5. SOLID65 Item and Sequence Numbers 297
68.1. LINK68 Element Output Definitions 301
68.2. LINK68 Item and Sequence Numbers 301
70.1. SOLID70 Element Output Definitions 308
70.2. SOLID70 Item and Sequence Numbers 309
71.1. MASS71 Real Constants 313
71.2. MASS71 Element Output Definitions 313
71.3. MASS71 Item and Sequence Numbers 314
75.1. PLANE75 Element Output Definitions 317
75.2. PLANE75 Item and Sequence Numbers 318
77.1. PLANE77 Element Output Definitions 323
77.2. PLANE77 Item and Component Labels 324
78.1. PLANE78 Element Output Definitions 329
78.2. PLANE78 Item and Sequence Numbers 330
83.1. PLANE83 Element Output Definitions 336
83.2. PLANE83 Item and Sequence Numbers 338
87.1. SOLID87 Element Output Definitions 343
87.2. SOLID87 Item and Sequence Numbers 344
90.1. SOLID90 Element Output Definitions 349
90.2. SOLID90 Item and Sequence Numbers 350
94.1. CIRCU94 Real Constants 356
94.2. CIRCU94 Element Output Definitions 358
94.3. CIRCU94 Item and Sequence Numbers 359
96.1. SOLID96 Element Output Definitions 364
96.2. SOLID96 Miscellaneous Element Output 364
96.3. SOLID96 Item and Sequence Numbers 365
97.1. DOF and Force Labels 367
97.2. SOLID97 Real Constants 372
97.3. SOLID97 Element Output Definitions 373
97.4. SOLID97 Miscellaneous Element Output 375
97.5. SOLID97 Item and Sequence Numbers 375
98.1. SOLID98 Element Output Definitions 383
98.2. SOLID98 Miscellaneous Element Output 384
98.3. SOLID98 Item and Sequence Numbers 385
110.1. INFIN110 Analyses 391
110.2. INFIN110 Material Properties 391
110.3. Element Output Definitions 392
110.4. INFIN110 Item and Sequence Numbers 393

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

111.1. INFIN111 Analyses ... 397
111.2. INFIN111 Material Properties ... 397
111.3. INFIN111 Element Output Definitions ... 398
111.4. INFIN111 Item and Sequence Numbers ... 399
116.1. FLUID116 Element Real Constants ... 406
116.2. FLUID116 Empirical Data Table (Optional) ... 407
116.3. FLUID116 Element Output Definitions ... 408
116.4. FLUID116 Item and Sequence Numbers ... 409
121.1. PLANE121 Element Output Definitions ... 416
121.2. PLANE121 Miscellaneous Element Output ... 417
121.3. PLANE121 Item and Sequence Numbers ... 417
122.1. SOLID122 Element Output Definitions ... 421
122.2. SOLID122 Miscellaneous Element Output ... 423
122.3. SOLID122 Item and Sequence Numbers ... 423
123.1. SOLID123 Element Output Definitions ... 427
123.2. SOLID123 Miscellaneous Element Output ... 428
123.3. SOLID123 Item and Sequence Numbers ... 429
124.1. CIRCU124 Real Constants ... 435
124.2. CIRCU124 Element Output Definitions ... 438
124.3. CIRCU124 Item and Sequence Numbers ... 441
125.1. CIRCU125 Real Constants ... 445
125.2. CIRCU125 Element Output Definitions ... 446
125.3. CIRCU125 Item and Sequence Numbers ... 447
126.1. TRANS126 Real Constants ... 453
126.2. TRANS126 Element Output Definitions ... 454
126.3. TRANS126 Item and Sequence Numbers ... 455
129.1. FLUID129 Element Output Definitions ... 461
129.2. FLUID129 Item and Sequence Numbers ... 461
130.1. FLUID130 Element Output Definitions ... 465
130.2. FLUID130 Item and Sequence Numbers ... 465
131.1. SHELL131 Element Output Definitions ... 471
131.2. SHELL131 Item and Sequence Numbers ... 472
132.1. SHELL132 Element Output Definitions ... 479
132.2. SHELL132 Item and Sequence Numbers ... 480
136.1. FLUID136 Real Constants ... 487
136.2. FLUID136 Element Output Definitions ... 490
136.3. FLUID136 Item and Sequence Numbers ... 491
138.1. FLUID138 Element Output Definitions ... 497
138.2. FLUID138 Item and Sequence Numbers ... 498
139.1. FLUID139 Element Output Definitions ... 502
139.2. FLUID139 Item and Sequence Numbers ... 503
144.1. ROM144 Element Output Definitions ... 507
144.2. ROM Item and Sequence Numbers ... 508
151.1. SURF151 Real Constants ... 514
151.2. SURF151 Element Output Definitions ... 515
151.3. SURF151 Item and Sequence Numbers ... 516
152.1. SURF152 Real Constants ... 524
152.2. SURF152 Element Output Definitions ... 525
152.3. SURF152 Item and Sequence Numbers ... 527
153.1. SURF153 Real Constants ... 533
153.2. SURF153 Element Output Definitions ... 534
153.3. SURF153 Item and Sequence Numbers ... 535

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

154.1. SURF154 Real Constants 541
154.2. SURF154 Element Output Definitions 542
154.3. SURF154 Item and Sequence Numbers 543
156.1. SURF156 Element Output Definitions 550
156.2. SURF156 Item and Sequence Numbers 551
157.1. SHELL157 Element Output Definitions 555
157.2. SHELL157 Item and Sequence Numbers 556
159.1. SURF159 Real Constants 564
159.2. SURF159 Element Output Definitions 565
159.3. SURF159 Item and Sequence Numbers 565
161.1. BEAM161 Real Constants 578
161.2. Properties of Beam Cross Sections for Several Common Cross Sections 582
161.3. BEAM161 Element Output Definitions 585
161.4. BEAM161 Item and Sequence Numbers 585
162.1. PLANE162 Element Output Definitions 591
162.2. PLANE162 Item and Sequence Numbers 592
163.1. SHELL163 Real Constants 599
163.2. SHELL163 Element Output Definitions 601
163.3. SHELL163 Item and Sequence Numbers 602
164.1. SOLID164 Element Output Definitions 608
165.1. COMBI165 Item and Sequence Numbers 613
168.1. SOLID168 Element Output Definitions 624
169.1. TARGE169 2-D Segment Types, Target Shape Codes, and Nodes 628
169.2. TARGE169 Element Output Definitions 633
169.3. TARGE169 Item and Sequence Numbers 634
170.1. TARGE170 3-D Segment Types, Target Shape Codes, and Nodes 637
170.2. TARGE170 Element Output Definitions 644
170.3. TARGE170 Item and Sequence Numbers 645
171.1. CONTA171 Real Constants 655
171.2. CONTA171 Element Output Definitions 657
171.3. CONTA171 Item and Sequence Numbers 660
172.1. CONTA172 Real Constants 673
172.2. CONTA172 Element Output Definitions 675
172.3. CONTA172 Item and Sequence Numbers 678
173.1. CONTA173 Real Constants 692
173.2. CONTA173 Element Output Definitions 694
173.3. CONTA173 Item and Sequence Numbers 697
174.1. CONTA174 Real Constants 710
174.2. CONTA174 Element Output Definitions 712
174.3. CONTA174 Item and Sequence Numbers 715
175.1. CONTA175 Real Constants 727
175.2. CONTA175 Element Output Definitions 729
175.3. CONTA175 (3-D) Item and Sequence Numbers 733
175.4. CONTA175 (2-D) Item and Sequence Numbers 734
176.1. CONTA176 Real Constants 748
176.2. CONTA176 Element Output Definitions 750
176.3. CONTA176 Item and Sequence Numbers 752
177.1. CONTA177 Real Constants 761
177.2. CONTA177 Element Output Definitions 763
177.3. CONTA177 (3-D) Item and Sequence Numbers 766
178.1. CONTA178 Real Constants 785
178.2. Real Constants and Corresponding Primary Variables for CONTA178 785

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

178.3. CONTA178 Element Output Definitions 787
178.4. CONTA178 Item and Sequence Numbers 788
180.1. LINK180 Element Output Definitions 800
180.2. LINK180 Item and Sequence Numbers 802
181.1. SHELL181 Element Output Definitions 815
181.2. SHELL181 Item and Sequence Numbers 817
182.1. PLANE182 Element Output Definitions 824
182.2. PLANE182 Item and Sequence Numbers 826
183.1. PLANE183 Element Output Definitions 832
183.2. PLANE183 Item and Sequence Numbers 834
184link.1. MPC184 Rigid Link/Beam Element Output Definitions 845
184link.2. MPC184 Rigid Link/Beam Item and Sequence Numbers 846
184slid.1. MPC184 Slider Element Output Definitions 850
184slid.2. MPC184 Slider Item and Sequence Numbers 851
184revo.1. MPC184 Revolute Joint Element Output Definitions 856
184revo.2. MPC184 Revolute Joint Element - NMISC Output 858
184revo.3. MPC184 Revolute Joint Item and Sequence Numbers - SMISC Items 859
184revo.4. MPC184 Revolute Joint Item and Sequence Numbers - NMISC Items 860
184univ.1. MPC184 Universal Joint Element Output Definitions 866
184univ.2. MPC184 Universal Joint Element - NMISC Output 868
184univ.3. MPC184 Universal Joint Item and Sequence Numbers - SMISC Items 868
184univ.4. MPC184 Universal Joint Item and Sequence Numbers - NMISC Items 869
184slot.1. MPC184 Slot Joint Element Output Definitions 875
184slot.2. MPC184 Slot Joint Element - NMISC Output 876
184slot.3. MPC184 Slot Joint Item and Sequence Numbers - SMISC Items 877
184slot.4. MPC184 Slot Joint Item and Sequence Numbers - NMISC Items 878
184poin.1. MPC184 Point-in-plane Joint Element Output Definitions 883
184poin.2. MPC184 Point-in-plane Joint Element - NMISC Output 885
184poin.3. MPC184 Point-in-plane Joint Item and Sequence Numbers - SMISC Items 885
184poin.4. MPC184 Point-in-plane Joint Item and Sequence Numbers - NMISC Items 886
184tran.1. MPC184 Translational Joint Element Output Definitions 891
184tran.2. MPC184 Translational Joint Element - NMISC Output 893
184tran.3. MPC184 Translational Joint Item and Sequence Numbers - SMISC Items 893
184tran.4. MPC184 Translational Joint Item and Sequence Numbers - NMISC Items 894
184cyl.1. MPC184 Cylindrical Joint Element Output Definitions 901
184cyl.2. MPC184 Cylindrical Joint Element - NMISC Output 903
184cyl.3. MPC184 Cylindrical Joint Item and Sequence Numbers - SMISC Items 904
184cyl.4. MPC184 Cylindrical Joint Item and Sequence Numbers - NMISC Items 905
184plan.1. MPC184 Planar Joint Element Output Definitions 912
184plan.2. MPC184 Planar Joint Element - NMISC Output 915
184plan.3. MPC184 Planar Joint Item and Sequence Numbers - SMISC Items 916
184plan.4. MPC184 Planar Joint Item and Sequence Numbers - NMISC Items 918
184weld.1. MPC184 Weld Joint Element Output Definitions 923
184weld.2. MPC184 Weld Joint Element - NMISC Output 923
184weld.3. MPC184 Weld Joint Item and Sequence Numbers - SMISC Items 924
184weld.4. MPC184 Weld Joint Item and Sequence Numbers - NMISC Items 924
184orie.1. MPC184 Orient Joint Element Output Definitions 929
184orie.2. MPC184 Orient Joint Element - NMISC Output 929
184orie.3. MPC184 Orient Joint Item and Sequence Numbers - SMISC Items 930
184orie.4. MPC184 Orient Joint Item and Sequence Numbers - NMISC Items 930
184sphe.1. MPC184 Spherical Joint Element Output Definitions 935
184sphe.2. MPC184 Spherical Joint Element - NMISC Output 936

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

184sphe.3. MPC184 Spherical Joint Item and Sequence Numbers -SMISC Items 937
184sphe.4. MPC184 Spherical Joint Item and Sequence Numbers - NMISC Items 937
184gen.1. MPC184 General Joint Element Output Definitions 943
184gen.2. MPC184 General Joint Element - NMISC Output 944
184gen.3. MPC184 General Joint Item and Sequence Numbers - SMISC Items 945
184gen.4. MPC184 General Joint Item and Sequence Numbers - NMISC Items 946
184scr.1. MPC184 Screw Joint Element Output Definitions 953
184scr.2. MPC184 Screw Joint Element - NMISC Output 954
184scr.3. MPC184 Screw Joint Item and Sequence Numbers - SMISC Items 955
184scr.4. MPC184 Screw Joint Item and Sequence Numbers - NMISC Items 955
185.1. SOLID185 Homogeneous Structural Solid Element Output Definitions 963
185.2. SOLID185 Homogeneous Structural Solid Item and Sequence Numbers 965
185.3. SOLID185 Layered Structural Solid Element Output Definitions 970
185.4. SOLID185 Layered Structural Solid Item and Sequence Numbers 973
186.1. SOLID186 Homogeneous Structural Solid Element Output Definitions 979
186.2. SOLID186 Homogeneous Structural Solid Item and Sequence Numbers 981
186.3. SOLID186 Layered Structural Solid Element Output Definitions 986
186.4. SOLID186 Layered Structural Solid Item and Sequence Numbers 988
187.1. SOLID187 Element Output Definitions 994
187.2. SOLID187 Item and Sequence Numbers 995
188.1. BEAM188 Element Output Definitions 1008
188.2. BEAM188 Item and Sequence Numbers 1011
189.1. BEAM189 Element Output Definitions 1028
189.2. BEAM 189 Item and Sequence Numbers 1030
190.1. SOLSH190 Element Output Definitions 1042
190.2. SOLSH190 Item and Sequence Numbers 1044
192.1. INTER192 Element Output Definitions 1049
193.1. INTER193 Element Output Definitions 1054
194.1. INTER194 Element Output Definitions 1058
195.1. INTER195 Element Output Definitions 1063
201.1. FOLLW201 Item and Sequence Numbers 1073
202.1. INTER202 Element Output Definitions 1077
203.1. INTER203 Element Output Definitions 1081
204.1. INTER204 Element Output Definitions 1085
205.1. INTER205 Element Output Definitions 1089
208.1. SHELL208 Element Output Definitions 1096
208.2. SHELL208 Item and Sequence Numbers 1098
209.1. SHELL209 Element Output Definitions 1106
209.2. SHELL209 Item and Sequence Numbers 1108
212.1. CPT212 Element Output Definitions 1113
212.2. CPT212 Item and Sequence Numbers 1114
213.1. CPT213 Element Output Definitions 1119
213.2. CPT213 Item and Sequence Numbers 1121
214.1. COMBI214 Element Output Definitions 1127
214.2. COMBI214 Item and Sequence Numbers 1127
215.1. CPT215 Output Definitions 1133
215.2. CPT215 Item and Sequence Numbers 1134
216.1. CPT216 Output Definitions 1140
216.2. CPT216 Item and Sequence Numbers 1141
217.1. CPT217 Element Output Definitions 1145
217.2. CPT217 Item and Sequence Numbers 1146
223.1. PLANE223 Field Keys 1154

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

223.2. PLANE223 Coupled-Field Analyses 1154
223.3. PLANE223 Material Properties and Material Models 1156
223.4. PLANE223 Surface and Body Loads 1158
223.5. Automatic Element Technology Selection 1160
223.6. PLANE223 Element Output Definitions 1162
223.7. PLANE223 Item and Sequence Numbers 1166
226.1. SOLID226 Field Keys 1170
226.2. SOLID226 Coupled-Field Analyses 1171
226.3. SOLID226 Material Properties and Material Models 1172
226.4. SOLID226 Surface and Body Loads 1174
226.5. Automatic Element Technology Selection 1176
226.6. SOLID226 Element Output Definitions 1178
226.7. SOLID226 Item and Sequence Numbers 1182
227.1. SOLID227 Field Keys 1186
227.2. SOLID227 Coupled-Field Analyses 1186
227.3. SOLID227 Material Properties and Material Models 1188
227.4. SOLID227 Surface and Body Loads 1190
227.5. Automatic Element Technology Selection 1192
227.6. SOLID227 Element Output Definitions 1194
227.7. SOLID227 Item and Sequence Numbers 1198
230.1. PLANE230 Element Output Definitions 1203
230.2. PLANE230 Item and Sequence Numbers 1204
231.1. SOLID231 Element Output Definitions 1209
231.2. SOLID231 Item and Sequence Numbers 1210
232.1. SOLID232 Element Output Definitions 1215
232.2. SOLID232 Item and Sequence Numbers 1216
233.1. PLANE233 Real Constants 1221
233.2. PLANE233 Element Output Definitions 1222
233.3. PLANE233 Item and Sequence Numbers 1224
233.4. THETA Default Values 1224
236.1. SOLID 236 Real Constants 1232
236.2. SOLID236 Element Output Definitions 1233
236.3. SOLID236 Item and Sequence Numbers 1234
236.4. THETA Default Values 1235
237.1. SOLID 237 Real Constants 1241
237.2. SOLID237 Element Output Definitions 1243
237.3. SOLID237 Item and Sequence Numbers 1244
237.4. THETA Default Values 1245
238.1. PLANE238 Element Output Definitions 1249
238.2. PLANE238 Item and Sequence Numbers 1250
239.1. SOLID239 Element Output Definitions 1253
239.2. SOLID239 Item and Sequence Numbers 1254
240.1. SOLID240 Element Output Definitions 1257
240.2. SOLID240 Item and Sequence Numbers 1257
241.1. HSFLD241 Element Output Definitions 1263
241.2. HSFLD241 Item and Sequence Numbers 1264
242.1. HSFLD242 Element Output Definitions 1271
242.2. HSFLD242 Item and Sequence Numbers 1272
251.1. SURF251 Item and Sequence Numbers 1275
252.1. SURF252 Item and Sequence Numbers 1279
263.1. REINF263 Element Output Definitions 1285
263.2. REINF263 Item and Sequence Numbers 1286

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

264.1. REINF264 Geometry 1289
264.2. REINF264 Element Output Definitions 1293
264.3. REINF264 Item and Sequence Numbers 1294
265.1. REINF265 Element Output Definitions 1302
265.2. REINF265 Item and Sequence Numbers 1304
272.1. SOLID272 Element Output Definitions 1308
272.2. SOLID272 Item and Sequence Numbers 1310
273.1. SOLID273 Element Output Definitions 1317
273.2. SOLID273 Item and Sequence Numbers 1318
278.1. SOLID278 Homogeneous Thermal Solid Output Definitions 1324
278.2. SOLID278 homogeneous Thermal Solid Item and Sequence Numbers 1325
278.3. SOLID278 Layered Thermal Solid Element Output Definitions 1329
278.4. SOLID278 Layered Thermal Solid Item and Sequence Numbers 1330
279.1. SOLID279 Element Output Definitions 1336
279.2. SOLID279 Item and Sequence Numbers 1337
279.3. SOLID279 Element Output Definitions 1342
279.4. SOLID279 Item and Sequence Numbers 1343
281.1. SHELL281 Element Output Definitions 1352
281.2. SHELL281 Item and Sequence Numbers 1355
285.1. SOLID285 Element Output Definitions 1361
285.2. SOLID285 Item and Sequence Numbers 1363
288.1. PIPE288 Element Output Definitions 1375
288.2. PIPE288 Item and Sequence Numbers 1378
289.1. PIPE289 Element Output Definitions 1393
289.2. PIPE289 Item and Sequence Numbers 1395
290.1. ELBOW290 Element Output Definitions 1411
290.2. ELBOW290 Item and Sequence Numbers 1413

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 1: About This Reference

Welcome to the *Element Reference*. The following topics are available to introduce you to ANSYS, Inc. element technology:

1.1. Overview
1.2. Conventions Used in This Reference
1.3. Product Capabilities

1.1. Overview

The purpose this reference is to help you understand, select, and use elements for your analysis.

The reference begins with a general description of the elements, focusing on common features or properties, and including information about how elements, loads, coordinate systems, and output are defined. See General Element Features.

Elements are then introduced by classifications based on dimensions, application or discipline, and elements with special properties and shapes. See Element Classifications.

Subsequent documentation includes specific information about each element class, such as continuum stress, shells, beams, pipes, and others. Basic assumptions, advantages, and limitations of each element class are discussed. Common formulation and element usage techniques are also described.

Finally, a complete library of detailed element descriptions, arranged in order by element number, is presented. The element library is the definitive reference for element documentation. See Part I: Element Library.

This reference is not a primary source of conceptual or procedural information. See the appropriate analysis guides for that information.

1.2. Conventions Used in This Reference

This reference uses the following conventions to help you identify various types of information:

Type style or text	Indicates
UPPERCASE or **UPPERCASE**	Uppercase indicates an element name (such as SOLID273 or ELBOW290). Bold uppercase indicates a command name (such as **K** or **DDELE**).
Bold> Bold	Bold text in mixed case indicates a menu path within the graphical user interface (GUI), a series of menu options for accessing a command from the GUI. One or more angle brackets (>) separate menu items in a menu path.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Type style or text	Indicates
	A command shown in the text is sometimes followed by its GUI equivalent in parentheses, as shown in this example: ***GET** command (**Utility Menu> Parameters> Get Scalar Data**).
ITALICS	Uppercase italic letters (such as *VALUE*, *INC*, *TIME*) generally indicate command arguments for *numeric* values. In some cases (where indicated), non-numeric convenience labels (such as ON, OFF and ALL) can also be specified for these arguments.
Italics	Mixed case italic letters (such as *Lab* or *Fname*) generally indicate command arguments for *alphanumeric* values.
COURIER	A courier font indicates command input or output.

Any mention of a given element implies a reference (link) to the appropriate element documentation, where you can find detailed information about that element.

1.2.1. Applicable Products

This reference applies to the following family of products:

ANSYS Multiphysics (combination of all structural, thermal, electromagnetics, and CFD-Flo capabilities; excludes explicit dynamics)
ANSYS Mechanical (includes all structural and thermal capabilities; excludes electromagnetics, CFD, and explicit dynamics capabilities)
ANSYS Structural (includes all structural linear and nonlinear capabilities)
ANSYS Professional (Nonlinear Thermal and Nonlinear Structural)
ANSYS Emag (Low Frequency)
ANSYS LS-DYNA
ANSYS PrepPost
ANSYS Mechanical/ANSYS Emag (combination of all Mechanical and Emag capabilities)
ANSYS Mechanical/ANSYS CFD-Flo (combination of all Mechanical and CFD-Flo capabilities)

Element Option Availability

Even if an element is available in a particular product, some of its options may not be. Most element descriptions include a Product Restrictions section which details any specific restrictions the element may have in each of the products.

ANSYS Multiphysics

Some element key option (KEYOPT) settings have defaults in the derived products that differ from those in ANSYS Multiphysics. Such cases are clearly documented under the Product Restrictions section of the documentation for the element. If you intend to use your derived product input file in ANSYS Multiphysics, you should explicitly input these settings in the derived product, rather than letting them default; otherwise, behavior in ANSYS Multiphysics will differ.

Connections and High Performance Computing

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

While connection capabilities and High Performance Computing are included as part of the release distribution, they are separately-licensed products. Contact your ASD if you want to install and run any of the separately-licensed products at your site.

1.2.2. Product Codes

A list of product codes appears near the top of the first page of each element description. The codes represent individual products in the ANSYS family of products. The element is valid only for the product codes listed.

An element that is valid within the entire set of products has the following list of product codes:

MP ME ST PR PRN DS EM DY PP EME MFS

The codes represent each of the products in the ANSYS family of products:

Code	Product
MP	ANSYS Multiphysics
ME	ANSYS Mechanical
ST	ANSYS Structural
PR	ANSYS Professional - Nonlinear Thermal
PRN	ANSYS Professional - Nonlinear Structural
DS	ANSYS DesignSpace
EM	ANSYS Emag - Low Frequency
DY	ANSYS LS-DYNA
PP	ANSYS PrepPost
EME	ANSYS Mechanical/ANSYS Emag
MFS	ANSYS Mechanical/ANSYS CFD-Flo

If the code for a given product appears, an element is valid for that product. Exceptions are noted under Product Restrictions in the documentation for that element.

1.3. Product Capabilities

A complete list of engineering capabilities available in the various ANSYS, Inc. products is available on the ANSYS, Inc. Web site at http://www.ansys.com/assets/brochures/capabilities-brochure.pdf.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 2: Element Classifications

A large library of element types is available for use in your analysis. To help you select elements for your analysis, basic characteristics of the element types, and their groupings, are described here.

The following topics related to element types and classifications are available:

2.1. Overview of Element Characteristics
2.2. Summary of Element Types
2.3. Selecting Elements for Your Analysis
2.4. Current-Technology Elements
2.5. Degenerated Shape Elements
2.6. Special-Purpose Elements
2.7. GUI-Inaccessible Elements

Detailed descriptions for each element type are given in Element Library (p. 115).

2.1. Overview of Element Characteristics

The following element overview topics are available:

2.1.1. Identifying Element Types
2.1.2. 2-D vs. 3-D Elements
2.1.3. Element Shape Characteristics
2.1.4. Element Degrees of Freedom and Discipline
2.1.5. User-Defined Elements

2.1.1. Identifying Element Types

An element type is identified by a name (eight characters maximum), such as PIPE288, consisting of a group label (PIPE) and a unique identifying number (288).

The element descriptions in Element Library (p. 115) are arranged in order of these identification numbers. The element is selected from the library for use in the analysis by inputting its name on the element type command (**ET**).

2.1.2. 2-D vs. 3-D Elements

Models may be either 2-D or 3-D depending upon the element types used.

A 2-D model must be defined in an X-Y plane. They are easier to set up, and run faster than equivalent 3-D models. Axisymmetric models are also considered to be 2-D.

If any 3-D element type is included in the element type (**ET**) set, the model becomes 3-D. Some element types (such as COMBIN14) may be 2-D or 3-D, depending upon the KEYOPT value selected. Other element types (such as COMBIN40) have no influence in determining the model dimensions. A 2-D element type may be used (with caution) in 3-D models.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.1.3. Element Shape Characteristics

In general, four shapes are possible: point, line, area, or volume.

- A *point* element is typically defined by one node, such as a mass element.
- A *line* element is typically represented by a line or arc connecting two or three nodes. Examples are beams, spars, pipes, and axisymmetric shells.
- An *area* element has a triangular or quadrilateral shape and may be a 2-D solid element or a shell element.
- A *volume* element has a tetrahedral or brick shape and is usually a 3-D solid element.

2.1.4. Element Degrees of Freedom and Discipline

The degrees of freedom of the element determine the discipline for which the element is applicable: structural, thermal, fluid, electric, magnetic, or coupled-field.

Select an element type with the necessary degrees of freedom characterize the model's response.

Including unnecessary degrees of freedom increases the solution memory requirements and running time. Similarly, selecting element types with unnecessary features--for example, using an element type with plastic capability in an elastic solution--also increases the analysis run time.

2.1.5. User-Defined Elements

You can create your own element type and use it in an analysis as a *user-defined element*.

User-defined elements are described in Creating a New Element in the *Guide to User-Programmable Features*.

2.2. Summary of Element Types

The following tables show elements listed by group label in numerical order, along with a description and graphical representation for each.

BEAM	INFIN	PLANE	SURF
CIRCU	INTER	PRETS	TARGE
COMBIN	LINK	REINF	TRANS
CONTAC	MASS	ROM	
CPT	MATRIX	SHELL	
FLUID	MESH	SOLID	
FOLLW	MPC	SOLSH	
HSFLD	PIPE	SOURC	

BEAM Elements	Pictorials
BEAM161 Explicit 3-D Beam 3 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ, VX, VY, AX, AY, AZ	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BEAM Elements	Pictorials
BEAM188 Structural 3-D 2-Node Beam 2 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ	
BEAM189 Structural 3-D 3-Node Beam 3 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ	

CIRCU Elements	Pictorials
CIRCU94 Coupled-Field Piezoelectric Circuit 2 or 3 nodes 2-D space DOF: VOLT, CURR	
CIRCU124 Magnetic Electric Circuit 2-6 nodes 3-D space DOF: VOLT, CURR, EMF	
CIRCU125 Magnetic Electric Diode 2 nodes 3-D space DOF: VOLT	

COMBIN Elements	Pictorials
COMBIN14 Combination Spring-Damper 2 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ	
COMBIN37 Combination Control 4 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ, PRES, TEMP	
COMBIN39 Combination Nonlinear Spring 2 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ, PRES, TEMP	
COMBIN40 Combination 2 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ, PRES, TEMP	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

COMBIN Elements	Pictorials
COMBI165 Explicit Dynamics Spring-Damper 2 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ, VX, VY, VZ, AX, AV, AZ	
COMBI214 Combination Spring-Damper Bearing 2 nodes 2-D space DOF: UX, UY, UZ,	

CONTAC Elements	Pictorials
CONTA171 2-D 2-Node Surface-to-Surface Contact 2 nodes 2-D space DOF: UX, UY, TEMP, VOLT, AZ	
CONTA172 2-D 3-Node Surface-to-Surface Contact 3 nodes 2-D space DOF: UX, UY, TEMP, VOLT, AZ	
CONTA173 3-D 4-Node Surface-to-Surface Contact 4 nodes 3-D space DOF: UX, UY, UZ, TEMP, VOLT, MAG	
CONTA174 3-D 8-Node Surface-to-Surface Contact 8 nodes 3-D space DOF: UX, UY, UZ, TEMP, VOLT, MAG	
CONTA175 2-D/3-D Node-to-Surface Contact 1 node 2-D/3-D space DOF: UX, UY, UZ, TEMP, VOLT, AX, MAG	
CONTA176 3-D Line-to-Line Contact 3 nodes 3-D space DOF: UX, UY, UZ	
CONTA177 3-D Line-to-Surface Contact 3 nodes 3-D space DOF: UX, UY, UZ	
CONTA178 3-D Node-to-Node Contact 2 nodes 3-D space DOF: UX, UY, UZ, TEMP, VOLT	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CPT Elements	Pictorials
CPT212 2-D 4-Node Coupled Pore-Pressure Mechanical Solid 4 nodes 2-D space DOF: UX, UY, PRES	
CPT213 2-D 8-Node Coupled Pore-Pressure Mechanical Solid 8 nodes 2-D space DOF: UX, UY, PRES	
CPT215 3-D 8-Node Coupled Pore-Pressure Mechanical Solid 8 nodes 3-D space DOF: UX, UY, UZ, PRES	
CPT216 3-D 20-Node Coupled Pore-Pressure Mechanical Solid 20 nodes 3-D space DOF: UX, UY, UZ, PRES	
CPT217 3-D 10-Node Coupled Pore-Pressure Mechanical Solid 10 nodes 3-D space DOF: UX, UY, UZ, PRES	

FLUID Elements	Pictorials
FLUID29 2-D Acoustic Fluid 4 nodes 2-D space DOF: UX, UY, PRES	
FLUID30 3-D Acoustic Fluid 8 nodes 3-D space DOF: UX, UY, UZ, PRES	
FLUID38 Dynamic Fluid Coupling 2 nodes 3-D space DOF: UX, UY, UZ	
FLUID116 Coupled Thermal-Fluid Pipe 2 nodes 3-D space DOF: PRES, TEMP	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FLUID Elements	Pictorials
FLUID129 2-D Infinite Acoustic 2 nodes 2-D space DOF: PRES	
FLUID130 3-D Infinite Acoustic 4 nodes 3-D space DOF: PRES	
FLUID136 3-D Squeeze Film Fluid 4, 8 nodes 3-D space DOF: PRES	
FLUID138 3-D Viscous Fluid Link 2 nodes 3-D space DOF: PRES	
FLUID139 3-D Slide Film Fluid 2, 32 nodes 3-D space DOF: UX, UY, UZ	
FLUID220 3-D Acoustic Fluid 20 nodes 3-D space DOF: UX, UY, UZ, PRES	
FLUID221 3-D Acoustic Fluid 10 nodes 3-D space DOF: UX, UY, UZ, PRES	

FOLLW Elements	Pictorials
FOLLW201 3-D Follower Load 1 node 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ	

HSFLD Elements	Pictorials
HSFLD241 2-D Hydrostatic Fluid 4 nodes 2-D space DOF: UX, UY, HDSP, PRES	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HSFLD Elements	Pictorials
HSFLD242 3-D Hydrostatic Fluid 9 nodes 3-D space DOF: UX, UY, UZ, HDSP, PRES	

INFIN Elements	Pictorials
INFIN9 2-D Infinite Boundary 2 nodes 2-D space DOF: AZ, TEMP	
INFIN47 3-D Infinite Boundary 4 nodes 3-D space DOF: MAG, TEMP	
INFIN110 2-D Infinite Solid 4 or 8 nodes 2-D space DOF: AZ, VOLT, TEMP	
INFIN111 3-D Infinite Solid 8 or 20 nodes 3-D space DOF: MAG, AX, AY, AZ, VOLT, TEMP	

INTER Elements	Pictorials
INTER192 Structural 2-D Interface 4-Node Gasket 4 nodes 2-D space DOF: UX, UY	
INTER193 Structural 2-D Interface 6-Node Gasket 6 nodes 2-D space DOF: UX, UY	
INTER194 Structural 3-D Interface 16-Node Gasket 16 nodes 3-D space DOF: UX, UY, UZ	
INTER195 Structural 3-D Interface 8-Node Gasket 8 nodes 3-D space DOF: UX, UY, UZ	
INTER202 Structural 2-D Interface 4-Node Cohes- ive 4 nodes 2-D space	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

INTER Elements	Pictorials
DOF: UX, UY	
INTER203 Structural 2-D Interface 6-Node Cohesive 6 nodes 2-D space DOF: UX, UY	
INTER204 Structural 3-D Interface 16-Node Cohesive 16 nodes 3-D space DOF: UX, UY, UZ	
INTER205 Structural 3-D Interface 8-Node Cohesive 8 nodes 3-D space DOF: UX, UY, UZ	

LINK Elements	Pictorials
LINK11 Structural 3-D Linear Actuator 2 nodes 3-D space DOF: UX, UY, UZ	
LINK31 Radiation Link 2 nodes 3-D space DOF: TEMP	
LINK33 Thermal 3-D Conduction Bar 2 nodes 3-D space DOF: TEMP	
LINK34 Convection Link 2 nodes 3-D space DOF: TEMP	
LINK68 Coupled Thermal-Electric Line 2 nodes 3-D space DOF: TEMP, VOLT	
LINK160 Explicit 3-D Spar (or Truss) 3 nodes 3-D space DOF: UX, UY, UZ, VX, VY, VZ, AX, AY, AZ	
LINK167 Explicit Tension-Only Spar 3 nodes 3-D space	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LINK Elements	Pictorials
DOF: UX, UY, UZ, VX, VY, VZ, AX, AY, AZ	
LINK180 Structural 3-D Spar (or Truss) 2 nodes 3-D space DOF: UX, UY, UZ	

MASS Elements	Pictorials
MASS21 Structural Mass 1 node 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ	
MASS71 Thermal Mass 1 node 3-D space DOF: TEMP	
MASS166 Explicit 3-D Structural Mass 1 node 3-D space DOF: UX, UY, UZ, VX, VY, VZ, AX, AY, AZ	

MATRIX Elements	Pictorials
MATRIX27 Stiffness, Damping, or Mass Matrix 2 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ	
MATRIX50 Superelement (or Substructure) 2-D or 3-D space DOF: Determined from included element types	

MESH Elements	Pictorials
MESH200 Meshing Facet 2-20 nodes 2-D/3-D space DOF: None KEYOPT Dependent	

MPC Elements	Pictorials
MPC184 Structural Multipoint Constraint 2 or 3 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ KEYOPT Dependent	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PIPE Elements	Pictorials
PIPE288 3-D 2-Node Pipe 2 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ	
PIPE289 3-D 3-Node Pipe 3 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ	
ELBOW290 3-D 3-Node Elbow 3 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ	

PLANE Elements	Pictorials
PLANE13 2-D Coupled-Field Solid 4 nodes 2-D space DOF: TEMP, AZ, UX, UY, VOLT	
PLANE25 Axisymmetric-Harmonic 4-Node Structural Solid 4 nodes 2-D space DOF: UX, UY, UZ	
PLANE35 2-D 6-Node Triangular Thermal Solid 6 nodes 2-D space DOF: TEMP	
PLANE53 2-D 8-Node Magnetic Solid 8 nodes 2-D space DOF: VOLT, AZ, CURR, EMF	
PLANE55 2-D 4-Node Thermal Solid 4 nodes 2-D space DOF: TEMP	
PLANE75 2-D Axisymmetric-Harmonic 4-Node Thermal Solid 4 nodes 2-D space DOF: TEMP	
PLANE77 2-D 8-Node Thermal Solid 8 nodes 2-D space DOF: TEMP	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PLANE Elements	Pictorials
PLANE78 2-D Axisymmetric-Harmonic 8-Node Thermal Solid 8 nodes 2-D space DOF: TEMP	
PLANE83 2-D Axisymmetric - Harmonic 8-Node Structural Solid 8 nodes 2-D space DOF: UX, UY, UZ	
PLANE121 2-D 8-Node Electrostatic Solid 8 nodes 2-D space DOF: VOLT	
PLANE162 Explicit 2-D Structural Solid 4 nodes 2-D space DOF: UX, UY, VX, VY, AX, AY	
PLANE182 2-D 4-Node Structural Solid 4 nodes 2-D space DOF: UX, UY	
PLANE183 2-D 8-Node Structural Solid 8 nodes 2-D space DOF: UX, UY	
PLANE223 2-D 8-Node Coupled-Field Solid 8 nodes 2-D space DOF: UX, UY, TEMP, VOLT	
PLANE230 2-D 8-Node Electric Solid 8 nodes 2-D space DOF: VOLT	
PLANE233 2-D 8-Node Electromagnetic Solid 8 nodes 2-D space DOF: AZ, VOLT	
PLANE238 2-D 8-Node Diffusion Solid 8 nodes 2-D space DOF: CONC	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PRETS Elements	Pictorials
PRETS179 2-D/3-D Pretension Combination 3 nodes 2-D/3-D space DOF: UX	

REINF Elements	Pictorials
REINF263 2-D Smeared Reinforcing Up to 8 nodes 2-D space DOF: UX, UY, UZ, ROTZ	
REINF264 3-D Discrete Reinforcing Up to 20 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ	
REINF265 3-D Smeared Reinforcing Up to 20 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ	

ROM Elements	Pictorials
ROM144 Reduced-Order Electrostatic-Structural Coupled-Field 20 or 30 nodes 3-D space DOF: EMF, VOLT, UX	

SHELL Elements	Pictorials
SHELL28 Structural 3-D Shear/Twist Panel 4 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ	
SHELL41 Structural 3-D Membrane Shell 4 nodes 3-D space DOF: UX, UY, UZ	
SHELL61 2-D Axisymmetric-Harmonic Structural Shell 2 nodes 2-D space DOF: UX, UY, UZ, ROTZ	
SHELL131 4-Node Thermal Shell 4 nodes 3-D space DOF: TBOT, TE2, TE3, TE4, . . . TTOP	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SHELL Elements	Pictorials
SHELL132 8-Node Thermal Shell 8 nodes 3-D space DOF: TBOT, TE2, TE3, TE4, . . . TTOP	
SHELL157 Thermal-Electric Shell 4 nodes 3-D space DOF: TEMP, VOLT	
SHELL163 Explicit Thin Structural Shell 4 nodes 3-D space DOF: UX, UY, UZ, VX, VY, VZ, AX, AY, AZ, ROTX, ROTY, ROTZ,	
SHELL181 4-Node Structural Shell 4 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ	
SHELL208 2-Node Axisymmetric Shell 2 nodes 2-D space DOF: UX, UY, ROTZ	
SHELL209 3-Node Axisymmetric Shell 3 nodes 2-D space DOF: UX, UY, ROTZ	
SHELL281 8-Node Structural Shell 8 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ	

SOLID Elements	Pictorials
SOLID5 3-D Coupled-Field Solid 8 nodes 3-D space DOF: UX, UY, UZ, TEMP, VOLT, MAG	
SOLID65 3-D Reinforced Concrete Structural Solid 8 nodes 3-D space DOF: UX, UY, UZ	
SOLID70 3-D Thermal Solid 8 nodes 3-D space DOF: TEMP	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID Elements	Pictorials
SOLID87 3-D 10-Node Tetrahedral Thermal Solid 10 nodes 3-D space DOF: TEMP	
SOLID90 3-D 20-Node Thermal Solid 20 nodes 3-D space DOF: TEMP	
SOLID96 3-D Magnetic Scalar Solid 8 nodes 3-D space DOF: MAG	
SOLID97 3-D Magnetic Solid 8 nodes 3-D space DOF: AX, AY, AZ, VOLT, CURR, EMF	
SOLID98 Tetrahedral Coupled-Field Solid 10 nodes 3-D space DOF: UX, UY, UZ, TEMP, VOLT, MAG	
SOLID122 3-D 20-Node Electrostatic Solid 20 nodes 3-D space DOF: VOLT	
SOLID123 3-D 10-Node Tetrahedral Electrostatic Solid 10 nodes 3-D space DOF: VOLT	
SOLID164 Explicit 3-D Structural Solid 8 nodes 3-D space DOF: UX, UY, UZ, VX, VY, VZ, AX, AY, AZ	
SOLID168 Explicit 3-D 10-Node Tetrahedral Structural Solid 10 nodes 3-D space DOF: UX, UY, UZ, VX, VY, VZ, AX, AY, AZ	
SOLID185 3-D 8-Node Structural Solid 8 nodes 3-D space DOF: UX, UY, UZ	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID Elements	Pictorials
SOLID186 3-D 20-Node Structural Solid 20 nodes 3-D space DOF: UX, UY, UZ	
SOLID187 3-D 10-Node Tetrahedral Structural Solid 10 nodes 3-D space DOF: UX, UY, UZ	
SOLID226 3-D 20-Node Coupled-Field Solid 20 nodes 3-D space DOF: UX, UY, UZ, TEMP, VOLT	
SOLID227 3-D 10-Node Coupled-Field Solid 10 nodes 3-D space DOF: UX, UY, UZ, TEMP, VOLT	
SOLID231 3-D 20-Node Electric Solid 20 nodes 3-D space DOF: VOLT	
SOLID232 3-D 10-Node Tetrahedral Electric Solid 10 nodes 3-D space DOF: VOLT	
SOLID236 3-D 20-Node Electromagnetic Solid 20 nodes 3-D space DOF: AZ, VOLT	
SOLID237 3-D 10-Node Electromagnetic Solid 10 nodes 3-D space DOF: AZ, VOLT	
SOLID239 3-D 20-Node Diffusion Solid 10 nodes 3-D space DOF: CONC	
SOLID240 3-D 10-Node Tetrahedral Diffusion Solid 10 nodes 3-D space DOF: CONC	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID Elements	Pictorials
SOLID272 4- to 48-Node General Axisymmetric Solid (KEYOPT(2) = 3) DOF: UX, UY, UZ	
SOLID273 8- to 96-Node General Axisymmetric Solid (KEYOPT(2) = 3) DOF: UX, UY, UZ	
SOLID278 3-D 8-Node Thermal Solid 8 nodes 3-D space DOF: TEMP	
SOLID279 3-D 20-Node Thermal Solid 20 nodes 3-D space DOF: TEMP	
SOLID285 3-D 4-Node Tetrahedral Structural Solid with Nodal Pressure 4 nodes 3-D space DOF: UX, UY, UZ, HDSP	

SOLSH Elements	Pictorials
SOLSH190 Structural Solid Shell 8 nodes 3-D space DOF: UX, UY, UZ	

SOURC Elements	Pictorials
SOURC36 Magnetic Electric Current Source 3 nodes 3-D space DOF: None	

SURF Elements	Pictorials
SURF151 2-D Thermal Surface Effect 2 or 4 nodes 2-D space DOF: TEMP	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SURF Elements	Pictorials
SURF152 3-D Thermal Surface Effect 4 to 10 nodes 3-D space DOF: TEMP	
SURF153 2-D Structural Surface Effect 2 or 3 nodes 2-D space DOF: UX, UY	
SURF154 3-D Structural Surface Effect 4 to 8 nodes 3-D space DOF: UX, UY, UZ	
SURF156 3-D Structural Surface Line Load 3 to 4 nodes 3-D space DOF: UX, UY, UZ	
SURF159 3-D General Axisymmetric Surface 2 to 36 nodes 3-D space DOF: UX, UY, UZ	
SURF251 2-D Radiosity Surface 2 nodes 2-D space	
SURF252 3-D Thermal Radiosity Surface 3 or 4 nodes, 3-D space	

TARGE Elements	Pictorials
TARGE169 Contact 2-D Target Segment 3 nodes 2-D space DOF: UX, UY, ROTZ, TEMP	
TARGE170 Contact 3-D Target Segment 8 nodes 3-D space DOF: UX, UY, UZ, TEMP	

TRANS Elements	Pictorials
TRANS126 Electromechanical Transducer 2 nodes 3-D space DOF: UX-VOLT, UY-VOLT, UZ-VOLT	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.3. Selecting Elements for Your Analysis

Use this list of elements sorted by application to select the best elements for your analysis. Primary applications are as follows:

Structural (p. 22)
Explicit Dynamics (p. 26)
Thermal (p. 27)
Acoustic (p. 29)
Diffusion (p. 29)
Fluid (p. 30)
Magnetic Electric (p. 30)
Electric Circuit (p. 32)
Coupled-Field (p. 32)
Contact (p. 33)
Combination (p. 34)
Matrix (p. 35)
Infinite (p. 35)
Load (p. 35)
Meshing (p. 36)
Reinforcing (p. 36)
User-Defined (p. 37)

Within each element category, the elements appear in order of shape function (topology). Current-technology elements take precedence.

Structural			
Type	**Dimen-sion**	**Element**	**Pictorial**
Structural Solid	3-D	SOLID185 8-Node Structural Solid	
		SOLID186 20-Node Structural Solid	
		SOLID187 10-Node Tetrahedral Structural Solid	
		SOLID285 4-Node Tetrahedral Structural Solid with Nodal Pressures	
		SOLID65 8-Node Reinforced Concrete Structural Solid	
	2-D	PLANE182 4-Node Structural Solid	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

		PLANE183 8-Node Structural Solid	
		PLANE25 4-Node Axisymmetric-Harmonic Structural Solid	
		PLANE83 8-Node Axisymmetric-Harmonic Structural Solid	
	General Axisymmetric	SOLID272 4- to 48-Node General Axisymmetric Solid	
		SOLID273 8- to 96-Node General Axisymmetric Solid	
Structural Solid Shell	3-D	SOLSH190 8-Node Structural Solid Shell	
Structural Shell	3-D	SHELL181 4-Node Structural Shell	
		SHELL281 8-Node Structural Shell	
		SHELL28 4-Node Structural Shear/Twist Panel	
		SHELL41 4-Node Structural Membrane Shell	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	2-D	SHELL208 2-Node Axisymmetric Shell	
		SHELL209 3-Node Axisymmetric Shell	
		SHELL61 2-Node Axisymmetric-Harmonic Structural Shell	
Structural Layered Composite	3-D	SOLID185 8-Node Layered Solid	
		SOLID186 20-Node Layered Structural Solid	
		SOLSH190 8-Node Solid Shell	
	The following elements can also be used for layered applications: SHELL181, SHELL208, SHELL209, SHELL281, ELBOW290		
Structural Beam	3-D	BEAM188 2-Node Structural Beam	
		BEAM189 3-Node Structural Beam	
Structural Pipe	3-D	PIPE288 2-Node Pipe	
		PIPE289 3-Node Pipe	
		ELBOW290 3-Node Elbow	
Structural Line	3-D	LINK180 2-Node Structural 3-D Spar (or Truss)	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

		LINK11 2-Node Structural Linear Actuator	
Structural Point	3-D	MASS21 1-Node Structural Mass	
Structural Interface	3-D	INTER194 Interface 16-Node Gasket	
		INTER195 Interface 8-Node Gasket	
		INTER204 Interface 16-Node Cohesive	
		INTER205 Interface 8-Node Cohesive	
	2-D	INTER192 Interface 4-Node Gasket	
		INTER193 Interface 6-Node Gasket	
		INTER202 Interface 4-Node Cohesive	
		INTER203 Interface 6-Node Cohesive	
Structural Multipoint Constraint	3-D	MPC184	
		MPC184-Link/Beam	
		MPC184-Slider	
		MPC184-Revolute	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

		MPC184-Universal	
		MPC184-Slot	
		MPC184-Point	
		MPC184Translational	
		MPC184Cylindrical	
		MPC184-Planar	
		MPC184-Weld	
		MPC184-Orient	
		MPC184-Spherical	
		MPC184-General	
		MPC184-Screw	
Explicit Dynamics (Explicit dynamics analyses can use only explicit dynamic elements.)			
Type	**Dimen-sion**	**Element**	**Pictorial**
Explicit Solid	3-D	SOLID164 8-Node Explicit Structural Solid	
		SOLID168 10-Node Explicit Tetrahedral Structural Solid	
	2-D	PLANE162 4-Node Explicit Solid	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Explicit Shell	3-D	SHELL163 4-Node Explicit Thin Structural Shell	
Explicit Beam	3-D	BEAM161 3-Node Explicit Beam	
Explicit Line	3-D	LINK160 3-Node Explicit Spar (or Truss)	
		LINK167 3-Node Explicit Tension-Only Spar	
		COMBI165 2-Node Explicit Spring-Damper	
Explicit Point	3-D	MASS166 1-Node Explicit Structural Mass	
Thermal			
Type	**Dimen-sion**	**Element**	**Pictorial**
Thermal Solid	3-D	SOLID278 8-Node Thermal Solid	
		SOLID279 20-Node Thermal Solid	
		SOLID70 8-Node Thermal Solid	
		SOLID87 10-Node Tetrahed-ral Thermal Solid	
		SOLID90 20-Node Thermal Solid	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	2-D	PLANE35 6-Node Triangular Thermal Solid	
		PLANE55 4-Node Thermal Solid	
		PLANE75 4-Node Axisymmetric-Harmonic Thermal Solid	
		PLANE77 8-Node Thermal Solid	
		PLANE78 4-Node Axisymmetric-Harmonic Thermal Solid	
Thermal Layered Solid	3-D	SOLID278 8-Node Thermal Solid	
		SOLID279 20-Node Thermal Solid	
Thermal Shell	3-D	SHELL131 4-Node Thermal Shell	
		SHELL132 8-Node Thermal Shell	
Thermal Line	3-D	LINK31 2-Node Radiation Link	
		LINK33 2-Node Thermal Conduction Bar	
		LINK34 2-Node Convection Link	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Thermal Point	3-D	MASS71 1-Node Thermal Mass	
Thermal Electric	3-D	LINK68 2-Node Coupled Thermal-Electric Line	
		SHELL157 4-Node Thermal-Electric Shell	
Acoustic			
Type	**Dimen-sion**	**Element**	**Pictorial**
Acoustic Volume	3-D	FLUID30 8-Node Acoustic Fluid	
		FLUID220 20-Node Acoustic Fluid	
		FLUID221 10-Node Acoustic Fluid	
	2-D	FLUID29 4-Node Acoustic Fluid	
Acoustic Planar	3-D	FLUID130 4-Node Infinite Acoustic	
Acoustic Line	2-D	FLUID129 2-Node Infinite Acoustic	
Diffusion			
Type	**Dimen-sion**	**Element**	**Pictorial**
Diffusion Solid	2-D	PLANE238 8-Node Diffusion Solid	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	3-D	SOLID239 20-Node Diffusion Solid	
		SOLID240 10-Node Tetrahedral Diffusion Solid	
Fluid			
Type	**Dimen-sion**	**Element**	**Pictorial**
Fluid Volume	3-D	HSFLD242 9-Node Hydrostatic Fluid	
	2-D	HSFLD241 4-Node Hydrostatic Fluid	
Fluid - Gap	3-D	FLUID38 2-Node Dynamic Fluid Coupling	
		FLUID136 4- or 8-Node Squeeze Film Fluid	
		FLUID138 2-Node Viscous Fluid Link	
		FLUID139 2- or 32-Node Slide Film Fluid	
Fluid Pipe	3-D	FLUID116 2-Node Coupled Thermal-Fluid Pipe	
Magnetic Electric			
Type	**Dimen-sion**	**Element**	**Pictorial**
---	3-D	SOLID96 8-Node Magnetic Scalar Solid	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

		SOLID97 8-Node Magnetic Solid	
		SOLID122 20-Node Electrostatic Solid	
		SOLID123 10-Node Tetrahedral Electrostatic Solid	
		SOLID231 20-Node Electric Solid	
		SOLID232 10-Node Tetrahedral Electric Solid	
		SOLID236 20-Node Electromagnetic Solid	
		SOLID237 10-Node Electromagnetic Solid	
	2-D	PLANE53 8-Node Magnetic Solid	
		PLANE121 8-Node Electrostatic Solid	
		PLANE230 8-Node Electric Solid	
		PLANE233 8-Node Electromagnetic Solid	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Electric Circuit			
Type	**Dimen-sion**	**Element**	**Pictorial**
---	3-D	SOURC36 3-Node Magnetic Electric Current Source	
		CIRCU124 2- to 6-Node Magnetic Electric Circuit	
		CIRCU125 2-Node Magnetic Electric Diode	
	2-D	CIRCU94 2- or 3-Node Coupled-Field Piezoelectric Circuit	

Coupled-Field			
Type	**Dimen-sion**	**Element**	**Pictorial**
---	3-D	SOLID226 20-Node Coupled-Field Solid	
		SOLID227 10-Node Coupled-Field Solid	
		SOLID5 8-Node Coupled-Field Solid	
		LINK68 2-Node-Coupled Thermal-Electric Line	
		SOLID98 10-Node Tetrahed-ral Coupled-Field Solid	
		ROM144 20- or 30-Node Re-duced-Order Electrostatic-Structural Coupled-Field	
		SHELL157 4-Node Thermal-Electric Shell	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

		CPT215 8-Node Coupled Pore-Pressure Mechanical Solid	
		CPT216 20-Node Coupled Pore-Pressure Mechanical Solid	
		CPT217 10-Node Coupled Pore-Pressure Mechanical Solid	
	2-D	PLANE223 8-Node Coupled-Field Solid	
		PLANE13 4-Node Coupled-Field Solid	
		CPT212 4-Node Coupled Pore-Pressure Mechanical Solid	
		CPT213 8-Node Coupled Pore-Pressure Mechanical Solid	
Contact			
Type	**Dimen-sion**	**Element**	**Pictorial**
---	3-D	TARGE170 8-Node Contact 3-D Target Segment	
		CONTA173 4-Node Surface-to-Surface Contact	
		CONTA174 8-Node Surface-to-Surface Contact	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

		CONTA176 3-Node Line-to-Line Contact	
		CONTA177 3-Node Line-to-Surface Contact	
		CONTA178 2-Node Node-to-Node Contact	
	2-D/3-D	CONTA175 1-Node Node-to-Surface Contact	
	2-D	TARGE169 3-Node Contact 2-D Target Segment	
		CONTA171 2-Node Surface-to-Surface Contact	
		CONTA172 3-Node Surface-to-Surface Contact	
Combination			
Type	**Dimen-sion**	**Element**	**Pictorial**
---	3-D	COMBIN14 2-Node Combin-ation Spring-Damper	
		COMBIN37 4-Node Combin-ation Control	
		COMBIN39 2-Node Combin-ation Nonlinear Spring	
		COMBIN40 2-Node Combin-ation	
	2-D/3-D	PRETS179 3-Node Pretension Combination	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	2-D	COMBI214 2-Node Combination Spring-Damper Bearing	
Matrix			
Type	**Dimension**	**Element**	**Pictorial**
---	3-D	MATRIX27 2-Node Stiffness, Damping, or Mass Matrix	
	2-D/3-D	MATRIX50 Superelement (or Substructure)	
Infinite			
Type	**Dimension**	**Element**	**Pictorial**
---	3-D	INFIN47 4-Node Infinite Boundary	
		INFIN111 8- or 20-Node Infinite Solid	
	2-D	INFIN9 2-Node Infinite Boundary	
		INFIN110 4- or 8-Node Infinite Solid	
Load			
Type	**Dimension**	**Element**	**Pictorial**
---	3-D	SURF152 4- to 10-Node Thermal Surface Effect	
		SURF154 4- to 8-Node Structural Surface Effect	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

		SURF156 3- or 4-Node Structural Surface Line Load	
		SURF159 2- to 36-Node General Axisymmetric Surface	
		FOLLW201 1-Node Follower Load	
		SURF252 3- or 4-Node Thermal Radiosity Surface	
	2-D	SURF151 2- or 4-Node Thermal Surface Effect	
		SURF153 2- or 3-Node Structural Surface Effect	
		SURF251 2-Node Radiosity Surface	
Meshing			
Type	**Dimension**	**Element**	**Pictorial**
---	2-D/3-D	MESH200 2- to 20-Node Meshing Facet	
Reinforcing			
Type	**Dimension**	**Element**	**Pictorial**
---	3-D	REINF264 2- to 20-Node Discrete Reinforcing	
		REINF265 2- to 20-Node Smeared Reinforcing	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	2-D	REINF263 2- to 8-Node Smeared Reinforcing	
User-Defined			
Type	**Dimen-sion**	**Element**	**Pictorial**
---	---	USER300	---

2.4. Current-Technology Elements

As technology advances, ANSYS, Inc. continues to develop robust new element types using the most current technologies available. Current-technology elements are more advanced and feature-rich than legacy elements. For example, support for the following capabilities is available only in applicable current-technology elements:

- A vast array of material constitutive options (such as anisotropic hyperelasticity, cast iron plasticity, enhanced Drucker-Prager plasticity, Hill plasticity, hyperelasticity, shape memory alloy, plasticity, rate-dependent plasticity, viscoelasticity, and others).
- Association of a single element with several material constitutive options (such as a combination of hyper-elasticity and viscoelasticity with Prony series expansion).
- A curve-fitting tool (**TBFT**) for calibrating material parameters through experimental data, which currently supports creep, hyperelasticity, and viscoelasticity.
- Fracture mechanics parameter calculation (**CINT**), which uses the domain-integration approach to calculate the J-Integrals for both linear and elastoplastic material behavior at a designated tip location (2-D) or at specific location along the crack front (3-D) through a structure component.
- Initial state conditions (**INISTATE**) that may exist in the structure.
- Custom user-defined material models created via the `UserMat` subroutine.
- 3-D smeared reinforcing, provided by elements such as REINF265 when used with 3-D solid and shell elements (referred to as the base elements) to achieve the effect of extra reinforcement to those elements.
- Nonlinear stabilization, a viscous-based algorithm for improving convergence behavior when instabilities are expected.
- Rezoning for solid elements.
- Linear perturbation based on nonlinear analysis.
- Precise control of the element technologies (**ETCONTROL**) used in element formulation for applicable elements.
- A layered-section option (via **SECDATA** and other section commands) for shell and solid elements.

In Selecting Elements for Your Analysis (p. 22), current-technology elements appear first. You can readily identify them, as they are not typically associated with specific material types, nor do they specify restrictions such as “linear.” Most of the elements listed are current-technology elements in the sense that they are the best technologies that ANSYS, Inc. is able to offer at the current product release.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A few legacy elements are still supported to meet the needs of longtime users who have input files containing those elements. Legacy elements may eventually be replaced by new elements.

2.4.1. Legacy vs. Current Element Technologies

It is good practice to use current-technology elements rather than legacy elements in your analysis wherever possible.

The following table lists the remaining legacy elements and suggests current-technology elements to use instead. In some cases, an element KEYOPT setting, degree-of-freedom constraint, or command may be necessary to more precisely reproduce the behavior of a given legacy element.

Legacy Element Type [1]	Suggested Current Element Type [2]	Suggested Setting(s) to Approximate Legacy Element Behavior [3]	
		Element KEYOPT	Comments
INFIN9	INFIN110	---	---
INFIN47	INFIN111	---	---
PLANE13	PLANE223	---	---
PLANE25	SOLID272	KEYOPT(6) = 0	
PLANE53	PLANE233	---	See the *Low-Frequency Electromagnetic Analysis Guide*.
PLANE83	SOLID273	KEYOPT(6) = 0	---
SHELL41	SHELL181	KEYOPT(1) = 1, KEYOPT(3) = 2	---
SOLID5	SOLID226	---	---
SOLID65	SOLID185	KEYOPT(3) = 0 and KEYOPT(2) = 0 -- Full integration with the $\bar{B}$ method -------- KEYOPT(3) = 0 and KEYOPT(2) = 3 -- simplified enhanced strain formulation	See Microplane in the *Material Reference*.

1. After considering element redundancy and consistency issues, ANSYS, Inc. may in future releases move legacy element documentation to the *Feature Archive* or undocument those elements altogether; however, the table does not imply that all legacy elements listed are immediate targets for such action.

2. The table is not a definitive listing of legacy-to-current element equivalents in terms of either formulation or use of shape functions; for example, a suggested current element may require a more refined mesh in some cases, or may require adaptation via appropriate constraints for specific 2-D analyses.

3. While a given KEYOPT setting can allow you to approximate the behavior of a *legacy* element, it may not be the most desirable KEYOPT for the *current* element. For structural-only analyses, try the **ETCONTROL**

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

command for element and KEYOPT recommendations. For more information, see Automatic Selection of Element Technologies and Formulations (p. 92).

2.5. Degenerated Shape Elements

A degenerated element is an element whose characteristic face shape is quadrilateral, but is modeled with at least one triangular face.

Degenerated elements are often used for modeling transition regions between fine and coarse meshes, or for modeling irregular and warped surfaces.

The following topics concerning degenerated shape elements are available:

2.5.1. Obtaining the Best Results from Degenerated Shape Elements
2.5.2. Degenerated Triangular 2-D Solid and Shell Elements
2.5.3. Degenerated Triangular Prism Elements
2.5.4. Degenerated Tetrahedral Elements
2.5.5. Surface Stresses on Condensed Element Faces

2.5.1. Obtaining the Best Results from Degenerated Shape Elements

Use degenerated shape elements with caution.

Degenerated elements formed from quadrilateral and brick elements *without midside nodes* are much less accurate than those formed from elements with midside nodes and should not be used in high-stress gradient regions.

When using triangular elements in a rectangular array of nodes, best results are obtained from an element pattern having alternating diagonal directions.

For shell elements, because the element coordinate system is relative to the I-J line, the stress results are most easily interpreted if the I-J lines of the elements are all parallel.

2.5.2. Degenerated Triangular 2-D Solid and Shell Elements

Degenerated triangular 2-D solid and shell elements may be formed from 4-node quadrilateral elements by defining duplicate node numbers for the third and fourth (K and L) node locations.

The node pattern then becomes I, J, K, K. If the L node is not input, it defaults to node K. If extra shape functions are included in the element, they are automatically suppressed (degenerating the element to a lower order). Element loads specified on a nodal basis should have the same loads specified at the duplicate node locations.

When forming a degenerated triangular element by repeating node numbers, the face numbering remains the same. Face 3, however, condenses to a point. The centroid location printed for a degenerated triangular element is usually at the geometric centroid of the element.

Elements should be oriented with alternating diagonals, if possible.

2.5.2.1. Triangular Shell Elements Are Preferred in Some Cases

Triangular shell elements are *preferred* for severely skewed or warped elements.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Quadrilateral shaped elements should not be skewed such that the included angle between two adjacent faces is outside the range of 90° ± 45° for non-midside-node elements or 90° ± 60° for midside-node elements.

Warping occurs when the four nodes of a quadrilateral shell element (or solid element face) are not in the same plane, either at input or during large deflection. Warping is measured by the relative angle between the normals to the face at the nodes. A flat face (no warping) has all normals parallel (zero relative angle). A warning message is output if warping is beyond a small, but tolerable value. If warping is excessive, the analysis terminates.

See the *Mechanical APDL Theory Reference* for element warping details and other element checking details. Triangular (or prism) elements should be used in place of a quadrilateral (or brick) element with large warping.

2.5.3. Degenerated Triangular Prism Elements

Degenerated triangular prism elements may be formed from 8-node 3-D solid elements by defining duplicate node numbers for the third and fourth (K and L) and the seventh and eighth (O and P) node locations.

The node pattern then becomes I, J, K, K, M, N, O, O. When forming a degenerated prism element by repeating node numbers, the face numbering remains the same. Face 4, however, condenses to a line.

The centroid location printed for a degenerated element is not at the geometric centroid but is at an average nodal location. The integration points are proportionately rearranged within the element.

Elements should be oriented with alternating diagonals, if possible. If extra shape functions are included in the element, they are partially suppressed.

Element loads should have the same loads specified at the duplicate node locations.

2.5.4. Degenerated Tetrahedral Elements

A degenerated tetrahedral element may be formed from a triangular prism element by a further condensation of face 6 to a point.

The input node pattern should be I, J, K, K, M, M, M, M. If extra shape functions are included in the element, they are automatically suppressed. Element nodal loads should have the same loads specified at the duplicate node locations.

2.5.5. Surface Stresses on Condensed Element Faces

Surface stress (or convection heat flow) printout should not be requested on a condensed face.

Pressures (or convection conditions) should not be defined on a condensed face.

2.6. Special-Purpose Elements

Two special element types are available:

2.6.1. Superelements
2.6.2. User-Defined Elements

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.6.1. Superelements

You can combine existing linear elements into a superelement (or substructure) named MATRIX50. This capability is especially useful when you are repeating the same action many times in the model, or using the same group of elements over a large number of load steps. In such cases, using superelements typically results in greater solution efficiency.

For further information, see:

- Substructuring in the *Advanced Analysis Guide* to learn how to perform an analysis using superelements
- Substructuring Analysis in the *Mechanical APDL Theory Reference* for a discussion of the substructure matrix procedure

2.6.2. User-Defined Elements

You can create a custom element, of virtually any type, named USER300.

For more information, see the documentation for the USER300 element. For detailed instructions, see Creating a New Element via the User-Defined Element API in the *Guide to User-Programmable Features*, available on the product-distribution media.

2.7. GUI-Inaccessible Elements

These elements are available via the **ET** family of commands only and are inaccessible from within the GUI:

REINF263	2-D Smeared Reinforcing
REINF264	3-D Discrete Reinforcing
REINF265	3-D Smeared Reinforcing
SURF251	2-D Radiosity Surface
SURF252	3-D Radiosity Surface

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 3: General Element Features

The element library offers many element formulations or *types*. Many features are common to *all* elements in the library. The following topics address the common features:

3.1. Element Input
3.2. Solution Output
3.3. Coordinate Systems

Some element features, or even element types, may be unavailable based on the product-licensing options in effect at your site. Any restrictions for an element are documented in the "Product Restrictions" section for that element.

3.1. Element Input

Element Library (p. 115) includes a summary table of element input. The table usually contains information about the following:

3.1.1. Element Name
3.1.2. Nodes
3.1.3. Degrees of Freedom
3.1.4. Material Properties
3.1.5. Special Element Features
3.1.6. Sections
3.1.7. Real Constants
3.1.8. Element Loading
3.1.9. Nodal Loading
3.1.10. Element Properties and Controls (KEYOPTs)

3.1.1. Element Name

An element type is identified by a name (eight characters maximum), such as SOLID285, consisting of a group label (SOLID) and a unique, identifying number (285). The element descriptions in Element Library (p. 115) are arranged in order of these identification numbers. The element is selected from the library for use in the analysis by inputting its name on the element type command (**ET**). See Element Classifications (p. 5) for a list of all available elements.

3.1.2. Nodes

The nodes associated with the element are listed as I, J, K, and so on. Elements are connected to the nodes in the sequence and orientation shown on the input figure for each element type.

Connectivity can be defined by automatic meshing, or can be input directly (**E**). The node numbers must correspond to the order indicated in the "Nodes" list. The I node is the first node of the element.

The node order determines the element coordinate system orientation for some element types. See Coordinate Systems (p. 55) for a description of the element coordinate system.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.1.2.1. Internal nodes

The program also uses internal nodes for elements with Lagrangian multiplier in their formulations, such as the mixed u-P formulation of PLANE182, MPC184 and Lagrangian-multiplier-based contact elements, and other elements such as BEAM188, SHELL208 and PIPE288 with quadratic and cubic options.

Internal nodes are generated automatically during the solution phase of an analysis to make more degrees of freedom available for the elements.

Internal nodes are inaccessible and require no input from you or any other action on your part.

3.1.3. Degrees of Freedom

Each element type has a degree-of-freedom set which constitutes the primary nodal unknowns to be determined by the analysis. The set can consist of displacements, rotations, temperatures, pressures, voltages, etc.

Derived results, such as stresses, heat flows, etc., are calculated from these degree-of-freedom results. You do not explicitly define degrees of freedom on the nodes, as they are implied by the element types attached to them. Your choice of element types is therefore of primary concern in any analysis.

3.1.3.1. Internal Degrees of Freedom

In addition to the standard (external) degrees of freedom, some elements or element-formulation options have *internal* degrees of freedom that are not user-accessible.

The program typically uses internal degrees of freedom to enhance the gradients of the primary unknowns of the elements so that better element behavior can be obtained, but they may also serve other purposes.

Internal degrees of freedom are not shared with nodes of any other elements so that they can be condensed out at the element level during the solution phase of the analysis.

3.1.4. Material Properties

Most element types use various material properties. Typical material properties include Young's modulus (of elasticity), density, coefficient of thermal expansion, and thermal conductivity. Each property is referenced by a label: EX, EY, and EZ for the directional components of Young's modulus, DENS for density, and so on. All material properties can be input as functions of temperature.

Linear Material Properties

Some properties for non-thermal analyses are called *linear* properties because typical solutions with these properties require only a single iteration. A basic description of the linear material properties is given in Linear Material Properties in the *Material Reference*.

Nonlinear Material Properties

Properties such as stress-strain data and B-H curves are called *nonlinear* because an analysis with these properties usually requires an iterative solution. A basic description of the nonlinear material properties is given in Material Models in the *Material Reference*.

Material Property Input

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Typically, *linear* material properties are input via the **MP** family of commands.

Nonlinear material properties are input via the **TB** family of commands.

The **TB** family of commands can, however, be used for some *linear* material properties (such as anisotropic elasticity, material structural damping, piezoelectric matrix, and piezoresistivity).

Some elements require other special data which must be input in tabular form. Such data are also input via the **TB** family of commands and are described with the element in Element Library (p. 115), or in Material Models in the *Material Reference* if they apply to a family of elements.

Materials for Explicit Dynamic Analyses

Material models used in explicit dynamic analyses are discussed in Material Models in the *ANSYS LS-DYNA User's Guide*.

Supported Material Properties for Each Element

See "Material Properties" in the documentation for each element type.

For a list of elements and the material properties supported for each, see Element Support for Material Models (p. 59).

3.1.5. Special Element Features

The "Special Features" list in the documentation for each element indicates that certain additional capabilities are available for the element. In most cases, the special features make the element nonlinear and therefore require an iterative solution.

Examples of special element features include:

Adaptive descent	Linear perturbation
Birth and death	Nonlinear stabilization
Debonding	Ocean loading
Initial state	Rezoning
Initial stress import	Section definition
Large deflection	Stress stiffening

3.1.6. Sections

Sections are generally used to describe the geometry in the missing direction(s) of an idealized model. By idealizing a structure and capturing calculations that are common to many elements, sections may significantly reduce the number of degrees of freedom and lower computational requirements.

Following are common section types:

- **Beam Sections** -- A beam cross section defines the geometry of the beam in a plane perpendicular to the element axial direction.
- **Shell Sections** -- A shell cross section defines the geometry of the shell in a plane parallel to the shell x-y plane.
- **Reinforcing Sections** -- Reinforcing sections allow you to describe the reinforcing fibers collectively within an existing structure rather than model each fiber individually.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Axisymmetric Sections** -- An axis of symmetry around which a plane of general axisymmetric solid elements is revolved.

Special section types are also available. For example, rather than describing a missing dimension of the model, a contact section corrects the geometry of a meshed circle, sphere, or cylinder.

For a complete list of available section types, see the **SECTYPE** command.

3.1.7. Real Constants

Data required for the calculation of the element matrices and load vectors, but which cannot be determined by other means, are input as real constants. Typical real constants include hourglass stiffness, contact parameters, stranded coil parameters, and plane thicknesses.

With each element type that supports them, a description of the real constants is given in the "Real Constants" list. The real constant values that you input (**R** and **RMORE**) must correspond to the order indicated in the list.

To learn more about how real constants are used within each element type, see the description of each element type in the *Mechanical APDL Theory Reference*.

3.1.8. Element Loading

Element loads are surface loads, body loads, inertial loads, and ocean loads. Element loads are always associated with a particular element (even if the input is at the nodes). Certain elements may also have *flags*.

Flags are not actually loads, but are used to indicate that a certain type of calculation is to be performed. For example, when the FSI (fluid-structure interaction) flag is active, a specified face of an acoustic element is treated as an interface between a fluid portion and a structural portion of the model. Similarly, MXWF and MVDI are flags used to trigger magnetic force (Maxwell surface) and Jacobian force (virtual displacement) calculations, respectively, in certain magnetics elements. Details about flags are discussed under the applicable elements in Element Library (p. 115).

Flags are associated either with a surface (FSI and MXWF) and are applied as surface loads (below), or with an element (MVDI) and are applied as body loads (below). For the FSI and MXWF flags, values have no meaning - these flags are simply turned on by specifying their label on the appropriate command. For the MVDI flag, its value (which can range from zero to one) is specified, along with the label, on the appropriate command. Flags are always step-applied (i.e., the **KBC** command does not affect them).

The following topics related to element loading are available:

3.1.8.1. Surface Loads
3.1.8.2. Body Loads
3.1.8.3. Inertial Loads
3.1.8.4. Initial Stresses
3.1.8.5. Ocean Loads

Also see Nodal Loading (p. 49), which refers to loads defined at the nodes and are not directly related to the elements.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.1.8.1. Surface Loads

Some element types allow surface loads. Surface loads are typically pressures for structural element types, convections or heat fluxes for thermal element types, and so on.

Table 3.1: Surface Loads Available in Each Discipline

Discipline	Surface Load	Label
Structural	Pressure	PRES [1] [2]
Thermal	Convection Heat Flux Infinite Surface	CONV HFLUX INF
Magnetic	Maxwell Surface Infinite Surface	MXWF INF
Electric	Maxwell Surface Surface Charge Density Infinite Surface Temperature	MXWF CHRGS INF TEMP
Fluid	Fluid-Structure Interface Impedance	FSI IMPD
All	Superelement Load Vector	SELV

1. Not to be confused with the PRES degree of freedom
2. Buoyancy, wave, current and other ocean loads can be applied to some line and surface element types. For more information, see **OCTYPE** and related ocean commands.

3.1.8.1.1. Applying Surface Loads

Surface loads (such as pressures for structural elements and convections for thermal element) are typically input via the **SF** and **SFE** commands.

Surface loads may also be input in a nodal format. For example, rather than applying surface loading to an element face, it may be convenient to apply the loading the face nodes of an element (which are then processed like face input). For more information, see Nodal Loading (p. 49).

3.1.8.1.2. Multiple Surface Loads

Some elements allow multiple types of surface loads (as shown with the load labels listed under "Surface Loads" in the input table for each element type). Also, some elements allow multiple loads on a single element face (as indicated with the load numbers after the load labels).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Load numbers are shown on the element figures (within circles) and point in the direction of positive load to the face upon which the load acts. A surface load applied on the edge of a shell element is on a per-unit-length basis, not per-unit area.

3.1.8.1.3. Surface Load Labels and Keys

Surface loads are designated by a label and a key. The label indicates the type of surface load and the key indicates where on the element the load acts.

The surface load can be defined on element faces via the **SFE** command by using a key (*LKEY*), the load label (*Lab*), and the load value. The **SF** command can define surface loads by using nodes to identify element faces. The CONV load label requires two values, the film coefficient and the bulk temperature.

3.1.8.1.4. Tapered Surface Load

A tapered surface load, which allows different values to be defined at the nodes of an element, can be entered via the **SFE** command. Tapered loads are input in the same order that the face nodes are listed.

3.1.8.2. Body Loads

Some element types allow body loads. Body loads are typically temperatures for structural element types, heat-generation rates for thermal element types, and so on.

Table 3.2: Body Loads Available in Each Discipline

Discipline	Body Load	Label
Structural	Temperature	TEMP [1]
	Fluence	FLUE
Thermal	Heat Generation Rate	HGEN
Magnetic	Temperature	TEMP [1]
	Current Density	JS
	Virtual Displacement	MVDI
	Voltage Drop	VLTG
Electric	Temperature	TEMP [1]
	Charge Density	CHRGD
Fluid	Heat Generation Rate	HGEN
	Force Density	FORC

1. Not to be confused with the TEMP degree of freedom

3.1.8.2.1. Applying Body loads

Body loads are designated in the "Input Summary" table of each element by a label and a list of load values at various locations within the element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Body loads are input via the **BF** or **BFE** commands. The load values input on the **BFE** command must correspond to the order indicated in the "Body Load" list.

Body loads can also be applied in a nodal format. For more information, see Nodal Loading (p. 49).

3.1.8.2.2. Temperature Body Loads

For some structural elements, the temperature does not contribute to the element load vector but is only used for material property evaluation. For thermal elements using the diagonalized specified heat matrix option in a transient analyses, a spatially varying heat generation rate is averaged over the element. Heat-generation rates are input per-unit-volume unless otherwise noted with the element. The element format is usually in terms of the element nodes but may be in terms of fictitious corner points as described for each element. Corner point numbers are shown on the element figures where applicable.

3.1.8.3. Inertial Loads

Inertial loads (gravity, spinning, etc.), are applicable to all elements with structural degrees of freedom and having mass (that is, elements having mass as an input real constant or having a density (DENS) material property).

Inertia loads are typically entered via the **ACEL** and **OMEGA** commands.

3.1.8.4. Initial Stresses

Initial stresses can be set as constant or read in from a file for most current-technology beam, link, plane, solid, and shell element types.

The **INISTATE** command sets constant initial stress for selected elements and, optionally, only for specified materials. The command also allows you to read in a file specifying the initial stresses. The stresses specified in the input file can be applied to the element centroids or element integration points, and can be applied to the same points for all selected elements or can be applied differently for each element. The stresses can also be a written to an external file.

For more information about the initial state capability, see the **INISTATE** command, and Initial State Loading in the *Basic Analysis Guide*.

3.1.8.5. Ocean Loads

Ocean loads can be applied to line and surface element types that support such loads. Ocean loading includes the effects of waves, current, drag, and buoyancy. Loading occurs via the ocean family of commands.

For more information, see Applying Ocean Loads in the *Basic Analysis Guide*.

3.1.9. Nodal Loading

Unlike element loads, nodal loads are defined at the nodes and are not directly related to the elements.

Nodal loads are associated with the degrees of freedom at the node and are typically entered via the **D** and **F** commands (such as nodal displacement constraints and nodal force loads).

The nodal or element loading format may be used for an element, with the element loading format taking precedence.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.1.10. Element Properties and Controls (KEYOPTs)

Elements use KEYOPTs (or *key options*) to control element behavior, formulation, and output for some beam and shell elements. The KEYOPT is the most important element input and should be selected with care.

Depending on the element type, the KEYOPTs can be used to control stress states, degrees of freedom, element technologies, formulation options, element coordinate systems, printout, results file (`.rst`) output, loads, temperatures, and so on.

A basic description of each KEYOPT is given with the detailed description of each element type. Because KEYOPTs can be combined, a great deal of flexibility is available for refining the element type to perfectly suit a specific problem.

KEYOPTs are identified by number, such as KEYOPT(1), KEYOPT(2), etc.. Each KEYOPT can be set to a specific value, as follows:

- Values for the first six KEYOPTs (KEYOPT(1) through KEYOPT(6)) can be input via the **ET** or **KEYOPT** command.
- Values for KEYOPT(7) or greater are input via the **KEYOPT** command.

3.1.10.1. Element KEYOPT Defaults

The following topics concern element KEYOPT defaults:

3.1.10.1.1. Consider Automatic Element Selection
3.1.10.1.2. KEYOPT Settings May Vary Between Products

3.1.10.1.1. Consider Automatic Element Selection

Although element KEYOPT defaults can accommodate many problem types, and KEYOPTs can be adjusted as needed, it is often more convenient to allow automatic selection of element technologies via the **ETCONTROL** command. The command sets element options based on the associated constitutive models of the element type.

For more information, see Automatic Selection of Element Technologies and Formulations.

3.1.10.1.2. KEYOPT Settings May Vary Between Products

Exercise caution when using the same input for other ANSYS, Inc. products, as some default KEYOPT settings may vary between products. Such cases are indicated in the "Product Restrictions" section of the affected elements. If you intend to use an input file in more than one product, it is a good practice to explicitly input these settings, rather than letting them default; otherwise, element behavior in the other product may differ.

3.2. Solution Output

Solution output is written to the:

- Output file (`Jobname.OUT`)
- Results file (`Jobname.RST`, `Jobname.RTH`, or `Jobname.RMG`)
- Program database (`Jobname.DB`)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The output file is viewable via the GUI. The database and results file data can be postprocessed.

The following solution output topics are available:

3.2.1. Output File Contents
3.2.2. Results File and Database Contents
3.2.3. Element Solution
3.2.4. Nodal Solution

3.2.1. Output File Contents

The output file contains the nodal degree-of-freedom solution, nodal and reaction loads, and the element solutions, depending on the **OUTPR** settings.

The element solutions are the solution values at the integration points or centroid. They are controlled by KEYOPTs in certain elements.

3.2.2. Results File and Database Contents

The results file contains data for all requested (**OUTRES**) solutions, or load steps. In POST1, the **SET** command identifies the load step that you intend to postprocess.

Results items for area and volume elements are generally retrieved from the database via standard results output commands (such as **PRNSOL**, **PLNSOL**, **PRESOL**, and **PLESOL**).

The labels on these commands correspond to the labels shown in the input and output description tables for each element. For example, postprocessing the stress in the material x direction (typically labeled SX) is identified as item S and component X on the postprocessing commands. Coordinate locations XC, YC, ZC are identified as item CENT and component X, Y, or Z.

Only items shown both on the given results command and in the element input/output tables are available for use with that command. (An exception is EPTO, the total strain, which is available for all structural solid and shell elements even though it is not shown in the output description tables for those elements.)

Generic labels do not exist for some results data, such as integration point data, all derived data for structural line elements (such as spars, beams, and pipes) and contact elements, all derived data for thermal line elements, and layer data for layered elements. Instead, a sequence number identifies those items.

3.2.3. Element Solution

The element output items (and their definitions) are shown along with the element type description. Not all of the items shown in the output table will appear at all times for the element. Normally, items not appearing are either not applicable to the solution or have all zero results and are suppressed to save space. However, except for the coupled-field elements PLANE223, SOLID226, and SOLID227, coupled-field forces appear if they are computed to be zero. The output is, in some cases, dependent on the input. For example, for thermal elements accepting either surface convection (CONV) or nodal heat flux (HFLUX), the output will be either in terms of convection or heat flux. Most of the output items shown appear in the element solution listing. Some items do not appear in the solution listing but are written to the results file.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Most elements have two tables which describe the output data and ways to access that data for the element. These tables are the "Element Output Definitions" table and the "Item and Sequence Numbers" tables used for accessing data through the **ETABLE** and **ESOL** commands.

Stresses and strains are two primary element solution quantities in structural elements used for stress analysis. Current-technology elements output Cauchy stresses, and logarithmic strains in large deflection analysis (**NLGEOM**,ON) or engineering strains in a small-deflection analysis (**NLGEOM**,OFF). Stresses and strains are directly evaluated at element integrations points, and may be extrapolated to element nodes or averaged at element centroid for output. Generalized stresses and strains, such as linearized stresses, forces, moments, and curvature changes, are available in beam, pipe, elbow, and shell elements.

3.2.3.1. The Element Output Definitions Table

The first table, "Element Output Definitions," describes possible output for the element. In addition, this table outlines which data are available for solution printout (`Jobname.OUT` and/or display to the terminal), and which data are available on the results file (`Jobname.RST`, `Jobname.RTH`, `Jobname.RMG`, etc.). Only the data which you request with the solution commands (**OUTPR** and **OUTRES**) are included in printout and on the results file, respectively.

As an added convenience, items in the table which are available via the Component Name method of the **ETABLE** command are identified by special notation (:) included in the output label.(See The General Postprocessor (POST1) in the *Basic Analysis Guide* for more information.) The label portion before the colon corresponds to the Item field on the **ETABLE** command, and the portion after the colon corresponds to the *Comp* field. For example, S:EQV is defined as equivalent stress, and the **ETABLE** command for accessing this data would be:

ETABLE,ABC,S,EQV

where ABC is a user-defined label for future identification on listings and displays. Other data having labels without colons can be accessed through the Sequence Number method, discussed with the "Item and Sequence Number" tables below.

In some cases there is more than one label which can be used after the colon, in which case they are listed and separated by commas. The **Definition** column defines each label and, in some instances, also lists the label used on the printout, if different. The **O** column indicates those items which are written to the output window and/or the output file. The **R** column indicates items which are written to the results file and which can be obtained in postprocessing; if an item is not marked in the **R** column, it cannot be stored in the "element table."

3.2.3.2. The Item and Sequence Number Table

Many elements also have a table, or set of tables, that list the *Item* and sequence number required for data access using the Sequence Number method of the **ETABLE** command. (See The General Postprocessor (POST1) in the *Basic Analysis Guide* for an example.)

The number of columns in each table and the number of tables per element vary depending on the type of data available and the number of locations on the element where data was calculated.

See Table 182.2: PLANE182 Item and Sequence Numbers for a sample item and sequence number table. Items listed as SMISC refer to summable miscellaneous items, while NMISC refers to non-summable miscellaneous items.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.2.3.3. Surface Loads

Pressure output for structural elements shows the input pressures expanded to the element's full linearly or bilinearly varying load capability.

See the **SF**, **SFE**, and **SFBEAM** commands for pressure input. Beam elements which allow an offset from the node on the **SFBEAM** command an have additional output labeled OFFST.

To save space, pressure output is often omitted when values are zero. Similarly, other surface load items (such as convection (CONV) and heat flux (HFLUX)), and body load input items (such as temperature (TEMP), fluence (FLUE), and heat generation (HGEN)), are often omitted when the values are zero (or, for temperatures, when the T-TREF values are zero).

To save space, surface output is often omitted when all values are zero.

For output of ocean-loading information on supported element types, see the **OCTYPE** and **OCDATA** commands.

3.2.3.4. Integration Point Solution (Output Listing Only)

Integration point output is available in the output listing with most current-technology elements. The location of the integration point is updated if large deflections are used. See the element descriptions in the *Mechanical APDL Theory Reference* for details about integration point locations and output. Also the **ERESX** command may be used to request integration point data to be written as nodal data on the results file.

3.2.3.5. Centroidal Solution (Output Listing Only)

Output (such as stress, strain, and temperature) in the output listing is given at the centroid (or near center) of certain elements. The location of the centroid is updated if large deflections are used.

The output quantities are calculated as the average of the integration point values. The component output directions for vector quantities correspond to the input material directions which, in turn, are a function of the element coordinate system. For example, the SX stress is in the same direction as EX.

In postprocessing, the **ETABLE** command can calculate the centroidal solution of each element from its nodal values.

3.2.3.6. Element Nodal Solution

The term *element nodal* means element data reported for each element at its nodes. This type of output is available for 2-D and 3-D solid elements, shell elements, and various other elements.

Element nodal data consist of the element derived data (such as strains, stresses, fluxes, and gradients) evaluated at each of the element's nodes. These data are usually calculated at the interior integration points and then extrapolated to the nodes.

Exceptions occur if an element has active (nonzero) plasticity, creep, or swelling at an integration point or if **ERESX**,NO is input. In such cases the nodal solution is the value at the integration point nearest the node.

Output is usually in the element coordinate system. Averaging of the nodal data from adjacent elements is done within the POST1 postprocessor.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.2.3.7. Element Nodal Loads

Element nodal loads refer to an element's loads (forces) acting on each of its nodes. They are printed out at the end of each element output in the nodal coordinate system and are labeled as static loads.

If the problem is dynamic, the damping loads and inertia loads are also printed.

You can control the output of element nodal loads via the **OUTPR**,NLOAD command (for printed output) and the **OUTRES**,NLOAD command (for results file output).

Element nodal loads relate to the reaction solution in the following way: the sum of the static, damping, and inertia loads at a particular degree of freedom, summed over all elements connected to that degree of freedom, plus the applied nodal load (**F** or **FK** command), is equal to the negative of the reaction solution at that same degree of freedom.

3.2.3.8. Nonlinear Solutions

For information about nonlinear solutions due to material nonlinearities, see Structures with Material Nonlinearities in the *Mechanical APDL Theory Reference*.

Nonlinear strain data (EPPL, EPCR, EPSW, etc.) are always the value from the nearest integration point.

If creep is present, stresses are computed after the plasticity correction but before the creep correction.

Elastic strains are printed after any creep corrections.

3.2.3.9. 2-D and Axisymmetric Solutions

A 2-D solid analysis is based upon a per-unit-of-depth calculation, and all appropriate output data are on a per-unit-of-depth basis. Many 2-D solids, however, allow an option to specify the depth (thickness).

An axisymmetric analysis is based on 360°. Calculation and all appropriate output data are on a full 360° basis. In particular, the total forces for the 360° model are output for an axisymmetric structural analysis and the total convection heat flow for the 360° model is output for an axisymmetric thermal analysis.

For axisymmetric analyses, the X, Y, Z, and XY stresses and strains correspond to the radial, axial, hoop, and in-plane shear stresses and strains, respectively. The global Y axis must be the axis of symmetry, and the structure should be modeled in the +X quadrants.

3.2.3.10. Member Force Solution

Member force output is available with most structural line elements. The listing of this output is activated via a KEYOPT described with the element and is in addition to the nodal load output.

Member forces are in the element coordinate system and the components correspond to the degrees of freedom available with the element.

3.2.3.11. Failure Criteria

Failure criteria are commonly used for determining the damage initiation in orthotropic materials. Based on various assumptions on the material damage mechanism, failure criteria are usually formulated with functions of element solution (stresses or strains) and material strength limits.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Two types of failure criteria results are possible: those that are available during any type of analysis, and those that are available only during a progressive failure damage analysis. Failure criteria results are accessible via standard postprocessing output commands (**PLESOL**, **PRESOL**, **PLNSOL**, and **PRNSOL**) and the **ETABLE** command:

- **Available during any type of analysis (accessible via the FAIL item)**

  These failure criteria are computed on the fly when requested, using stored element strains and stresses in the result file. This type of failure criteria are suitable for determining whether and where the material damage may first occur. If the material is damaged, the stresses used for this calculation are calculated from the strains and the damaged material stiffnesses.

- **Available only during a progressive failure damage analysis (accessible via the PFC item)**

  These failure criteria are calculated from element strains and effective stresses (stresses that would occur if the material were undamaged) during solution and stored in the result file. They indicate whether the material damage may be expected to continue.

For more information, see Failure Criteria in the *Mechanical APDL Theory Reference* and Specifying Failure Criteria for Composites in the *Structural Analysis Guide*.

3.2.4. Nodal Solution

The nodal solution from an analysis consists of:

- the degree-of-freedom solution, such as nodal displacements, temperatures, and pressures
- the reaction solution calculated at constrained nodes - forces at displacement constraints, heat flows at temperature degree-of-freedom constraints, fluid flows at pressure degree-of-freedom constraints, and so on.

The degree-of-freedom solution is calculated for all active degrees of freedom in the model, which are determined by the union of all degree-of-freedom labels associated with all the active element types. It is output at all degrees of freedom that have a nonzero stiffness or conductivity and can be controlled by **OUTPR**,NSOL (for printed output) and **OUTRES**,NSOL (for results file output).

The reaction solution is calculated at all nodes that are constrained (**D**, **DSYM**, etc.). Its output can be controlled by **OUTPR**,RSOL and **OUTRES**,RSOL.

For vector degrees of freedom and corresponding reactions, the output during solution is in the nodal coordinate system. If a node was input with a rotated nodal coordinate system, the output nodal solution will also be in the rotated coordinate system. For a node with the rotation $\theta_{xy} = 90°$, the printed UX solution will be in the nodal X direction, which in this case corresponds to the global Y direction. Rotational displacements (ROTX, ROTY, ROTZ) are output in radians, and phase angles from a harmonic analysis are output in degrees.

3.3. Coordinate Systems

The following coordinate system topics are available:

3.3.1. Understanding the Element Coordinate System
3.3.2. Elements That Operate in the Nodal Coordinate System
3.3.3. Solution Coordinate Systems

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.3.1. Understanding the Element Coordinate System

The element coordinate system is used for:

- Orthotropic material input
- Pressure loading input on certain faces of the surface effect elements
- Output of element quantities, such as stresses, strains, and thermal gradients.

A default element coordinate system orientation is associated with each element type. In general, these systems are described below. Elements departing from this description have their default element coordinate system orientation described in Element Library (p. 115).

Element coordinate systems are right-handed, orthogonal systems.

Line elements For line elements, the default orientation is generally with the x-axis along the element I-J line.

Solid elements For solid elements (such as PLANE182 or SOLID185), the default orientation is generally parallel to the global Cartesian coordinate system.

General axisymmetric elements For general axisymmetric solid elements (SOLID272 and SOLID273), the general orientation is determined by the cylindrical coordinate system, with z and the origin defined via **SECTYPE** and **SECDATA** commands, and $\theta = 0$ on the master plane; the r, θ, and z directions adhere to the right-hand rule.

Shell elements For area shell elements (such as SHELL181), the default orientation generally has the x-axis aligned with element I-J side, the z-axis normal to the shell surface (with the outward direction determined by the right-hand rule around the element from node I to J to K), and the y-axis perpendicular to the x and z-axes.

3.3.1.1. Modifying the Element Orientation

Unless otherwise changed, the element coordinate system orientation is the default orientation for that element type. The orientation may be changed for area and volume elements by making it parallel to a previously defined local system (**ESYS**) or, for some elements, by a KEYOPT selection. If both are specified, the **ESYS** definition overrides.

A further rotation, relative to the previous orientation, is allowed for some elements by a real constant angle specification (see, for example, the real constant THETA for SHELL181). If no **ESYS** or KEYOPT orientation is specified, the real constant angle rotation (if any) is relative to the default orientation. The coordinate systems of axisymmetric elements may only be rotated about the global Z-axis; however, general axisymmetric elements can be rotated about any axis.

Shell elements For shell elements, the **ESYS** orientation uses the projection of the local system on the shell surface. The element x-axis is determined from the projection of the local x-axis on the shell surface. If the projection is a point (or the angle between the local x-axis and the normal to the shell is 0° (plus a tolerance of 45°)), the local y-axis projection is used for the element x-axis direction. The z and y-axes are determined as described for the default orientation.

For non-midside-node shell elements, the projection is evaluated at the element centroid and is assumed constant in direction throughout the element. For midside-node shell

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

elements, the projection is evaluated at each integration point and may vary in direction throughout the element.

Axisymmetric elements For axisymmetric elements, only rotations in the X-Y plane are valid. Some elements also allow element coordinate system orientations to be defined by user written subroutines (see the *Guide to User-Programmable Features*).

Layered elements Layered elements use the x-axis of the element coordinate system as a base from which to rotate each layer to the layer coordinate system. The layers are rotated by the angles input on the **SECDATA** or **RMORE** commands. Material properties, stresses, and strains for layered elements are based on the layer coordinate system, not the element coordinate system.

All element coordinate systems shown in the element figures assume that no **ESYS** orientation is specified.

The orientation of output element quantities can be adjusted. See Rotating Results to a Different Coordinate System in the *Basic Analysis Guide*

Element coordinate systems can be displayed as a triad with the **/PSYMB** command or as an **ESYS** number (if specified) with the **/PNUM** command. Triad displays do not include the effects of any real constant angle.

For large-deflection analyses, the element coordinate system rotates from the initial orientation described above by the amount of rigid body rotation of the element.

3.3.2. Elements That Operate in the Nodal Coordinate System

A few special elements operate solely within the nodal coordinate system:

COMBIN14 Spring-Damper with KEYOPT(2) = 1, 2, 3, 4, 5, or 6
MATRIX27 Stiffness, Damping, or Mass Matrix
COMBIN37 Control Element
FLUID38 Dynamic Fluid Coupling
COMBIN39 Nonlinear Spring with KEYOPT(4) = 0
COMBIN40 Combination Element
TRANS126 Electromechanical Transducer
COMBI214 2-D Spring-Damper Bearing

These elements, defined in the nodal coordinate system, allow for easy directional control, especially for the case of two-node elements with coincident nodes.

If UX, UY, or UZ degrees of freedom are being used, the nodes are not coincident, and the load is not acting parallel to the line connecting the two nodes, there is no mechanism for the element to transfer the resulting moment load, resulting in loss of moment equilibrium. The sole exception is MATRIX27, which can include moment coupling when appropriate additional terms are added to the matrix.

You cannot sum nodal force and moment contributions (**FSUM**) for these elements if the 1-D option is activated and nodes are rotated (**NROTAT**).

If any of the nodes have been rotated (**NROTAT**), consider the following:

- If the nodes of elements containing more than one node are not rotated in precisely the same way, force equilibrium may not be maintained.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Accelerations operate normally in the global Cartesian system. Because no transformation is done between the nodal and global systems, however, the accelerations effectively act on any element mass in the nodal system, giving unexpected results. It is therefore good practice not to apply accelerations when these elements use rotated nodes.
- Mass and inertia relief calculations will not be correct.

3.3.3. Solution Coordinate Systems

For homogeneous elements, element solution coordinate systems are generally identical to the nodal or the element coordinate systems adopted by the elements.

For composite elements, solution coordinate systems can be independently defined for different components, such as layers in a layered-shell element (SHELL181) and fibers in a discrete reinforcing element (REINF264). The solution coordinate systems in composite elements are also called *layered coordinate systems*, and are specifically identified via the **RSYS**,LSYS command.

For more information, see Rotating Results to a Different Coordinate System in the *Basic Analysis Guide*.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 4: Element Support for Material Models

Following is a list of element types that support various material constitutive properties. To take advantage of the many material properties available, use current-technology (rather than legacy) elements in your analysis.

Material models are specified via the **TB**,*Lab* command, where *Lab* represents the material model label (shortcut name). For more information about each material model, see the *Material Reference*.

For a list of *material models* and the elements that support them, see Material Model Element Support in the *Material Reference*.

Element	Description	Supported Material Properties
SOLID5 (legacy)	3-D Coupled-Field Solid	Anisotropic elasticity, Magnetic, Piezoelectric matrix
PLANE13 (legacy)	2-D Coupled-Field Solid	Anisotropic elasticity, Magnetic, Multilinear kinematic hardening, Piezoelectric matrix
FLUID30	3-D Acoustic Fluid	Johnson-Champoux-Allard Equivalent Fluid Model of a Porous Media
PLANE53 (legacy)	2-D 8-Node Magnetic Solid	Magnetic
SOLID65 (legacy)	3-D Reinforced Concrete Solid	Anisotropic plasticity, Bilinear isotropic hardening, Bilinear kinematic hardening, Concrete *or* concrete damage, Creep, Drucker-Prager plasticity, Multilinear elasticity, Multilinear isotropic hardening, Multilinear kinematic hardening, State variables (user-defined), User-defined
SOLID96	3-D Magnetic Scalar Solid	Magnetic
SOLID97	3-D Magnetic Solid	
SOLID98 (legacy)	Tetrahedral Coupled-Field Solid	Anisotropic elasticity, Magnetic, Piezoelectric matrix
LINK160 (explicit)	Explicit 3-D Spar (or Truss)	Bilinear kinematic hardening, Plasticity laws
BEAM161 (explicit)	Explicit 3-D Beam	Bilinear kinematic hardening, Viscoelasticity, Plasticity laws
PLANE162 (explicit)	Explicit 2-D Structural Solid	Bilinear isotropic hardening, Bilinear kinematic hardening, Composite damage, Viscoelasticity, Foam, Honeycomb, Mooney-Rivlin hyperelasticity, Plasticity laws

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Element	Description	Supported Material Properties
SHELL163 (explicit)	Explicit Thin Structural Shell	Bilinear isotropic hardening, Bilinear kinematic hardening, Composite damage, Mooney-Rivlin hyperelasticity, Plasticity laws
SOLID164 (explicit)	Explicit 3-D Structural Solid	Anisotropic elasticity, Bilinear isotropic hardening, Bilinear kinematic hardening, Composite damage, Concrete damage, Viscoelasticity, Foam, Geological cap, Honeycomb, Mooney-Rivlin hyperelasticity, Plasticity laws
COMBI165 (explicit)	Explicit Spring-Damper	Explicit spring-damper (discrete)
SOLID168 (explicit)	Explicit 3-D 10-Node Tetrahedral Structural Solid	Anisotropic elasticity, Bilinear isotropic hardening, Bilinear kinematic hardening, Composite damage, Concrete damage, Viscoelasticity, Foam, Geological cap, Honeycomb, Mooney-Rivlin hyperelasticity, Plasticity laws
CONTA171	2-D 2-Node Surface-to-Surface Contact	Cohesive zone, Coefficient of friction, User-defined friction, User-defined contact interaction, Contact surface wear Orthotropic friction (**TB**,FRIC,,,,ORTHO) is not applicable to the 2-D contact elements CONTA171 and CONTA172. Contact surface wear is not available for CONTA176 and CONTA177.
CONTA172	2-D 3-Node Surface-to-Surface Contact	
CONTA173	3-D 4-Node Surface-to-Surface Contact	
CONTA174	3-D 8-Node Surface-to-Surface Contact	
CONTA175	2-D/3-D Node-to-Surface Contact	
CONTA176	3-D Line-to-Line Contact	
CONTA177	3-D Line-to-Surface Contact	
CONTA178	3-D Node-to-Node Contact	Coefficient of friction, User-defined friction, User-defined contact interaction Orthotropic friction (**TB**,FRIC,,,,ORTHO) is not applicable to CONTA178.
LINK180	3-D Spar (or Truss)	Bilinear isotropic hardening, Bilinear kinematic hardening, Chaboche nonlinear kinematic hardening, Creep, Coefficient of thermal expansion, Damage evolution law, Damage initiation criteria, Elasticity, Hill anisotropy, Multilinear isotropic hardening. Multilinear kinematic hardening, Voce isotropic hardening law, Prony series constants for viscoelastic materials, Rate-dependent plasticity (viscoplasticity), Rate-independent plasticity, Shift function for viscoelastic materials, State variables (user-defined), User-defined
SHELL181	4-Node Structural Shell	Anisotropic hyperelasticity, Anisotropic elasticity, Bergstrom-Boyce, Bilinear isotropic hardening, Bilinear kinematic hardening, Creep, Coefficient of thermal ex-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Element	Description	Supported Material Properties
		pansion, Mullins effect, Chaboche nonlinear kinematic hardening, Creep, Damage evolution law, Damage initiation criteria, Elasticity, Hill anisotropy, Hyperelasticity, Multilinear isotropic hardening, Multilinear kinematic hardening, Voce isotropic hardening law, Prony series constants for viscoelastic materials, Rate-dependent plasticity (viscoplasticity), Rate-independent plasticity, Material structural damping, Shape memory alloy (SMA), Shift function for viscoelastic materials, State variables (user-defined), Swelling, User-defined
PLANE182	2-D 4-Node Structural Solid	Anisotropic hyperelasticity, Anisotropic elasticity, Bergstrom-Boyce, Bilinear isotropic hardening, Bilinear kinematic hardening, Cast iron, Mullins effect, Crack growth fracture criterion, Chaboche nonlinear kinematic hardening, Creep, Coefficient of thermal expansion, Damage evolution law, Damage initiation criteria, Elasticity, Extended Drucker-Prager, Gurson pressure-dependent plasticity, Hill anisotropy, Hyperelasticity, Microplane, Multilinear isotropic hardening, Multilinear kinematic hardening, Voce isotropic hardening law, Prony series constants for viscoelastic materials, Rate-dependent plasticity (viscoplasticity), Rate-independent plasticity, Material structural damping, Shift function for viscoelastic materials, Shape memory alloy, State variables (user-defined), Swelling, Uniaxial stress-strain relation, User-defined
PLANE183	2-D 8-Node or 6-Node Structural Solid	
MPC184	Multipoint Constraint	Joint (linear and nonlinear elastic stiffness, linear and nonlinear damping, and frictional behavior)
SOLID185	3-D 8-Node Structural Solid	Anisotropic hyperelasticity, Anisotropic elasticity, Bergstrom-Boyce, Bilinear isotropic hardening, Bilinear kinematic hardening, Cast iron, Mullins effect, Crack growth fracture criterion, Chaboche nonlinear kinematic hardening, Creep, Coefficient of thermal expansion, Damage evolution law, Damage initiation criteria, Elasticity, Extended Drucker-Prager, Gurson pressure-dependent plasticity, Hill anisotropy, Hyperelasticity, Microplane, Multilinear isotropic hardening, Multilinear kinematic hardening, Voce isotropic hardening law, Prony series constants for viscoelastic materials, Rate-dependent plasticity (viscoplasticity), Rate-independent plasticity, Material structural damping, Shift function for viscoelastic materials, Shape memory alloy, State variables (user-defined), Swelling, Uniaxial stress-strain relation, User-defined
SOLID186	3-D 20-Node Structural Solid	Anisotropic hyperelasticity, Anisotropic elasticity, Bergstrom-Boyce, Bilinear isotropic hardening, Bilinear kinematic hardening, Cast iron, Mullins effect, Chaboche nonlinear kinematic hardening, Creep, Coefficient of thermal expansion, Damage evolution
SOLID187	3-D 10-Node Tetrahedral Structural Solid	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Element	Description	Supported Material Properties
		law, Damage initiation criteria, Elasticity, Extended Drucker-Prager, Gurson pressure-dependent plasticity, Hill anisotropy, Hyperelasticity, Microplane, Multilinear isotropic hardening, Multilinear kinematic hardening, Voce isotropic hardening law, Prony series constants for viscoelastic materials, Rate-dependent plasticity (viscoplasticity), Rate-independent plasticity, Material structural damping, Shift function for viscoelastic materials, Shape memory alloy, State variables (user-defined), Swelling, Uniaxial stress-strain relation, User-defined
BEAM188	3-D 2-Node Beam	Bilinear isotropic hardening, Bilinear kinematic hardening, Chaboche nonlinear kinematic hardening, Creep, Coefficient of thermal expansion, Damage evolution law, Damage initiation criteria, Elasticity, Hill anisotropy, Multilinear isotropic hardening, Multilinear kinematic hardening, Voce isotropic hardening law, Prony series constants for viscoelastic materials, Rate-dependent plasticity (viscoplasticity), Rate-independent plasticity, Shift function for viscoelastic materials, Shape memory alloy (SMA), State variables (user-defined), Swelling, User-defined
BEAM189	3-D 3-Node Beam	
SOLSH190	3-D 8-Node Structural Solid Shell	Anisotropic elasticity, Bergstrom-Boyce, Bilinear isotropic hardening, Bilinear kinematic hardening, Cast iron, Mullins effect, Chaboche nonlinear kinematic hardening, Creep, Coefficient of thermal expansion, Damage evolution law, Damage initiation criteria, Extended Drucker-Prager, Elasticity, Gurson pressure-dependent plasticity, Hill anisotropy, Hyperelasticity, Multilinear kinematic hardening, Multilinear isotropic hardening, Voce isotropic hardening law, Prony series constants for viscoelastic materials, Rate-dependent plasticity (viscoplasticity), Rate-independent plasticity, Material structural damping, Shift function for viscoelastic materials, Shape memory alloy, State variables (user-defined), Swelling, Uniaxial stress-strain relation, User-defined
INTER192	2-D 4-Node Gasket	Gasket
INTER193	2-D 6-Node Gasket	
INTER194	3-D 16-Node Gasket	
INTER195	3-D 8-Node Gasket	
INTER202	2-D 4-Node Cohesive	Cohesive zone
INTER203	2-D 6-Node Cohesive	
INTER204	3-D 16-Node Cohesive	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Element	Description	Supported Material Properties
INTER205	3-D 8-Node Cohesive	
SHELL208	2-Node Axisymmetric Shell	Anisotropic hyperelasticity, Anisotropic elasticity, Bergstrom-Boyce, Bilinear isotropic hardening, Bilinear kinematic hardening, Mullins effect, Chaboche nonlinear kinematic hardening, Creep, Coefficient of thermal expansion, Damage evolution law, Damage initiation criteria, Elasticity, Hill anisotropy, Hyperelasticity, Multilinear isotropic hardening, Multilinear kinematic hardening, Voce isotropic hardening law, Prony series constants for viscoelastic materials, Rate-dependent plasticity (viscoplasticity), Rate-independent plasticity, Material structural damping, Shift function for viscoelastic materials, Shape memory alloy (SMA), State variables (user-defined), Swelling, User-defined
SHELL209	3-Node Axisymmetric Shell	Anisotropic hyperelasticity, Anisotropic elasticity, Bergstrom-Boyce, Bilinear isotropic hardening, Bilinear kinematic hardening, Mullins effect, Chaboche nonlinear kinematic hardening, Creep, Coefficient of thermal expansion, Damage evolution law, Damage initiation criteria, Elasticity, Hill anisotropy, Hyperelasticity, Multilinear isotropic hardening, Multilinear kinematic hardening, Voce isotropic hardening law, Prony series constants for viscoelastic materials, Rate-dependent plasticity (viscoplasticity), Rate-independent plasticity, Material structural damping, Shift function for viscoelastic materials, Shape memory alloy (SMA), State variables (user-defined), Swelling, User-defined
CPT212	2-D 4-Node Coupled Pore-Pressure Mechanical Solid	Elasticity, Coefficient of thermal expansion, Hyperelasticity, Coupled Pore-Fluid Diffusion and Structural Model of Porous Media
CPT213	2-D 8-Node Coupled Pore-Pressure Mechanical Solid	
CPT215	3-D 8-Node Coupled Pore-Pressure Mechanical Solid	
CPT216	3-D 20-Node Coupled Pore-Pressure Mechanical Solid	
CPT217	3-D 10-Node Coupled Pore-Pressure Mechanical Solid	
FLUID220	3-D Acoustic Fluid 20-Node Solid Element	Johnson-Champoux-Allard Equivalent Fluid Model of a Porous Media

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Element	Description	Supported Material Properties
FLUID221	3-D Acoustic Fluid 10-Node Solid Element	Johnson-Champoux-Allard Equivalent Fluid Model of a Porous Media
PLANE223	2-D 8-Node Coupled-Field Solid	Anisotropic elasticity, Bilinear isotropic hardening, Bilinear kinematic hardening, Cast iron, Chaboche nonlinear kinematic hardening, Creep, Coefficient of thermal expansion, Anisotropic electric permittivity, Extended Drucker-Prager, Elasticity, Gurson pressure-dependent plasticity, Hill anisotropy, Multilinear isotropic hardening, Multilinear kinematic hardening, Voce isotropic hardening law, Piezoelectric matrix, Prony series constants for viscoelastic materials, Piezoresistivity, Rate-dependent plasticity (viscoplasticity), Rate-independent plasticity, Material structural damping, Shift function for viscoelastic materials, Shape memory alloy, Uniaxial stress-strain relation These material properties do not apply to piezoelectric analyses (**TB**,PIEZ) where KEYOPT(1) = 1001 or 1011.
SOLID226	3-D 20-Node Coupled-Field Solid	
SOLID227	3-D 10-Node Coupled-Field Solid	
PLANE233	2-D 8-Node Electromagnetic Solid	Magnetic
SOLID237	3-D 10-Node Electromagnetic Solid	
HSFLD241	2-D Hydrostatic Fluid	Fluid
HSFLD242	3-D Hydrostatic Fluid	
REINF263	2-D Smeared Reinforcing	Bilinear isotropic hardening, Bilinear kinematic hardening, Chaboche nonlinear kinematic hardening, Creep, Coefficient of thermal expansion, Elasticity, Hill anisotropy, Multilinear isotropic hardening, Multilinear kinematic hardening, Microplane, Voce isotropic hardening law, Prony series constants for viscoelastic materials, Rate-dependent plasticity (viscoplasticity), Rate-independent plasticity, Shift function for viscoelastic materials, State variables (user-defined), Swelling, User-defined
REINF264	3-D Discrete Reinforcing	
REINF265	3-D Smeared Reinforcing	
SOLID272	General Axisymmetric Solid with 4 Base Nodes	Anisotropic hyperelasticity, Anisotropic elasticity, Bergstrom-Boyce, Bilinear isotropic hardening, Bilinear kinematic hardening, Cast iron, Mullins effect, Chaboche nonlinear kinematic hardening, Creep, Coefficient of thermal expansion, Damage evolution law, Damage initiation criteria, Extended Drucker-Prager, Shape memory alloy (SMA)
SOLID273	General Axisymmetric Solid with 8 Base Nodes	
SHELL281	8-Node Structural Shell	Anisotropic hyperelasticity, Anisotropic elasticity, Bergstrom-Boyce, Bilinear isotropic hardening, Bilinear kinematic hardening, Mullins effect, Chaboche nonlin-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Element	Description	Supported Material Properties
		ear kinematic hardening, Creep, Coefficient of thermal expansion, Damage evolution law, Damage initiation criteria, Elasticity, Hill anisotropy, Hyperelasticity, Multilinear isotropic hardening, Multilinear kinematic hardening, Voce isotropic hardening law, Prony series constants for viscoelastic materials, Rate-dependent plasticity (viscoplasticity), Rate-independent plasticity, Material structural damping, Shift function for viscoelastic materials, Shape memory alloy (SMA), State variables (user-defined), Swelling, User-defined
SOLID285	3-D 4-Node Tetrahedral Structural Solid with Nodal Pressures	Anisotropic hyperelasticity, Anisotropic elasticity, Bergstrom-Boyce, Bilinear isotropic hardening, Bilinear kinematic hardening, Cast iron, Mullins effect, Chaboche nonlinear kinematic hardening, Creep, Coefficient of thermal expansion, Damage evolution law, Damage initiation criteria, Extended Drucker-Prager, Elasticity, Hill anisotropy, Hyperelasticity, Multilinear isotropic hardening, Multilinear kinematic hardening, Voce isotropic hardening law, Prony series constants for viscoelastic materials, Rate-dependent plasticity (viscoplasticity), Rate-independent plasticity, Material structural damping, Shift function for viscoelastic materials, Shape memory alloy, State variables (user-defined), Swelling, Uniaxial stress-strain relation, User-defined
PIPE288	3-D 2-Node Pipe	Anisotropic hyperelasticity, Anisotropic elasticity, Bergstrom-Boyce, Bilinear isotropic hardening, Bilinear kinematic hardening, Cast iron, Mullins effect, Chaboche nonlinear kinematic hardening, Creep, Coefficient of thermal expansion, Damage evolution law, Damage initiation criteria, Extended Drucker-Prager, Elasticity, Hill anisotropy, Hyperelasticity, Multilinear isotropic hardening, Multilinear kinematic hardening, Voce isotropic hardening law, Prony series constants for viscoelastic materials, Rate-dependent plasticity (viscoplasticity), Rate-independent plasticity, Shift function for viscoelastic materials, Shape memory alloy (SMA), State variables (user-defined), Swelling, User-defined
PIPE289	3-D 3-Node Pipe	
ELBOW290	3-D 3-Node Elbow	Anisotropic hyperelasticity, Anisotropic elasticity, Bergstrom-Boyce, Bilinear isotropic hardening, Bilinear kinematic hardening, Mullins effect, Chaboche nonlinear kinematic hardening, Creep, Coefficient of thermal expansion, Damage evolution law, Damage initiation criteria, Extended Drucker-Prager, Elasticity, Hill anisotropy, Hyperelasticity, Multilinear isotropic hardening, Multilinear kinematic hardening, Voce isotropic hardening law, Prony series constants for viscoelastic materials, Rate-dependent plasticity (viscoplasticity), Rate-independent plasticity, Material structural damp-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Element	Description	Supported Material Properties
		ing, Shift function for viscoelastic materials, State variables (user-defined), Swelling, User-defined

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 5: Elements for Stress Analysis

Elements for stress analysis are used primarily in mechanical and civil engineering. Various categories of elements are available to meet the needs of many types of industries. The element categories can be identified by how the elements represent the geometry of the structures, the basic deformation and/or formulation assumptions, and the primary purpose of the elements.

The following related topics are available:

5.1. Continuum Stress Elements
5.2. Shell Elements
5.3. Beam and Link (Truss) Elements
5.4. Pipe Elements
5.5. Reinforcing Elements
5.6. Surface Elements
5.7. Automatic Selection of Element Technologies and Formulations
5.8. Elements Under Linear Perturbation

5.1. Continuum Stress Elements

Continuum stress elements represent the volume of a model used for deflection and stress analysis. They do *not* include shell, surface, contact, beam, pipe or other special elements.

Continuum stress elements have few, if any, basic deformation assumptions and are the primary elements for stress analysis, especially for solid structures.

Elements can be either 2-D or 3-D and are usually obtained by meshing areas or volumes. They can be used for any structural analysis.

Although continuum stress elements may not be computationally efficient for structures having one or two dimensions that are much smaller than the remaining dimension(s), they are still useful for benchmarking purposes, or to obtain more detailed results for such structures.

The following topics related to continuum elements are available:

5.1.1. Stress States in 2-D Continuum Elements
5.1.2. Element Technologies
5.1.3. Mixed u-P Formulation
5.1.4. Generalized Plane Strain Options
5.1.5. Axisymmetric Elements with Nonaxisymmetric Loads

5.1.1. Stress States in 2-D Continuum Elements

The following topics about stress states in continuum elements assumes global Cartesian directions and applies to the PLANE182 and PLANE183 elements unless otherwise noted:

5.1.1.1. Plane Stress
5.1.1.2. Plane Strain
5.1.1.3. Generalized Plane Strain
5.1.1.4. Axisymmetric

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.1.1.5. General Axisymmetric

5.1.1.1. Plane Stress

Plane stress is a stress state that varies only in the X and Y directions. The stress in the Z direction is always zero, as are the XZ and YZ shear stresses.

Plane stress is generally used on flat, thin structures, where the deformation is assumed to be solely in the plane of the structure. For example, flat plates pulled in tension may be analyzed with plane stress. Although a unit thickness is generally assumed, you can input a thickness as well.

5.1.1.2. Plane Strain

Plane strain is a stress state that varies only in the X and Y directions. The strain in the Z direction is always zero, as are the XZ and YZ shear strains.

Plane strain is generally used on thick structures such as the cross-section of a dam.

5.1.1.3. Generalized Plane Strain

Generalized plane strain is similar to plane strain, except that the Z direction strain is not required to be zero. The Z direction strain may vary linearly in the X and Y directions (but not in the Z direction), removing the implied restriction of plane strain that the model extends without limit in the Z direction. The model in the Z direction can also have a uniform curvature.

Generalized plane strain can be used for modeling a section of cylindrical geometries, or to model long structures with a load at the end.

For more information, see Generalized Plane Strain Options (p. 74).

5.1.1.4. Axisymmetric

Axisymmetric geometry is defined in the X-Y plane with positive X values. The geometry can be thought of as the cross-section, which, when rotated about the global Cartesian Y axis, forms the actual model. Unless otherwise described, therefore, the model *and* the loading and deformations are constant around the circumference.

An axisymmetric analysis can be used for a nuclear containment building under internal pressure or vertical earthquake.

Besides PLANE182 and PLANE183, SHELL208 and SHELL209 support axisymmetric analyses.

5.1.1.5. General Axisymmetric

General axisymmetric is the same as axisymmetric except that the deformations can be nonaxisymmetric.

ANSYS, Inc. recommends using the SOLID272 and SOLID273 elements to simulate general axisymmetric characteristics. The general axisymmetric elements offer the following benefits:

- The deformations can be nonaxisymmetric.
- The deformations are modeled internally as a Fourier series, with all terms of the Fourier series solved simultaneously.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- All nonlinearities are supported.

An example general axisymmetric application is simulating threaded pipe connections.

For more information, see General Axisymmetric Elements (p. 76).

5.1.1.5.1. Alternate General Axisymmetric Method for Linear Problems

Alternatively, general axisymmetric can be simulated with harmonic axisymmetric elements when the problem is linear.

The deformations are modeled as a Fourier series, with each term of the Fourier series solved separately and then combined in postprocessing, and you must perform both tasks manually. Supported elements are PLANE25, PLANE83, and SHELL61.

Only linear behavior is supported.

An example axisymmetric-harmonic application is simulating a nuclear containment building subjected to a horizontal earthquake.

For more information, see Harmonic Axisymmetric Elements (p. 83).

Due to the limitations of axisymmetric-harmonic elements, ANSYS, Inc. recommends using general axisymmetric elements instead. The method presented here is provided for legacy support only.

5.1.2. Element Technologies

Traditional isoparametric continuum elements have never been sufficiently robust or accurate enough for general stress analysis.

Even in a linear analysis, lower-order elements are prone to shear locking in bending-dominated problems. Both lower- and higher-order elements experience volumetric locking when materials are incompressible or nearly incompressible, yet most nonlinear materials are either nearly or fully incompressible.

Current-technology continuum elements are available to overcome such problems. As technology advances, ANSYS, Inc. continues to develop robust new element types using the latest technologies available. Current-technology elements are more advanced and feature-rich than legacy elements.

The following topics concerning current element technologies for general stress analysis are available:

5.1.2.1. Technologies for Lower-Order Quad and Brick Elements
5.1.2.2. Technologies for Lower-Order Tetrahedron Elements
5.1.2.3. Technologies for Higher-Order Quad or Brick Elements
5.1.2.4. Technologies for Higher-Order Triangle or Tetrahedron Elements

5.1.2.1. Technologies for Lower-Order Quad and Brick Elements

The following current technologies are available for lower-order quad and brick elements:

5.1.2.1.1. B-bar Method (Selective Reduced Integration)
5.1.2.1.2. Enhanced Strain Formulation
5.1.2.1.3. Simplified Enhanced Strain Formulation
5.1.2.1.4. Uniform Reduced Integration

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.1.2.1.1. B-bar Method (Selective Reduced Integration)

The B-bar ($\bar{B}$) method helps to prevent volumetric locking in nearly incompressible cases. It replaces volumetric strain at the Gauss integration point with the average volumetric strain of the elements.

This method cannot, however, prevent any shear locking in bending-dominated problems. In such cases, use the enhanced strain formulation of the element. If it is not clear whether the deformation is bending-dominated, the enhanced strain formulation is still recommended.

This technology can be used with mixed u-P formulations.

5.1.2.1.2. Enhanced Strain Formulation

The enhanced strain formulation prevents shear locking in bending-dominated problems and volumetric locking in nearly incompressible cases. The formulation introduces certain number of internal (and inaccessible) degrees of freedom to overcome shear locking, and an additional internal degree of freedom for volumetric locking (except for the case of plane stress in 2-D elements). All internal degrees of freedom are introduced automatically at the element level and condensed out during the solution phase of the analysis.

Because of the extra internal degrees of freedom and static condensation, the enhanced strain formulation option is less efficient than either the $\bar{B}$ method (selective reduced integration) option or the uniform reduced integration option.

This technology can be used with mixed u-P formulations.

5.1.2.1.3. Simplified Enhanced Strain Formulation

The simplified enhanced strain formulation prevents shear locking in bending-dominated problems. This formulation is similar to the enhanced strain formulation, except that it introduces only the internal degrees of freedom necessary to prevent shear locking. For the plane stress state, this formulation is identical to the enhanced strain formulation.

Because there are no internal degrees of freedom to handle volumetric locking, this formulation is not ideal when the material is nearly incompressible, except when also using the mixed u-P formulation. When used with mixed u-P, the simplified enhanced strain formulation yields the same results as the enhanced strain formulation. All internal degrees of freedom are introduced automatically at the element level and condensed out during the solution phase of the analysis.

5.1.2.1.4. Uniform Reduced Integration

The uniform reduced integration method helps to prevent shearing locking in bending-dominated problems and volumetric locking in nearly incompressible cases. Because it has only one integration point, this option is more efficient than the $\bar{B}$ method (selective reduced integration) and the enhanced strain formulations. Because the artificial energy introduced to control the hourglass effect can diminish solution accuracy, however, consider this option last.

When using uniform reduced integration, verify the solution accuracy by comparing the total energy (SENE label on the **ETABLE** command) and the artificial energy (AENE label on the **ETABLE**) introduced by hourglass control. If the ratio of artificial energy to total energy is less than five percent, the solution is generally acceptable. If the ratio exceeds five percent, refine the mesh. You can also monitor the total energy and artificial energy during the solution phase of the analysis (**OUTPR**,VENG).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.1.2.2. Technologies for Lower-Order Tetrahedron Elements

Linear tetrahedron elements use mixed u-P formulation with pressure stabilization. The program achieves stabilization by enhancing the strain field with three internal degrees of freedom (which are condensed out at the element level). The stabilization allows for a robust and accurate element.

5.1.2.3. Technologies for Higher-Order Quad or Brick Elements

Higher-order elements (actually second-order here) do not have shear locking due to the quadratic shape functions, but they are prone to volumetric locking if the full-integration scheme is used. To avoid this problem, higher-order elements use uniform reduced integration as the default or only option.

The hourglass effect is not an issue if the model has at least two elements for 2-D problems, or at least two elements in each direction for 3-D problems, as it is difficult for hourglass mode to propagate beyond one element. Hourglass control is therefore not needed.

Uniform reduced integration can be used with mixed u-P formulation.

The full integration option is used primarily for linear analysis, and for nonlinear analysis, uniform reduced integration is the better choice. If full integration is necessary for an analysis involving nonlinear material, use it with mixed u-P formulation to limit the risk of volumetric locking.

5.1.2.4. Technologies for Higher-Order Triangle or Tetrahedron Elements

The second-order elements do not have shear locking, and volumetric locking is generally not an issue. If you observe volumetric locking, used mixed u-P formulation.

5.1.3. Mixed u-P Formulation

Mixed u-P elements use both displacement and hydrostatic pressure as primary unknown variables. Mixed u-P formulation is available in SOLID285, and in most other current-technology elements where KEYOPT(6) > 0, such as: PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLSH190, SOLID272, and SOLID273.

The following related topics are available:

5.1.3.1. Overview of Mixed u-P Element Technologies
5.1.3.2. Mixed u-P Element Summary
5.1.3.3. Applications of Mixed u-P Formulations
5.1.3.4. Mixed u-P Models and Overconstraint or No Unique Solution

5.1.3.1. Overview of Mixed u-P Element Technologies

Incompressible material behavior may lead to difficulties in numerical simulation, such as volumetric locking, inaccuracy of solution, checkerboard pattern of stress distributions, or, occasionally, divergence. Mixed u-P elements are intended to overcome these problems.

Lagrange multiplier based mixed u-P elements (current-technology solid elements with KEYOPT(6) > 0 and SOLID285) can also be used to overcome incompressible material problems. They are designed to model material behavior with high incompressibility such as fully or nearly incompressible hyperelastic materials and nearly incompressible elastoplastic materials (high Poisson ratio or undergoing large plastic strain). Lagrange multipliers extend the internal virtual work so that the volume constraint equation is included explicitly. This introduces hydrostatic pressure (or volume change rate for nearly incompressible hyperelastic materials) as a new independent variable. Unlike the hyperelastic elements,

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the hydrostatic pressure variables are not condensed on the element level, but are solved at the global level. See the *Mechanical APDL Theory Reference* for further details.

The mixed u-P formulation of the current-technology solid elements offers options for handling incompressible material behavior. You can combine the mixed u-P formulation with other element technologies such as the $\bar{B}$ method (also known as the selective reduced integration method) and the uniform reduced integration method. With current-technology elements, you can combine the mixed u-P formulation with the enhanced strain formulation method. Furthermore, the mixed u-P formulation is associated with hyperelastic models, such as Mooney-Rivlin, Neo-Hookean, Ogden, Arruda-Boyce, polynomial form, and user-defined.

5.1.3.2. Mixed u-P Element Summary

The number of independent hydrostatic pressure degrees of freedom depends on the element type, element technology, and the value of KEYOPT(6), as shown in Table 5.1: Number of Independent Pressure Degrees of Freedom in One Element (p. 72).

Table 5.1: Number of Independent Pressure Degrees of Freedom in One Element

Element	Basic Element Technology	KEYOPT(6)	Number of Pressure DOFs	Interpolation Function
PLANE182	$\bar{B}$ method (selective reduced integration) *or* uniform reduced integration	1	1	Constant
PLANE182	Enhanced strain formulation	1	3	Linear
PLANE183	Uniform reduced integration (KEYOPT(1) = 0)	1	3	Linear
PLANE183	3-point integration (KEYOPT(1) = 1)	1	3	Linear
SOLID185	$\bar{B}$ method (selective reduced integration) *or* uniform reduced integration	1	1	Constant
SOLID185	Enhanced strain formulation	1	4	Linear
SOLID186	Uniform reduced integration	1	4	Linear
SOLID186	Full integration	1	4	Linear
SOLID187	4-point integration	1	1	Constant
SOLID187	4-point integration	2	4	Linear
SOLSH190	Enhanced strain formulation	1	4	Linear
SOLID272	Full integration in the r-z plane.	1	KEYOPT(2)	Constant in the r-z plane and Fourier interpolation in the circumferential (θ) direction
SOLID273	Uniform integration in the r-z plane.	1	KEYOPT(2) x 3	Linear in the r-z plane and Fourier inter-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Element	Basic Element Technology	KEYOPT(6)	Number of Pressure DOFs	Interpolation Function
				polation in the circumferential (θ) direction
SOLID285	Stabilized pressure formulation	---	4	Linear

The hydrostatic pressure has an interpolation function one order lower than the one for volumetric strain. Therefore, elastic strain only agrees with stress in an element on the average instead of pointwise.

For current-technology mixed u-P elements, the volume constraint equation is checked for each element of a nonlinear analysis. The number of elements in which the constraint equation is not satisfied is reported in the output file. The default tolerance for the volumetric compatibility or volume constraint check is 1.0×10^{-5}. You can change this value if necessary (**SOLCONTROL**).

For the SOLID285 element, hydrostatic pressure is added as a degree of freedom at each node. The incompressibility is therefore checked at each node (similar to forces). For more information, see the documentation for the **CNVTOL** command.

The mixed u-P formulation is not needed in plane stress; the incompressibility condition is assumed, and the thickness is adjusted based on this incompressible assumption. For 2-D elements PLANE182 and PLANE183, using the mixed u-P formulation with either of the plane stress options (KEYOPT(3) = 0 or 3), the program resets any KEYOPT(6) setting to 0 automatically for pure displacement formulation.

5.1.3.3. Applications of Mixed u-P Formulations

Incompressible material behavior can be divided into two categories: fully incompressible materials and nearly incompressible materials. Typical fully incompressible materials are hyperelastic materials. You must use a current-technology element with mixed u-P formulation or SOLID285 to model this material behavior. For element SOLID187, set KEYOPT(6) = 1.

Nearly incompressible materials include hyperelastic materials and elastoplastic materials. The current-technology elements with mixed u-P formulation and SOLID285 are available for nearly incompressible hyperelastic materials.

The best element choice varies from problem to problem, as indicated in these general guidelines:

- **For material behavior with very small compressibility:** Use a current-technology element with mixed u-P formulation or SOLID285.
- **For moderate compressibility:** Use PLANE182/SOLID185 with $\bar{B}$ (most efficient), a current-technology element with mixed u-P formulation, or SOLID285.
- **When deformation is highly confined:** Use a current-technology element with mixed u-P formulation.

Nearly incompressible elastoplastic materials are materials with Poisson's ratio close to 0.5, or elastoplastic materials undergoing very large plastic deformation. For such cases, especially when the deformation is close to being fully incompressible, the mixed u-P formulation of the current-technology elements

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and SOLID285 is more robust, yielding better performance. Try pure displacement formulation (KEYOPT(6) = 0) first, however, because the extra pressure degrees of freedom are involved in mixed formulation. If you are using mixed formulation with element SOLID187, it is good practice to specify hydrostatic pressure interpolation (KEYOPT(6) = 2).

5.1.3.4. Mixed u-P Models and Overconstraint or No Unique Solution

Avoid overconstrained models when using current-technology elements with mixed u-P formulation. An overconstrained model means that globally or locally, the number of displacement degrees of freedom without any prescribed values (*Nd*) is less than the number of hydrostatic pressure degrees of freedom (*Np*). *Np* can be calculated by the number of elements times the number of pressure degrees of freedom within each element. (See Table 5.1: Number of Independent Pressure Degrees of Freedom in One Element (p. 72) for the specific numbers used with the current-technology elements.) If different type elements are included, *Np* is the summation of the products over each type of mixed formulation element. The overconstrained problem can be overcome by increasing the number of nodes without any displacement boundary conditions, or by refining the mesh. The optimal values for *Np* and *Nd* are *Nd*/*Np* = 2 for 2-D problems and *Nd*/*Np* = 3 for 3-D problems.

On the other hand, SOLID285 is not likely to be overconstrained. At each node, it has one pressure degree of freedom and three degrees of freedom (UX, UY, and UZ). These are the best values for *Nd*/*Np*.

When the mixed u-P formulation of the current-technology plane and solid elements is applied to fully incompressible hyperelastic materials, you must also avoid the “no unique solution” situation. No unique solution occurs if all boundary nodes have prescribed displacements in each direction and the material is fully incompressible. Since hydrostatic pressure is independent from deformation, it should be determined by the force/pressure boundary condition. If no force/pressure boundary condition is applied, the hydrostatic pressure cannot be determined, or the solution is not unique. In this case, any stress field in equilibrium would be still in equilibrium and not cause any further deformation by adding an arbitrary value of hydrostatic pressure. This problem can be solved simply by having at least one node on the boundary in at least one direction without a prescribed displacement boundary condition. This direction cannot be the tangential direction of the boundary at this node. That means the solution is the particular one with zero force at the node in that direction where no displacement is prescribed.

5.1.4. Generalized Plane Strain Options

The generalized plane strain feature assumes a finite deformation domain length in the Z direction, rather than the infinite value assumed for standard plane strain. Generalized plane strain, therefore, gives more practical results for deformation problems where the Z-direction dimension is not long enough. It also offers a more efficient way to simulate certain 3-D deformations using 2-D element options.

The deformation domain or structure is formed by extruding a plane area along a curve with a constant curvature, as shown in the following figure:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.1: Generalized Plane Strain Deformation

The extrusion begins at the starting (or reference) plane and stops at the ending plane. The curve direction along the extrusion path is called the fiber direction. The starting and ending planes must be perpendicular to this fiber direction at the beginning and ending intersections.

If the boundary conditions and loads in the fiber direction do not change over the course of the curve, and if the starting plane and ending plane remain perpendicular to the fiber direction during deformation, then the amount of deformation of all cross sections is identical throughout the curve, and does not vary at any curve position in the fiber direction. Any deformation, therefore, can be represented by the deformation on the starting plane, and the 3-D deformation can be simulated by solving the deformation problem on the starting plane. The existing plane strain and axisymmetric options are particular cases of the generalized plane strain option.

All inputs and outputs are in the global Cartesian coordinate system. The starting plane must be the X-Y plane, and must be meshed. The applied nodal force on the starting plane is the total force along the fiber length. The geometry in the fiber direction is specified by the rotation about X and Y of the ending plane and the fiber length passing through a point that you specify on the starting plane, called the starting (or reference) point. The starting point creates an ending point on the ending plane through the extrusion process. The boundary conditions and loads in the fiber direction are specified by applying displacements or forces at the ending point. This ending point can be different from regular nodes, in that it is designated by the same X-Y coordinates that are fixed in plane during deformation.

The generalized plane strain option introduces three new degrees of freedom for each element. Two internal nodes are created automatically at the solution stage for the generalized plane strain option to carry the extra three degrees of freedom. You can apply boundary conditions and loads and check the results of the fiber length and rotation angle changes, and reaction forces, using the commands **GSBDATA**, **GSGDATA**, **GSSOL**, and **GSLIST**. The results of the fiber length change, rotation angle change, and reaction forces can also be viewed through **OUTPR**.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The fiber length change is positive when the fiber length increases. The sign of the rotation angle or angle change is determined by how the fiber length changes when the coordinates of the ending point change. If the fiber length decreases when the X coordinate of the ending point increases, the rotation angle about Y is positive. If the fiber length increases when the Y coordinate of the ending point increases, the rotation angle about X is positive.

In Eigenvalue analyses, such as Eigen buckling and modal analysis, the generalized plane strain option usually reports fewer Eigenvalues and Eigenvectors than you would obtain in a 3-D analysis. Because it reports only homogeneous deformation in the fiber direction, generalized plane strain employs only three degrees of freedom to account for these deformations. The same 3-D analysis would incorporate many more degrees of freedom in the fiber direction.

Because the mass matrix terms relating to degrees of freedom in the fiber direction are approximated for modal and transient analyses, you cannot use the lumped mass matrix for these types of analyses, and the solution may be slightly different from regular 3-D simulations when any of the three designated degrees of freedom is not restrained.

5.1.5. Axisymmetric Elements with Nonaxisymmetric Loads

Axisymmetric problems should be modeled using axisymmetric elements. An axisymmetric model greatly reduces the modeling and analysis time compared to that of an equivalent 3-D model.

If loading is also axisymmetric, the analysis is relatively straightforward. However, loading often varies around the circumference. Two methods are available for handling nonaxisymmetric loading:

- General Axisymmetric Elements -- Support any type of material or loading
- Harmonic Axisymmetric Elements -- Restricted to linear materials and sinusoidal loading.

5.1.5.1. General Axisymmetric Elements

Different from harmonic axisymmetric elements, general axisymmetric elements (SOLID272, SOLID273, and SURF159) introduce the Fourier series into the interpolation functions to describe the change of displacements in the circumferential (θ) direction. The elements can therefore apply to any analysis type, including geometric nonlinear analyses, and can support any load and deformation mode. For more information, see SOLID272 - General Axisymmetric Solid with 4 Base Nodes and SOLID273 - General Axisymmetric Solid with 8 Base Nodes in the *Mechanical APDL Theory Reference*.

The elements can have any axis as the axisymmetric axis (defined via **SECTYPE** and **SECDATA** commands). You need only define base elements on a flat plane.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Axisymmetric Element Terminology

The plane on which quadrilaterals or triangles are defined is called the *master plane*. The quadrilaterals, triangles, or lines on the master plane are called *base elements*, and the nodes of the quadrilaterals, triangles, and lines are called *base nodes*.

A *nodal plane* (copied from the master plane) is a radial plane of Fourier nodes. There are n nodal planes (where KEYOPT(2) = n), equally spaced around the circumference.

An *integration plane* is a plane which sits equally between any two adjacent nodal planes. Integration points in the circumferential direction are located on both the nodal and integration planes. For more information, see SOLID272 - General Axisymmetric Solid with 4 Base Nodes in the *Mechanical APDL Theory Reference*.

The **NAXIS** command automatically creates a full 3-D finite-element model using the base elements, the axisymmetric axis, and the number of Fourier nodes specified via KEYOPT(2). The axisymmetric axis must be on the master plane, and the base elements must be on one side of the axis. A base element or base node must be associated with one axisymmetric axis *before* issuing the **NAXIS** command. All generated nodes are equally distributed circumferentially.

The default element coordinate system is a cylindrical coordinate system having the axisymmetric axis as z, the origin defined via the **SECDATA** command, and $\theta = 0$ at the master plane. The right-hand rule determines the direction of θ, resulting in x(r), y(θ), and z from the right-hand coordinate system, as shown:

Figure 5.2: General Axisymmetric Elements and Their Coordinate Systems (KEYOPT(2) = 3)

The displacements in the r, θ, and z directions, however, are interpolated by

$$u_i = h_i(s,t)(c_1 + a_1\cos\theta + b_1\sin\theta + a_2\cos2\theta + b_2\sin2\theta + a_3\cos3\theta + b_3\sin3\theta \ldots)$$

where

$i = r, \theta, z$

and

$h_i(s,t)$ is the standard interpolation function for 2-D elements.

The nodes can have their own coordinate systems defined in any direction, as is the case with the nodes of any other solid element. For more detailed information, see General Axisymmetric Solids in the *Mechanical APDL Theory Reference*.

General axisymmetric elements differ from harmonic axisymmetric elements in that you do not specify the symmetric and unsymmetric terms, input peak values of loads, and solve for each term as one load step. Instead, you apply all physical loads (forces and displacements) at the nodes and solve the problem only once to obtain the solution.

The order of $h_i(s,t)$ is determined by the element type. (SOLID272 is a lower-order element and SOLID273 is a higher-order element.) The order of Fourier terms is determined by the number of Fourier nodes, or the number of nodes created in the circumferential direction for one base node (including the base node itself). This number is also the number of nodal planes in the circumferential direction, which you input via the element's KEYOPT(2). The KEYOPT(2) setting, the number of Fourier nodes, and the Fourier terms are shown in Table 5.2: Fourier Terms for General Axisymmetric Elements (p. 78). Generally, each Fourier term has different coefficients except when KEYOPT(2) is an even number (in which case the highest terms of the sine and cosine have the same coefficient).

Table 5.2: Fourier Terms for General Axisymmetric Elements

KEYOPT(2) Value	Number of Fourier Nodes (and Nodal Planes)	Fourier Terms	Degrees Between Nodal Planes
1	1	c_1	360
3	3	c_1, $a_1\cos\theta$, $b_1\sin\theta$	120
4	4	c_1, $a_1\cos\theta$, $b_1\sin\theta$, $a_2\cos2\theta$, $a_2\sin2\theta$	90
5	5	c_1, $a_1\cos\theta$, $b_1\sin\theta$, $a_2\cos2\theta$, $b_2\sin2\theta$	72
6	6	c_1, $a_1\cos\theta$, $b_1\sin\theta$, $a_2\cos2\theta$, $b_2\sin2\theta$, $a_3\cos3\theta$, $a_3\sin3\theta$	60
7	7	c_1, $a_1\cos\theta$, $b_1\sin\theta$, $a_2\cos2\theta$, $b_2\sin2\theta$, $a_3\cos3\theta$, $b_3\sin3\theta$	51.43
8	8	c_1, $a_1\cos\theta$, $b_1\sin\theta$, ... , $a_4\cos4\theta$, $a_4\sin4\theta$	45
9	9	c_1, $a_1\cos\theta$, $b_1\sin\theta$, ... , $a_4\cos4\theta$, $b_4\sin4\theta$	40
10	10	c_1, $a_1\cos\theta$, $b_1\sin\theta$, ... , $a_5\cos5\theta$, $a_5\sin5\theta$	36
11	11	c_1, $a_1\cos\theta$, $b_1\sin\theta$, ... , $a_5\cos5\theta$, $b_5\sin5\theta$	32.73
12	12	c_1, $a_1\cos\theta$, $b_1\sin\theta$, ... , $a_6\cos6\theta$, $a_6\sin6\theta$	30

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For KEYOPT(2) = 1, the deformation is also axisymmetric but, unlike the axisymmetric option of 2-D elements such as PLANE182, the general axisymmetric elements allow torsion. If no torsion load exists, it is more efficient to use the axisymmetric elements.

To simulate unsymmetric deformation, set KEYOPT(2) > 1. The greater the number of Fourier nodes, the more accurate the simulation of complicated deformation modes.

Solve localized deformation problems with a greater number of Fourier nodes (higher-order Fourier terms). Be aware that the deformation is not as localized as it would be when using standard 3-D solid elements; with general axisymmetric elements, displacements are interpolated using Fourier terms in the circumferential direction (rather than being interpolated piecewise using linear/quadratic functions as in 3-D solid elements).

Contact is usually a local phenomenon. Generally, higher-order Fourier terms are necessary.

The maximum allowed number of Fourier nodes is 12. If more than 12 nodes are necessary, a general axisymmetric element may not be computationally efficient; therefore, it is better to use a standard 3-D solid element for such cases.

5.1.5.1.1. Example: 3-D General Axisymmetric Model

The following example input file shows how to use commands and the SOLID272 element to generate a 3-D general axisymmetric model. The model represents a simple thread connection made of two parts, a bolt and a nut, both modeled with 12 Fourier nodes (KEYOPT(2) = 12) and the global Y axis as the axis of symmetry.

```
/batch,list
/title, Sample input for generating general axisymmetric solid elements
/prep7
!
et,1,272,,12       !define general axisymmetric SOLID272 with 12 nodes along circumferential direction
et,2,272,,12
!
sect,1,axis        !define general axisymmetric section
secd,2,0,2         !use pattern 2 with global cartesian coordinate system (0)and y as axis of symmetry (2)
!
mp,ex,1,2.0e5      !define elastic material properties
mp,nuxy,1,0.3
!
!
! Create  2D  geometry of thread connection (bolt/nut)
!
k,1,1.0,0
k,2,4.0,0
k,3,4,1.5
k,4,5.0,2.5
k,5,5.0,3.5
k,6,4.0,4.5
k,7,4.0,5.5
k,8,5.0,6.5
k,9,5.0,7.5
k,10,4.0,8.5
k,11,4.0,10
k,12,1.0,10
k,13,7.0,0
k,14,7.0,10
!
k,15,4.0,0
k,16,4,1.5
k,17,5.0,2.5
k,18,5.0,3.5
k,19,4.0,4.5
k,20,4.0,5.5
k,21,5.0,6.5
```

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
k,22,5.0,7.5
k,23,4.0,8.5
k,24,4.0,10
!
a,1,2,3,4,5,6,7,8,9,10,11,12
a,15,16,17,18,19,20,21,22,23,24,14,13
!
!
!Mesh 2D geometry
!
esize,0.5
type,1             !use type 1 for bolt component
mat,1
secn,1             !use general axisymmetric section
amesh,1            !generate the master plane nodes and elements for bolt component
!
esize,1            !use different mesh
type,2             !use type 2 for nut component
mat,1
secn,1             !use general axisymmetric section
amesh,2            !generate the master plane nodes and elements for nut component
!
/PNUM,TYPE,1       !display element type numbers
/NUMBER,1          !numbering shown with colors only
eplot              !visualise the 2D model
!
!
!Generate full 3D axisymmetric model of bolt and nut using 12 planes
!
naxis
!
/view,1,1,1,1      !isometric view
!
nplot              !display the 12 nodal planes along circumferential direction
!
/eshape,1          !display elements with general axisymmetric shape
!
esel,s,type,,1     !display 3D axisymmetric model of bolt
eplot
!
esel,s,type,,2     !display 3D axisymmetric model of nut
eplot
!
esel,all
!
eplot              !display full 3D axisymmetric model of bolt and nut
!
fini
```

The input listing generates the following 2-D and 3-D meshes:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The general axisymmetric elements representing the bolt and nut are shown separately in the figures below:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.1.5.2. Harmonic Axisymmetric Elements

An axisymmetric structure can be represented by a plane (X,Y) finite-element model.

ANSYS, Inc. recommends general axisymmetric elements SOLID272 and SOLID273 for such applications because they can accept any type of load and can support nonlinear analyses. However, you can also use a special class of axisymmetric elements called *harmonic elements*: PLANE25, SHELL61, PLANE75, PLANE78, and PLANE83. The harmonic elements allow a nonaxisymmetric load and support linear analyses.

Axisymmetric elements are modeled on a 360° basis, so all input and output nodal heat flows, forces, moments, fluid flows, current flows, electric charges, magnetic fluxes, and magnetic current segments must be input in this manner. Similarly, input real constants representing volumes, convection areas, thermal capacitances, heat generations, spring constants, and damping coefficients must also be input in on a 360° basis.

Unless otherwise stated, the model must be defined in the Z = 0.0 plane. The global Cartesian Y axis is assumed to be the axis of symmetry. Further, the model is developed only in the +X quadrants. Hence, the radial direction is in the +X direction.

The boundary conditions are described in terms of the structural elements. The forces (FX, FY, etc.) and displacements (UX, UY, etc.) for the structural elements are input and output in the nodal coordinate system. All nodes along the y axis centerline (at x = 0.0) should have the radial displacements (UX if not rotated) specified as zero, unless a pinhole effect is desired. At least one value of UY should be specified or constrained to prevent rigid body motions. Torsion, while axisymmetric, is available only for a few element types. If an element type allows torsion, all UZ degrees of freedom should be set to 0.0 on the centerline, and one node with a positive X coordinate must also have a specified or constrained

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

value of UZ. Pressures and temperatures may be applied directly. Acceleration, if any, is usually input only in the axial (Y) direction. Similarly, angular velocity, if any, is usually input only about the Y axis.

For more information, see Harmonic Axisymmetric Elements with Nonaxisymmetric Loads (p. 84).

5.1.5.2.1. Harmonic Axisymmetric Elements with Nonaxisymmetric Loads

An axisymmetric structure (defined with the axial direction along the global Y axis and the radial direction parallel to the global X axis) can be represented by a plane (X, Y) finite-element model. The use of an axisymmetric model greatly reduces the modeling and analysis time compared to that of an equivalent 3-D model. The harmonic axisymmetric elements allow nonaxisymmetric loads. For these elements (PLANE25, SHELL61, PLANE75, PLANE78, and PLANE83) , the load is defined as a series of harmonic functions (Fourier series). For example, a load F is given by:

$$F(\theta) = A_0 + A_1 \cos\theta + B_1 \sin\theta + A_2 \cos 2\theta + B_2 \sin 2\theta + A_3 \cos 3\theta + B_3 \sin 3\theta + ...$$

Each term of the above series must be defined as a separate load step. A term is defined by the load coefficient (A^{ℓ} or B^{ℓ}), the number of harmonic waves (ℓ), and the symmetry condition ($\cos^{\ell}\theta$ or $\sin^{\ell}\theta$). The number of harmonic waves, or the mode number, is input with the **MODE** command. Note that $\ell = 0$ represents the axisymmetric term (A_0). θ is the circumferential coordinate implied in the model. The load coefficient is determined from the standard boundary condition input (i.e., displacements, forces, pressures, etc.). Input values for temperature, displacement, and pressure should be the peak value. The input value for force and heat flow should be a number equal to the peak value per unit length times the circumference. The symmetry condition is determined from the *ISYM* value also input on the **MODE** command. The description of the element given in Element Library (p. 115) and in the appropriate sections of the *Mechanical APDL Theory Reference* should be reviewed to see which deformation shape corresponds to the symmetry conditions.

Results of the analysis are written to the results file. The deflections and stresses are output at the peak value of the sinusoidal function. The results may be scaled and summed at various circumferential (θ) locations with POST1. This may be done by storing results data at the desired θ location using the *ANGLE* argument of the **SET** command. A load case may be defined with **LCWRITE**. Repeat for each set of results, then combine or scale the load cases as desired with **LCOPER**. Stress (and temperature) contour displays and distorted shape displays of the combined results can also be made.

Caution should be used if the harmonic elements are mixed with other, nonharmonic elements. The harmonic elements should not be used in nonlinear analyses, such as large deflection and/or contact analyses.

The element matrices for harmonic elements are dependent upon the number of harmonic waves *(MODE*) and the symmetry condition (*ISYM*). For this reason, neither the element matrices nor the triangularized matrix is reused in succeeding substeps if the *MODE* and *ISYM* parameters are changed. In addition, a superelement generated with particular *MODE* and *ISYM* values must have the same values in the "use" pass.

For stress stiffened (prestressed) structures, only the stress state of the most recent previous *MODE* = 0 load case is used, regardless of the current value of *MODE*.

Loading Cases - The following cases are provided to aid the user in obtaining a physical understanding of the *MODE* parameter and the symmetric (*ISYM*=1) and antisymmetric (*ISYM*=-1) loading conditions. The loading cases are described in terms of the structural elements. The forces (FX, FY, etc.) and displacements (UX, UY, etc.) for the structural elements are input and output in the nodal coordinate system. In all cases illustrated, it is assumed that the nodal coordinate system is parallel to the global Cartesian

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

coordinate system. The loading description may be extended to any number of modes. The harmonic thermal elements (PLANE75 and PLANE78) are treated the same as PLANE25 and PLANE83, respectively, with the following substitutions: UY to TEMP, and FY to HEAT. The effects of UX, UZ, ROTZ, FX, FZ and MZ are all ignored for thermal elements.

Case A: (*MODE* = 0, *ISYM* not used) - This is the case of axisymmetric loading except that torsional effects are included. Figure 5.3: Axisymmetric Radial, Axial, Torsion and Moment Loadings (p. 85) shows the various axisymmetric loadings. Pressures and temperatures may be applied directly. Acceleration, if any, is usually input only in the axial (Y) direction. Similarly, angular velocity, if any, is usually input only about the Y axis.

Figure 5.3: Axisymmetric Radial, Axial, Torsion and Moment Loadings

The total force (F) acting in the axial direction due to an axial input force (FY) is:

$$F = \int_0^{2\pi} (\text{force per unit length})*(\text{increment length})$$

$$F = \int_0^{2\pi} (FY / 2\pi R) * (Rd\theta) = FY$$

where FY is on a full 360° basis.

The total applied moment (M) due to a tangential input force (FZ) acting about the global axis is:

$$M = \int_0^{2\pi} (\text{force per unit length})*(\text{lever arm})*(\text{increment length})$$

$$M = \int_0^{2\pi} (-FZ / 2\pi R) * (R) * (Rd\theta) = -R * FZ$$

where FZ is on a full 360° basis. Calculated reaction forces are also on a full 360° basis and the above expressions may be used to find the total force. Nodes at the centerline (X = 0.0) should have UX and UZ (and ROTZ, for SHELL61) specified as zero, unless a pinhole effect is desired. At least one value of UY should be specified or constrained to prevent rigid body motions. Also, one node with a nonzero, positive X coordinate must have a specified or constrained value of UZ if applicable. When Case A defines the stress state used in stress stiffened analyses, torsional stress is not allowed.

Case B: (*MODE* = 1, *ISYM*=1) - An example of this case is the bending of a pipe. Figure 5.4: Bending and Shear Loading (ISYM = 1) (p. 86) shows the corresponding forces or displacements on a nodal circle. All functions are based on sin θ or cos θ. The input and output values of UX, FX, etc., represent the peak values of the displacements or forces. The peak values of UX, UY, FX and FY (and ROTZ and MZ for SHELL61) occur at $\theta = 0°$, whereas the peak values of UZ and FZ occur at $\theta = 90°$. Pressures and temperatures are applied directly as their peak values at $\theta = 0°$. The thermal load vector is computed from T_{peak}, where T_{peak} is the input element or nodal temperature. The reference temperature for thermal strain calculations (**TREF**) is internally set to zero in the thermal strain calculation for the harmonic ele-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ments if *MODE* > 0. Gravity (g) acting in the global X direction should be input (**ACEL**) as *ACELX* = g, *ACELY* = 0.0, and *ACELZ* = -g. The peak values of σ_x, σ_y, σ_z and σ_{xy} occur at $\theta = 0°$, whereas the peak values of σ_{yz} and σ_{xz} occur at 90 °.

Figure 5.4: Bending and Shear Loading (ISYM = 1)

The total applied force in the global X direction (F) due to both an input radial force (FX) and a tangential force (FZ) is:

$$F = \int_0^{2\pi} (\text{force per unit length})*(\text{directional cosine})*(\text{increment length})$$

$$F = \int_0^{2\pi} ((FX(\cos\theta)/2\pi R)*(\cos\theta) + (FZ(\sin\theta/2\pi R)*(-\sin\theta))*(Rd\theta)$$

$$F = (FX - FZ)/2$$

where FX and FZ are the peak forces on a full 360° basis. Calculated reaction forces are also the peak values on a full 360° basis and the above expression may be used to find the total force. These net forces are independent of radius so that they may be applied at any radius (including X = 0.0) for the same net effect.

An applied moment (M) due to an axial input force (FY) for this case can be computed as follows:

$$M = \int_0^{2\pi} (\text{force per unit length})*(\text{lever arm})*(\text{increment length})$$

$$M = \int_0^{2\pi} (FY(\cos\theta)/2\,\pi R)*(R\cos\theta)*(Rd\theta) = (FY)\,R/2$$

An additional applied moment (M) is generated based on the input moment (MZ):

$$M = \int_0^{2\pi} (\text{force per unit length})*(\text{directional cosine})*(\text{increment length})$$

$$M = \int_0^{2\pi} (MZ(\cos\theta)/2vR)*(\cos\theta)(Rd\theta) = (MZ)/2$$

If it is desired to impose a uniform lateral displacement (or force) on the cross section of a cylindrical structure in the global X direction, equal magnitudes of UX and UZ (or FX and FZ) may be combined as shown in Figure 5.5: Uniform Lateral Loadings (p. 87).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.5: Uniform Lateral Loadings

When UX and UZ are input in this manner, the nodal circle moves in an uniform manner. When FX and FZ are input in this manner, a uniform load is applied about the circumference, but the resulting UX and UZ are not, in general, of the same magnitude. If it is necessary to have the nodal circle moving in a rigid manner, it can be done via constraint equations (**CE**) so that UX = -UZ.

Node points on the centerline (X = 0.0) should have UY specified as zero. Further, UX must equal -UZ at all points along the centerline, which may be enforced with constraint equations. To prevent rigid body motions, at least one value of UX or UZ, as well as one value of UY (not at the centerline), or ROTZ, should be specified or constrained in some manner. For SHELL61, if plane sections (Y = constant) are to remain plane, ROTZ should be related to UY by means of constraint equations at the loaded nodes.

Case C: (*MODE* = 1, *ISYM* = -1) - This case (shown in Figure 5.6: Bending and Shear Loading (*ISYM* = -1) (p. 87)) represents a pipe bending in a direction 90° to that described in Case B.

Figure 5.6: Bending and Shear Loading (*ISYM* = -1)

The same description applying to Case B applies also to Case C, except that the negative signs on UZ, FZ, and the direction cosine are changed to positive signs. Also, the location of the peak values of various quantities are switched between the 0° and 90° locations.

Case D: (*MODE* = 2, *ISYM* = 1) - The displacement and force loadings associated with this case are shown in Figure 5.7: Displacement and Force Loading Associated with *MODE* = 2 and *ISYM* = 1 (p. 87). All functions are based on sin 2 θ and cos 2 θ.

Figure 5.7: Displacement and Force Loading Associated with *MODE* = 2 and *ISYM* = 1

Additional Cases: There is no programmed limit to the value of *MODE*. Additional cases can be defined.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.2. Shell Elements

Use shell elements to model thin structures (where one dimension is much smaller than the other two dimensions). Shell elements decouple the deformation on the surface and the deformation in the normal direction, allowing for a simple and efficient simulation of a thin structure.

No firm rules exist to indicate when it is best to use shell elements. If the structure acts as a shell, you can use shell elements.

Shell elements are usually created by meshing a surface representing the position and shape of the structure, and associating them with section data to represent shell thickness and properties.

The following additional topics concerning shell elements are available:

5.2.1. Axisymmetric Shells
5.2.2. Shell Stress and Stiffness
5.2.3. Membrane and Surface Stress Options
5.2.4. Shell Shear and Warping
5.2.5. Curved Shell Elements
5.2.6. Shell Element Coordinate System
5.2.7. Shell Element Nodes

5.2.1. Axisymmetric Shells

In addition to 3-D shell elements, axisymmetric shell elements are available for efficiently modeling axisymmetric shell structures in 2-D.

Axisymmetric shell elements, such as SHELL208 and SHELL209, may include out-of-plane translational degrees of freedom to model uniform torsion about the axis.

5.2.2. Shell Stress and Stiffness

Only plane stress state is allowed in shell elements. Normal thickness stress may be recovered in some shell elements; however, recovered normal thickness stress is a postprocessing quantity and does not contribute to the total element strain energy.

Shell elements withhold external load with membrane, bending, and transverse shear stiffness. The in-plane rotational (drill) stiffness is added at the nodes for solution stability, as shell elements do not have a true in-plane rotational stiffness; consequently, do not expect the in-plane rotational stiffness to carry a load. For lower-order shells with uniform reduced integration, a small artificial stiffness may be included for suppressing the hourglass modes.

5.2.3. Membrane and Surface Stress Options

Some shell elements have an option allowing them to be used as membrane elements. Such elements allow deformation in the plane of the surface only (that is, stresses do not vary through the thickness). Only membrane stiffness is accounted for. Shell bending, transverse shear, and drill stiffness are excluded; therefore, only translational degrees of freedom are retained. Membrane elements are suitable for modeling extremely thin shells or composite shells with small bending stiffness. Specifying the membrane option for these structures can generally avoid ill-conditioned systems; however, because membrane elements are capable of multiple free-energy modes, using those elements alone is not recommended.

A surface-stress option is available for some shell elements, which excludes all element stiffness and mass contribution. The option allows shell elements to serve as a strain gauge for precision measurement

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of stresses, strains, and other element solution quantities at selected locations in the model. Similar to the membrane option, the surface-stress option requires only translational degrees of freedom.

5.2.4. Shell Shear and Warping

The only shear on the free surfaces of a shell element is in-plane. Normals to the shell middle surface stay straight, but not necessarily normal. As a result, the in-plane strain variation through the thickness cannot be more complex than linear.

The assumption of linear in-plane strain variation through the thickness is invalid at the edges of layered composite shell elements that have different material properties at each layer. For accurate stresses in this area, consider using submodeling.

The program does not verify that the element thickness exceeds its width (or many times its width), as such an element may be part of a fine mesh of a larger model that acts as a shell.

If the initial shape of the model is curved, then the radius/thickness ratio is important because the strain distribution through the thickness departs from linear as the ratio decreases. With the exception of SHELL61, all shell elements allow shear deformation, important for relatively thick shells.

Various shell element types tolerate a different degree of warping before their results become questionable (as described in Warping Factor in the *Mechanical APDL Theory Reference*). Four-node shell elements that do not have all of their nodes in the same plane are considered to be warped. Eight-node shell elements can accept a much greater degree of warping, but unlike other midside-node elements, their midside nodes cannot be dropped.

5.2.5. Curved Shell Elements

Some shell elements, such as SHELL181 and SHELL281, use an advanced element formulation that accurately incorporates initial curvature effects.

The calculation for effective curvature change accounts for both shell-membrane and thickness strains. The formulation generally offers excellent accuracy in curved-shell-structure simulations, especially when thickness strain is significant or the material anisotropy in the thickness direction cannot be ignored.

5.2.6. Shell Element Coordinate System

The element coordinate system for all shell elements has the z axis normal to the plane. The element x axis is in the plane, with its orientation determined by one of the following:

- The **ESYS** command
- Side I-J of the element
- Real constants

5.2.7. Shell Element Nodes

Nodes are normally located on the center plane of the element. You can offset nodes from the center plane using either of the following methods:

- The **SECOFFSET** command
- A rigid link (MPC184) that connects a middle surface node to an out-of-plane node.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use node offsets with care when modeling initially curved structures with either flat or curved elements. For curved elements, an increased mesh density in the circumferential direction may improve the results.

5.3. Beam and Link (Truss) Elements

Use beam or link (truss) elements to represent relatively long, thin pieces of structural continua (where two dimensions are much smaller than the other dimension).

Beam elements assume the direct stresses in the nonaxial direction to be zero, and ignore the deformations in the nonaxial directions (although cross sections can be scaled in a nonlinear analysis). For link elements, shear stress, stress gradients, and deformation are also ignored.

Simulating beam/truss structures is relatively simple, as elements can be created by meshing lines or curves representing the position and shape of the structure (rather than by meshing the volumes). The elements are associated with section data to represent the beam or truss cross sections.

Although beams can be used in cases where link/truss elements suffice, they are not as computationally efficient as link elements.

The available beam element types are:

- BEAM188, a 3-D two-node beam
- BEAM189, a 3-D three-node beam

Both elements are based on Timoshenko beam theory which includes shear-deformation effects. The elements can account for unrestrained warping and restrained warping of cross-sections.

Because bending resistances, element loads, or both are normally different in the y and z directions, a beam element must be oriented about its own axis. Typically, the orientation is accomplished via an *orientation node*, a node not lying on its own axis.

If bending resistance is negligible, consider using LINK180, a 3-D spar (or truss).

5.4. Pipe Elements

Use pipe elements to represent straight, slender to moderately stubby/thick pipe structures. The elements are especially useful in the offshore drilling industry, as their cross-section data includes interior fluid and outside insulation.

A pipe element is a special form of beam element. As with beams, pipe elements are created by meshing lines or curves, and the elements must be associated with section data. A pipe differs from a beam in the following ways, however:

- A pipe cross-section is normally axisymmetric so that the bending resistance is the same in both directions.

  Element orientation (via an orientation node) is still necessary, however, as the element loads can differ in the y and z directions.

- A pipe can account for both internal and external pressures.
- A pipe element has support for many nonlinear material models.

The available pipe element types are:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- PIPE288, a 3-D two-node pipe
- PIPE289, a 3-D three-node pipe

Both elements are based on Timoshenko beam theory which includes shear-deformation effects.

Also available is ELBOW290, a special 3-D three-node pipe used for modeling curved pipes. The element can account for cross-sectional distortion.

5.5. Reinforcing Elements

Reinforced materials are used extensively in civil engineering construction and composite structures. Use reinforcing elements to model reinforcement in either of two forms: cable (such as steel rebar in reinforced concrete) or fiber (such as the carbon fiber in fiber-reinforced composites).

Two methods are supported for modeling reinforcing cables or fibers:
5.5.1. Discrete Reinforcing
5.5.2. Smeared Reinforcing

For further information, see the documentation for the reinforcing elements (REINF*nnn*).

5.5.1. Discrete Reinforcing

The *discrete* reinforcing method allows reinforcing fibers to be accounted for individually. This method is suitable for modeling sparsely placed fibers, or fibers with nonuniform properties, such as cross-section area, material, spacing, and orientation.

The discrete reinforcing element REINF264 is used to add individual fibers to 3-D link, beam, shell and solid elements. Multiple reinforcing fibers are allowed in a single element.

5.5.2. Smeared Reinforcing

The *smeared* reinforcing method is suitable for modeling clusters of reinforcing fibers appearing in layer or sheet form. Each layer of fibers is simplified as a homogeneous membrane having unidirectional stiffness.

Fibers in the same layer must have uniform cross-section, material, spacing, and orientation.

The smeared reinforcing elements are:

- REINF263 -- Used with 2-D solids such as plane stress, plane strain, axisymmetric and generalized plane strain, and axisymmetric shell elements.
- REINF265 -- Used with 3-D solid and shell elements.

5.6. Surface Elements

As its name implies, a surface element (SURF*nnn*) is normally attached to the surface of a solid element. By using a surface element, you can include a number of specialized surface effects without having to apply them individually to the solid elements in your model.

Examples of surface effects include:

- Hydrostatic pressure

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Ocean loading
- Directed pressure
- Radiation
- Surface mass
- Thermal connection with FLUID116
- Surface tension
- Foundation stiffness

Not all effects are available with all surface elements.

5.7. Automatic Selection of Element Technologies and Formulations

With the variety of technologies available in many elements, choosing the best settings to solve your problem efficiently can be challenging. Particularly in the newer structural and solid elements for stress analysis, an element might offer two to four technologies.

The program can offer suggestions (or reset the technology settings) to help you achieve the best solution via the **ETCONTROL** command. The command helps you to select the appropriate element technologies by considering the stress state, the options set on the element type, and the associated constitutive models of the element type. **ETCONTROL** supports all applicable current-technology elements.

The materials associated with each element type are detected at the solution stage and are classified as:

- Elastoplastic materials, including:
  - Linear elastic materials with Poisson's ratio <= 0.49
  - Linear elastic materials with Poisson's ratio > 0.49
  - Anisotropic materials
  - Elastoplastic materials (other nonlinear materials except hyperelastic materials)
- Hyperelastic materials, including:
  - Nearly incompressible hyperelastic materials
  - Fully incompressible hyperelastic materials

In practical application, one element type could be associated with more than one constitutive model, for example, linear elastic and elastoplastic materials. For this case, the suggestions/resettings are based on elastoplastic materials, which need more complicated element technology. The list of constitutive models above is organized in order of complexity, with the bottom ones more difficult to solve numerically. When an element type is associated with multiple constitutive models, the more complex one is used in the automatic selection of the element technology.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When using **ETCONTROL**, the suggestions given or settings changed are described in the tables below.

Table 5.3: Recommendation Criteria for Element Technology (Linear Material)

Linear Materials Only			
Element	**Stress State / Option**	**Poisson's ratio (ν) <= 0.49**	**Poisson's ratio (ν) > 0.49 (or anisotropic materials)**
SHELL181	Default	KEYOPT(3) = 2 for higher accuracy of membrane stress, otherwise, KEYOPT(3) = 0; KEYOPT(8) = 2 for all cases	
	Membrane-only option (KEYOPT(1) = 1)	KEYOPT(3) = 2 for higher accuracy of stress, otherwise, KEYOPT(3) = 0; KEYOPT(8) = 0 for all cases	
SOLID272		KEYOPT(6) = 0	KEYOPT(6) = 1
SOLID273		No change. (Default element settings are best.)	
SOLID285		No change. (Default element settings are best.)	
PLANE182	Plane stress	KEYOPT(1) = 2	
	Not plane stress	KEYOPT(1) = 3	KEYOPT(1) = 2
PLANE183	Plane stress	No change. (Default element settings are best.)	
	Not plane stress	No change. (Default element settings are best.)	
SOLID185		KEYOPT(2) = 3	KEYOPT(2) = 2
SOLID186		KEYOPT(2) = 0	
SOLID187		No change. (Default element settings are best.)	
BEAM188		KEYOPT(3) = 2, KEYOPT(1) = 1[1]	
BEAM189		KEYOPT(1) = 1[1]	
SHELL208		KEYOPT(8) = 2	
SHELL209		KEYOPT(8) = 2	
SHELL281		No change. (Default element settings are best.)	
PIPE288		KEYOPT(3) = 2 KEYOPT(4) = 2 KEYOPT(15) = 1[2]	KEYOPT(3) = 2 KEYOPT(4) = 1 KEYOPT(15) = 1[2]
PIPE289		KEYOPT(4) = 2 KEYOPT(15) = 1[2]	KEYOPT(4) = 1 KEYOPT(15) = 1[2]

1. Only when the element section is not circular; otherwise, KEYOPT(1) = 0

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Only when the element cross section is composite; otherwise, KEYOPT(15) = 0

Table 5.4: Recommendation Criteria for Element Technology (Nonlinear Materials)

Nonlinear Materials			
Element	**Stress State / Option**	**Elastoplastic materials (may also have hyperelastic materials)**	**Hyperelastic materials only**
SHELL181	Default	KEYOPT(3) = 2 for higher accuracy of membrane stress, otherwise, KEYOPT(3) = 0; KEYOPT(8) = 2 for all cases	KEYOPT(3) = 0, KEYOPT(8) = 0
	Membrane-only option (KEYOPT(1) = 1)	KEYOPT(3) = 2 for higher accuracy of stress, otherwise, KEYOPT(3) = 0; KEYOPT(8) = 0 for all cases	KEYOPT(3) = 0, KEYOPT(8) = 0
PLANE182	Plane stress	KEYOPT(1) = 2	KEYOPT(1) = 0
	Not plane stress	KEYOPT(1) = 2[1]	KEYOPT(1) = 0[1]
PLANE183	Plane stress	No change. (Default element settings are best.)	
	Not plane stress	no change (default element settings are best)[1]	
SOLID185		KEYOPT(2) = 2[1]	KEYOPT(2) = 0[1]
SOLID186		KEYOPT(2) = 0[1]	
SOLID187		No change. (Default element settings are best.)[1]	
SOLID272		KEYOPT(6) = 1	
SOLID273		No change. (Default element settings are best.)[1]	
SOLID285		No change. (Default element settings are best.)	
BEAM188		KEYOPT(3) = 2, KEYOPT(1) = 1[2]	
BEAM189		KEYOPT(1) = 1[2]	
SHELL208		KEYOPT(8) = 2	KEYOPT(8) = 0
SHELL209		KEYOPT(8) = 2	KEYOPT(8) = 0
SHELL281		No change. (Default element settings are best.)	
PIPE288		KEYOPT(3) = 2 KEYOPT(4) = 2 KEYOPT(15) = 1[3]	
PIPE289		KEYOPT(4) = 2 KEYOPT(15) = 1[3]	

1. If even one of your hyperelastic materials is fully incompressible, then KEYOPT(6) = 1 is required. Therefore, ANSYS, Inc. recommends using a different element type for such materials to avoid using the more costly mixed u-P formulation where it is not needed.

2. Only when the element section is not circular, otherwise KEYOPT(1) = 0.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Only when the element cross section is composite; otherwise, KEYOPT(15) = 0

For the solid elements listed above, **ETCONTROL** provides suggestions or resets the KEYOPTs for the element technology settings (i.e., KEYOPT(1) for PLANE182, KEYOPT(2) for SOLID185 and SOLID186). The Mixed u-P formulation KEYOPT(6) is reset when necessary; that is, when fully incompressible hyper-elastic materials are associated with non-plane stress states.

For BEAM188 and BEAM189, KEYOPT(1) is always suggested for non-circular cross section beams and should be reset when **ETCONTROL**,SET,.. is issued. However, the KEYOPT setting changes the number of degrees of freedom at each node, so it must be set before you issue any **D**, **DK**, **DA**, and similar commands. Because the **ETCONTROL** reset is done at the beginning of the solution stage, a message appears indicating stating that you should change KEYOPT(1) to 1, but the change does not occur automatically.

For SHELL181, the setting for KEYOPT(3) depends on your problem and your purpose if the element type is associated with any non-hyperelastic materials. In this case, the program cannot reset it automatically, but it displays a message with the recommended setting (even if you used **ETCONTROL**,SET,..). You should reset this manually if the defined criteria matches your problem.

For the coupled-field elements PLANE223, SOLID226, and SOLID227, see the individual element descriptions in the *Element Reference* for information on **ETCONTROL** suggestions/resettings.

If an element type is defined but not associated with any material, no suggestions or resets are done for that element type. The stress states and options are mentioned only when they affect the suggestions or settings. The suggestions are available in output.

All nonlinear stress/strain data should be input as true-stress/true-(logarithmic-) strain. Accordingly, all output data is also indicated as true-stress/true-strain. For small strains, the true-stress/true-strain data and engineering-stress/engineering-strain data are essentially identical.

5.8. Elements Under Linear Perturbation

Most current-technology elements support linear perturbation analysis.

Because linear perturbation is based on a solution at a particular time from a linear or nonlinear analysis, known as the base (or prior) analysis, the element behaviors are based on the properties and behavior of the base analysis, but are different from the base analysis.

The following elements can be used in a linear perturbation analysis as well as any downstream analyses:

Table 5.5: Elements Supporting Linear Perturbation Analysis

Category	Element Name(s)
Spars	LINK180
Beams	BEAM188, BEAM189 [1]
Pipes	PIPE288, PIPE289
2-D Solids	FLUID29, PLANE121, PLANE182, PLANE183, PLANE223, PLANE233
3-D Solids	SOLID185, SOLID186, SOLID187, SOLID285, SOLID226, SOLID227, SOLID237, FLUID30, FLUID220, FLUID221
Shells	SHELL181, SHELL208, SHELL209, SHELL281
Solid-Shell	SOLSH190

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Element Name(s)
Interface	INTER192, INTER193, INTER194, INTER195
Contact	TARGE169, TARGE170, CONTA171, CONTA172, CONTA173, CONTA174, CONTA175, CONTA176, CONTA177, CONTA178
Specialty	COMBIN14, MASS21, MATRIX27, COMBIN39, SURF153, SURF154, SURF156, PRETS179, MPC184, MESH200, FOLLW201, COMBI214, HSFLD241, HSFLD242
CMS superelement (or substructure)	MATRIX50 [2]
Circuit	CIRCU94, CIRCU124

1. Not supported for nonlinear general beam sections (**SECTYPE**,GENB).

2. MATRIX50 can be used in a linear perturbation analysis the same way it is used in a typical substructure use pass. However, MATRIX50 is treated as a linear element in the linear perturbation analysis and is not prestressed.

5.8.1. Material Properties of Structural Elements in Linear Perturbation

The base analysis of the linear perturbation can be a nonlinear analysis with nonlinear materials, geometric nonlinearity, or contact elements included in the model. In the linear perturbation analysis, the geometric nonlinearity at the substep where linear perturbation is initiated is completely inherited in $\left[K_i^T\right]$ of Equation 15.225 in the *Mechanical APDL Theory Reference*. This includes large deformation, large rotation, and contact effects. The linear perturbation analysis can be understood as an iteration in the nonlinear base analysis. In the linear perturbation analysis, all of the nonlinear effects are taken into consideration and "frozen" so that the loading and deformation is linear. The nonlinear effects are also carried to the stress expansion pass and the following downstream analysis, if any.

5.8.1.1. Specifying Material Behavior in Linear Perturbation

Any nonlinear material must behave linearly in the linear perturbation analysis. For all materials, the same material properties used in the linear perturbation are also used in the stress-expansion pass and in all subsequent downstream analyses, if any.

Linear Materials

For pure linear elastic materials used in the base analysis, the same properties are used in the linear perturbation. Because the material constitutive curve is a line, the tangent is the same as the line. The material-handling key (`MatKey`) specified via the **PERTURB** command is ignored.

Hyperelastic Materials

For hyperelastic materials used in the base analysis, the material properties are assumed to be linear elastic. The program obtains the material property data (material Jacobian) based on the tangent of the hyperelastic material's constitutive law at the point where restart or linear perturbation occurs. The material-handling key (`MatKey`) specified via the **PERTURB** command is ignored.

Nonlinear Materials Other Than Hyperelastic Materials

For any nonlinear materials other than hyperelastic materials used in the base analysis, the material properties are assumed to be linear elastic. The program determines the material properties based on the material-handling key (`MatKey`) specified via the **PERTURB** command, as follows:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If the default automatic material-handling behavior (`MatKey` = AUTO) is in effect, the program uses the linear portion of the nonlinear materials (that is, the parts defined via **MP** commands) for the linear perturbation.
- If the tangent option (`MatKey` = TANGENT) is in effect, the program uses the consistent tangent on the material constitutive curve for the linear perturbation. The program obtains the consistent tangent at the point where restart or linear perturbation occurs.

For most materials, the tangent option gives different material properties than those given by the automatic option. It is possible, however, that they could be identical or very similar if a.) the material is elasto-plastic rate-independent and is unloading (or has neutral loading) at the restart point, or b.) the material is rate-dependent, depending on the material properties and loading conditions.

5.8.2. Interpretation of Structural Element Results After a Linear Perturbation Analysis

After a linear perturbation analysis, the reported stress, elastic strain, and thermal strains are the values due to the linear perturbation analysis; for example, for linear perturbation modal analysis, the stress and elastic strain are the values due to the mode shapes. The effects of any geometric nonlinearity are taken into consideration in the stress expansion.

Because a linear perturbation analysis can be understood as an extra iteration of a base analysis, all history-dependent results of the base analysis are inherited in the results of the linear perturbation analysis. Therefore, any plastic strains, creep strains, swelling strains, and contact results from the base analysis are available in the results data of the linear perturbation analysis.

The total strain is the sum of *all* strains (for example, **PLNSOL**,EPTO). The nonlinear solution quantities such as equivalent stress, stress state ratio, and plastic state variable (for example, **PLESOL**,NL,...) are also available. The one exception is that hydrostatic pressure is the value from the linear perturbation analysis instead of from the base analysis.

For hyperelastic materials, the strain after the linear perturbation analysis is the elastic strain due to the linear perturbation analysis, and the total strain is the same as the elastic strain. For plane stress cases, the direct strain in the Z direction is calculated with the incompressible condition for hyperelastic materials, and with linear elastic material properties for any material other than hyperelastic materials.

The energy densities (for example, **PLESOL**,SEND,...) are inherited from the base analysis, except for the elastic energy density which is energy density calculated in the linear perturbation analysis. However, the reported strain energy (for example, **PRESOL**,SENE) includes the elastic part due to linear perturbation and others inherited from the base analysis (if any).

If the base analysis includes geometric nonlinearity, the Euler angles or the geometric nonlinear effects of the base analysis are available in the results file. Thus, all the output quantities of the linear perturbation analysis are reported consistently in global, local, and rotated local coordinate systems, as in the base analysis.

Temperatures saved in the linear perturbation results (`Jobname.rstp`) are the temperatures which have been modified during the second phase of the linear perturbation procedure (that is, as the perturbation load). This is only applicable to linear perturbation eigenvalue buckling analysis.

5.8.3. Loads, Initial Conditions, and Other Limitations in Linear Perturbation

The following restrictions apply to linear perturbation analysis:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Perturbation loads cannot be non-mechanical loading such as initial conditions (**INISTATE** command) or swelling effects. These loads cannot be modified or deleted during the linear perturbation analysis.
- For a linear perturbation static or eigenvalue buckling analysis, a thermal load can be modified by specifying a new temperature in the second phase of the linear perturbation procedure. In a linear perturbation static or buckling analysis, assuming that the base analysis is nonlinear, the reference temperature is the temperature from which the linear perturbation analysis is restarted (and not the reference temperature [**TREF**] from the base analysis).
- If the base analysis is linear, then the reference temperature used in the linear perturbation is the **TREF** temperature from the base analysis.
- Thermal loads from the base analysis are ignored in a linear perturbation modal or full harmonic analysis. In addition, you cannot apply thermal loads in the second phase of a linear perturbation modal or full harmonic analysis.
- No element can be activated or deactivated using the birth and death feature during the second phase of a linear perturbation analysis; however, this is allowed in the first phase of linear perturbation.
- Linear perturbation analyses do not support user-defined materials (user subroutine `UserMat`).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 6: Elements for Multiphysics Analysis

Many elements are available for creating models for analyses that involve various physics and that are not strictly structural in nature.

The following related topics are available:

6.1. Elements for Coupled-Physics Analysis
6.2. Elements for Thermal Analysis
6.3. Elements for Acoustic Analysis
6.4. Elements for Electric and Magnetic Analysis
6.5. Elements for Diffusion Analysis

6.1. Elements for Coupled-Physics Analysis

Coupled-field elements are available for combining various types of physics. A coupled-field element has all necessary degrees of freedom necessary for solving a coupled-field problem. When a coupled-field element is used, the multiple physics are solved simultaneously (in a single analysis with a single mesh and model).

The program handles coupling between the physics by calculating the necessary element matrices or element load vectors.

Coupled-field elements are recommended for strongly coupled physics such as piezoelectric, thermo-electric, electromagnetic, and highly nonlinear coupled-field analyses that include material and geometric nonlinearities, temperature-dependent material properties, and multiphysics contact.

The following additional topics are available:

6.1.1. Coupled-Field Element Classifications
6.1.2. Selecting Elements for Coupled-Physics Analysis

For more information, see the *Coupled-Field Analysis Guide*.

6.1.1. Coupled-Field Element Classifications

The following types of coupled-field elements are available:

- **Continuum Elements (p. 102)**
  - General purpose coupled-field elements
  - Coupled pore-pressure mechanical elements
  - Thermal-electric link and shell elements
  - Magneto-structural element
  - Electromagnetic elements
  - Fluid elements

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Reduced-Order Elements (p. 101)**
- **Contact Elements (p. 101)**

6.1.2. Selecting Elements for Coupled-Physics Analysis

Continuum Elements		
General Purpose Coupled-Field	**Degrees of Freedom**	**Coupled-Physics Capabilities**
PLANE223 -- Higher-order quadrilateral	UX, UY, TEMP, VOLT, CONC	Structural-thermal Piezoresistive Electroelastic Piezoelectric Thermal-electric Structural-thermo-electric Thermo-piezoelectric Thermo-diffusion Structural-diffusion Structural-thermo-diffusion
SOLID226 -- Higher-order hexahedral	UX, UY, UZ, TEMP, VOLT, CONC	
SOLID227 -- Higher-order tetrahedral		
SOLID5 -- Lower-order hexahedral	UX, UY, UZ, TEMP, VOLT, MAG	Structural-thermo-electro-magnetic Thermo-electro-magnetic Piezoelectric
PLANE13 -- Lower-order quadrilateral	UX, UY, TEMP, VOLT, AZ	Structural-thermo-electro-magnetic Piezoelectric Electromagnetic
SOLID98 -- Higher-order tetrahedral	UX, UY, UZ, TEMP, VOLT, MAG	Structural-thermo-electro-magnetic Thermo-electro-magnetic Piezoelectric
Coupled Pore-Pressure Mechanical	**Degrees of Freedom**	**Coupled-Physics Capabilities**
CPT212 --Lower-order quadrilateral	UX, UY, PRES	Pore-pressure
CPT213 -- Higher-order quadrilateral		
CPT215 -- Lower-order hexahedral	UX, UY, UZ, PRES	
CPT216 -- Higher-order hexahedral		
CPT217 -- Higher-order tetrahedral		
Thermal-Electric	**Degrees of Freedom**	**Coupled-Physics Capabilities**
LINK68 -- Lower-order line	TEMP, VOLT	Thermal-electric
SHELL157 -- Lower-order shell		

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Continuum Elements		
Electromagnetic	**Degrees of Freedom**	**Coupled-Physics Capabilities**
PLANE233 -- Higher-order quadrilateral	AZ, VOLT, EMF	Electromagnetic Stranded coil
SOLID236 -- Edge-based mixed-order hexahedral		
SOLID237 -- Edge-based mixed-order tetrahedral		
PLANE53 -- Higher-order quadrilateral	AZ, VOLT, CURR, EMF	Electromagnetic Stranded coil Circuit-coupled stranded coil Circuit-coupled electromagnetic
SOLID97 -- Lower-order hexahedral	AX, AY, AZ, VOLT, CURR, EMF	
Fluid	**Degrees of Freedom**	**Coupled-Physics Capabilities**
FLUID29 -- Lower-order quadrilateral [1]	UX, UY, PRES	Acoustic fluid-structure interaction
FLUID30 -- Lower-order hexahedral	UX, UY, UZ, PRES	
FLUID116 -- Lower-order pipe [2]	PRES, TEMP	Thermal-fluid
FLUID136 -- Low-order and high-order quadrilateral	UX, UY, UZ, PRES	Squeeze film fluid-structure interaction
FLUID220 -- Higher-order hexahedral [1]	UX, UY, UZ, PRES	Acoustic fluid-structure interaction
FLUID221 -- Higher-order tetrahedral [1]		
Reduced-Order Elements		
TRANS126 -- Electromechanical transducer	UX (UY, UZ), VOLT	Reduced-order electrostatic-structural
ROM144 -- Reduced-order electrostatic-structural	EMF, VOLT, UX	
Contact Elements		
CONTA171 -- 2-D lower-order surface-to-surface contact	UX, UY, TEMP, VOLT	Structural-thermal Structural-thermo-electric Thermal-electric Structural-electric
CONTA172 -- 2-D higher-order surface-to-surface contact		
CONTA173 -- 3-D lower-order surface-to-surface contact	UX, UY, UZ, TEMP, VOLT	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Continuum Elements		
CONTA174 -- 3-D higher-order surface-to-surface contact		
CONTA175 -- 2-D / 3-D node-to-surface contact	UX, UY, (UZ), TEMP, VOLT	

Use the following table to select coupled-field elements:

1. See Elements for Acoustic Analysis (p. 104) for more information
2. See Elements for Thermal Analysis (p. 102) for more information

For related information, see Selecting Elements for Your Analysis.

6.2. Elements for Thermal Analysis

When performing a thermal analysis, consider the dimensionality of the problem as well as the loads required. Use the following information to select elements for modeling various regions as well as elements to apply loads:

6.2.1. Thermal Elements for the Structural Region
6.2.2. Thermal Elements for the Fluid Region
6.2.3. Thermal Loading Elements
6.2.4. Thermal Control Element
6.2.5. Other Elements Useful in a Thermal Analysis

A complex thermal model often has a combination of all or most of the element types described.

For more information, see the *Thermal Analysis Guide*.

6.2.1. Thermal Elements for the Structural Region

Typically, the solid region should be meshed with 2-D or 3-D continuum elements. 1-D elements serve to couple these regions. The 0-D element can be used to represent a lumped mass (that is, a reduced-order model of a continuum region).

Use the following table to select thermal elements for the structural region:

Continuum	Application	Elements
3-D	Modeling homogeneous materials and layered solids / composites	SOLID278 -- Lower-order element that can degenerate
		SOLID279 -- Higher-order element that can degenerate
	Modeling homogeneous materials	SOLID70 -- Lower-order thermal solid that can degenerate
		SOLID87 -- Higher-order tetrahedral-only thermal solid
		SOLID90 -- Higher-order, accurate thermal solid that can degenerate

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Continuum	Application	Elements
2.5-D	Analyzing the in-plane temperature distribution of thin structures such as plates or shells. [1]	SHELL131 -- 3-D lower-order shell with in-plane and through-thickness thermal conduction capability
		SHELL132 -- 3-D higher-order shell with in-plane and through-thickness thermal conduction
2-D	Modeling irregular meshes	PLANE35 -- Triangular thermal solid (compatible with PLANE77)
	Modeling a plane or an axisymmetric ring	PLANE55 -- Thermal solid with 2-D thermal conduction
	Modeling curved boundaries	PLANE77 -- Higher-order thermal solid
1-D	Conduction link between regions	LINK33 -- Uniaxial 3-D conduction bar [2]
	Convection link between regions	LINK34 -- Uniaxial convection link [2]
	Radiation link between regions	LINK31 -- Uniaxial radiation link [2]
0-D	Representing a body having thermal capacitance but with no significant temperature gradients	MASS71 -- Thermal mass point element

1. If the model containing the conducting shell element is to be analyzed *structurally*, use an equivalent structural shell element (such as SHELL281) instead.

2. In most cases, contact elements can be used in place of this element.

For related information, see Selecting Elements for Your Analysis.

6.2.2. Thermal Elements for the Fluid Region

For the 1-D continuum, use FLUID116, a reduced-order coupled thermal-fluid pipe, to represent fluid temperature for flow-through pipes/internal channels. The element can be connected to nodes in the structural region.

6.2.3. Thermal Loading Elements

Two methods are available for applying loads to thermal elements:

6.2.3.1. Using Thermal Surface-Loading Elements
6.2.3.2. Using Contact Elements to Load Thermal Models

6.2.3.1. Using Thermal Surface-Loading Elements

In a thermal model, the surface loads can consist of convection, heat flux, or radiation. Although you can apply the loads directly on the continuum elements, it is more convenient to apply them via surface-effect elements, which are overlaid onto other thermal elements.

Use these surface-effect elements to apply convection, heat flux, and radiation (to a point) on solid regions:

- SURF151 -- 2-D thermal surface effect

- SURF152 -- 3-D thermal surface effect

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use these surface-radiosity elements to apply grey body radiation (where view factors are unknown):

- SURF251 -- 2-D radiosity surface
- SURF252 -- 3-D radiosity surface

6.2.3.2. Using Contact Elements to Load Thermal Models

Besides surface loads, it is also possible to load thermal models via contact elements.

When two or more structural regions are in close vicinity or contact, it is sometimes necessary to account for convection, conduction, or radiation between the regions. Frictional forces between the contacting surfaces can also be a source of heat generation.

Use these contact elements between solid regions:

2-D Contact Elements

- CONTA171 -- Lower-order surface-to-surface contact element pasted on the surface of a region
- CONTA172 -- Higher-order surface-to-surface contact element pasted on the surface of a region
- TARGE169 -- Target segment, used with CONTA171 and CONTA172, which must be pasted on the target region

3-D Contact Elements

- CONTA173 -- Lower-order surface-to-surface contact element pasted on the surface of a region
- CONTA174 -- Higher-order surface-to-surface contact element pasted on the surface of a region
- TARGE170 -- Target segment, used with CONTA173 and CONTA174, which must be pasted on the target region

A special 2-D/3-D node-to-surface contact element, CONTA175, is also available. You can use it with either TARGE169 or TARGE170.

6.2.4. Thermal Control Element

Use the COMBIN37 control element to enable or disable heat flow between two points when the temperature (or other control parameters) reach a specified threshold value.

6.2.5. Other Elements Useful in a Thermal Analysis

In addition to the thermal and related loading elements described here, also consider coupled-physics elements, which can couple thermal and other physics.

You can also create a custom element via USER300 to model special physics, or combine a group of linear thermal elements into a MATRIX50 superelement for greater solution efficiency.

6.3. Elements for Acoustic Analysis

The following topics related to the elements used for acoustic analyses are available:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.1. Acoustic Element Types

An acoustic analysis calculates either the propagation properties of pure acoustic waves in the given environment or the coupled acoustic structural interaction (FSI).

Support is available for time-harmonic, modal and transient acoustic analysis.

Use the following fluid elements to simulate 2-D acoustic or coupled acoustic problems:

- FLUID29 -- Axisymmetric harmonic acoustic fluid.
- FLUID129 -- Infinite acoustic.

  This element is a companion to FLUID29. It is used to envelop a model made of FLUID29 finite elements for the truncation of infinite propagating domain.

Use these fluid elements for 3-D simulation:

- FLUID30 -- Acoustic fluid
- FLUID130 -- Infinite acoustic

  This element is a companion to FLUID30, FLUID220, and FLUID221. It is used to envelop a model made of FLUID30, FLUID220, and FLUID221 finite elements for the truncation of infinite propagating domain.
- FLUID220 -- 20-node acoustic solid
- FLUID221 -- 10-node acoustic tetrahedral solid

6.3.2. 3-D Acoustic Element Usage

Use FLUID30, FLUID220 and FLUID221 to model the fluid medium and the interface in fluid/structure interaction problems. Typical applications include sound wave propagation and submerged structure dynamics. The governing equation for acoustics, namely the 3-D wave equation, has been discretized taking into account the coupling of acoustic pressure and structural motion at the interface. The element node has four degrees of freedom per node: translations in the nodal x, y and z directions, and pressure. The translations are applicable only at nodes on the interface. Acceleration effects like those in sloshing problems may be included. For more information, see Acoustics in the *Mechanical APDL Theory Reference*.

The elements have the capability to include damping of sound-absorbing material at the interface as well as damping within the fluid. The elements can be used with or without other 3-D structural elements to perform symmetric, unsymmetric or damped modal (**MODOPT**), full harmonic (**HROPT**), and full transient method analyses (**TRNOPT**).

See FLUID30 in the *Mechanical APDL Theory Reference* for more information about the low-order element. See FLUID220 for more information about the high-order hexahedral element and FLUID221 for more information about the high-order tetrahedral element.

6.3.3. 3-D Acoustic Element Input

The geometry, node locations, and coordinate system for each 3-D acoustic element is shown in Figure 30.1: FLUID30 Geometry (p. 185), Figure 220.1: FLUID220 Geometry (p. 1149), and Figure 221.1: FLUID221 Geometry (p. 1151). The elements are defined by eight nodes (FLUID30), 20 nodes (FLUID220), or 10 nodes

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(FLUID221), a reference pressure, and the isotropic material properties. The reference pressure (PREF) is used to calculate the element sound pressure level (defaults to 20×10^{-6} N/m^2). The speed of sound ($\sqrt{k/\rho_o}$) in the fluid is input by SONC where k is the bulk modulus of the fluid (Force/Area) and ρ_o is the mean fluid density (Mass/Volume) (input as DENS). The dissipative effect due to fluid viscosity can be included (input as **MP**,VISC). DENS, SONC, and VISC are evaluated at the average of the nodal temperatures.

The **TB**,PERF command defines the equivalent fluid of the perforated material using the Johnson-Champoux-Allard, Delany-Bazley, Miki, impedance-propagating constant, or complex density-velocity model. The material properties, thermal conductivity (**MP**,KXX), heat coefficient at constant volume per unit of mass (**MP**,CVH), the dynamic viscosity (**MP**,VISC) (default 1.84×10^{-5} N•s/m^2), and specific heat (**MP**,C) may be defined for the Prandtl number calculation (default 0.713) and the specific heat ratio (default 1.4) if necessary. Viscosity defaults to 1.84×10^{-5}s for elements where the **TB**,PERF command has been issued. For all other elements, the viscosity is assumed to be zero when undefined.

The **TB**,PERF command is also used to define the transfer admittance matrix for the equivalence of complex perforated structures, including plates with hole arrays. The **TB**,AFDM command defines the frequency-dependent acoustic materials combined with the **TB**,FIELD command. The parameters set via the **TB**,PERF command can be frequency-dependent along with the **TB**,FIELD command. Both the boundary layer (**SF**,BLI) and the low-frequency reduced model (**TB**,AFDM) are available to simulate the interaction between acoustic fluid and rigid wall in visco-thermal medium

Element loads are described in Element Loading (p. 46). Fluid-structure interfaces (FSIs) can be flagged by surface loads at the element faces as shown by the circled numbers in Figure 30.1: FLUID30 Geometry (p. 185), Figure 220.1: FLUID220 Geometry (p. 1149), and Figure 221.1: FLUID221 Geometry (p. 1151). Specifying the FSI label (without a value) (**SF**, **SFA**, **SFE**) couples the structural motion and fluid pressure at the interface. Deleting the FSI specification (**SFDELE**, **SFADELE**, **SFEDELE**) removes the flag. The flag specification should be on the fluid elements at the interface.

The fluid-structure interface (FSI) may be automatically flagged if acoustic elements are adjacent to solid structural elements (except for shell elements) and FSIs have not been flagged manually. The surface load label IMPD with a given complex impedance value can be used to include damping that may be present at a structural boundary with a sound absorption lining. These impedance boundary conditions may also be applied to a flagged FSI interface. A zero value of IMPD removes the damping calculation. The displacement degrees of freedom (UX, UY and UZ) at the element nodes not on the interface should be set to zero to avoid zero-pivot warning messages if the coupled acoustic elements are used (KEYOPT(2) = 1). The surface load label SHLD with a given amplitude and initial phase angle is used to define a normal sound speed in a harmonic analysis or a normal sound acceleration in a transient analysis on the exterior surface. The sloshing surface that must be parallel to the coordinate plane of the global Cartesian system can be flagged by the surface load label FREE. When near or far field parameters are required, the surface load label MXWF should be applied to the equivalent source surface. The label MXWF may be automatically applied to a PML-acoustic medium interface or exterior surface with the label INF, if MXWF surfaces have not been flagged manually. The surface load label ATTN with the attenuation coefficient is used to define an absorbing surface in modal and harmonic response analyses. The surface load label BLI is applied on the rigid wall to introduce the boundary layer model in viscous-thermal fluid. The surface load label PORT is used to define the network ports. When the transfer admittance matrix is used, a pair of ports must be defined on the opposite faces in the same element. The surface loads with load label IMPD, ATTN and SHLD can be frequency- or time-dependent using tabular inputs

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

Mass source (in units of mass/length^3/time) can be defined in a harmonic analysis and mass source rate (partial time derivative of mass source in units of mass/length^3/time^2) can be defined in a transient analysis using the **BF**, **BFL**, **BFA** or **BFV** command with the label JS. For harmonic response analyses, both the amplitude and initial phase may be applied so that the inhomogeneous Helmholtz equation is solved. The impedance sheet inside a fluid may be defined using the **BF** or **BFA** command with the label IMPD in a harmonic analysis. For a non-uniform acoustic medium, the reference temperature T_0 (**TREF** command) and the reference static pressure (real constant PSREF) should be defined. PSREF defaults to 101325, which is standard atmospheric pressure in units of N/m^2. Nodal temperatures may be input by body load commands. The nodal static pressure can also be input by the **BF** command with Lab = CHRGD. The **BF** command can be used with Lab = EF to define non-uniform velocity in a harmonic analysis or non-uniform acceleration in a transient analysis.

One-way coupling from structure to acoustics is more computationally efficient, while the acoustic impact on the structure can be neglected. The structural solution is performed first. If the conforming mesh is used on the FSI interface, you can flag the FSI in the structural model by the **SF**,FSIN command and write the structural results on the FSI to a file with extension `.asi` using the **ASIFILE** command. The velocities or accelerations on the FSI are loaded into the sequential acoustic solution using the **ASIFILE** command with multiple frequencies or time steps. If the non-conforming mesh is used on the FSI, you can map the structural results on the FSI of the structural model via the Workbench External Data to the acoustic model, then perform an acoustic solution.

The nonlinear static analysis for the structural deformation can be performed, before performing a coupled or pure acoustic solution. Because of the structural deformation, the mesh in acoustic domain is morphed during the static structural solution (**MORPH**). The linear perturbation scheme (**ANTYPE** and **PERTURB**) is used to perform a sequential acoustic solution with updated mesh.

KEYOPT(2) = 1 specifies the absence of a structure at the interface and the absence of coupling between the fluid and structure. Because the absence of coupling produces symmetric element matrices, a symmetric eigensolver (**MODOPT**) may be used within the modal analysis. KEYOPT(2) = 0 (the default) specifies a coupled (unsymmetric) problem, requiring a corresponding unsymmetric eigensolver (**MODOPT**). If KEYOPT(1) = 1 or 2, specifying symmetric algorithms in the presence of FSI coupling, a symmetric eigensolver may also be used for modal analysis (**MODOPT**) and a symmetric linear equation solver may be used for full harmonic analysis.

KEYOPT(4) specifies the existence of perfectly matched layers (PML) to absorb the outgoing sound waves. The pressure on the exterior enclosure of PML must be constrained to zero, unless the pressure is on the symmetric planes. For more information about using PML, see Acoustics and Perfectly Matched Layers (PML) in the *Mechanical APDL High-Frequency Electromagnetic Analysis Guide*.

If free surface effects are present (**SF**,,FREE), vertical acceleration (*ACELZ* on the **ACEL** command) is necessary to specify gravity, even for a modal analysis.

A harmonic analysis can be performed over the octave band, 1/3 octave band, 1/6 octave band, or general frequency band with logarithm frequency span (**HARFRQ**,,,,*LogOpt*).

For acoustic scattering analysis, acoustic incident waves may be specified outside of the model by using the **AWAVE** command. These incident waves can be combined with a PML or Robin boundary surface (**SF** command with the INF flag). Either the total field or pure scattered field formulation may be used. The acoustic near- and far-field parameters may be calculated by **PRNEAR**, **PLNEAR**, **PRFAR**, or **PLFAR**.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the pure scattered formulation is used, acoustic incident waves may also be specified inside of the model by using the **AWAVE** command.

When excitation sources are defined inside of the model (**AWAVE**), the scattered formulation must be used (**ASOL**).

The acoustic near- and far-field parameters can be calculated via the **PRNEAR**, **PLNEAR**, **PRFAR**, and **PLFAR** commands. The sound power data, including the transmission loss, can be calculated on the defined ports via the **SPOWER** command.

Element input is given in 3-D Acoustic Element Input Summary (p. 108). A general description of element input is given in Element Input (p. 43).

6.3.4. 3-D Acoustic Element Input Summary

Nodes

FLUID30: I, J, K, L, M, N, O, P

FLUID220: I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B

FLUID221: I, J, K, L, M, N, O, P, Q, R

Degrees of Freedom

UX, UY, UZ, PRES if KEYOPT (2) = 0
PRES if KEYOPT (2) = 1

Real Constants

PREF -- Reference pressure
PSREF -- Reference static pressure

Material Properties

DENS, SONC, VISC, KXX, C, CVH, BVIS (**MP** command)
PERF, AFDM (**TB** command)

Surface Loads

Fluid-structure interface (FSI) flag, impedance (IMPD), normal speed or normal acceleration (SHLD), sloshing surface (FREE), equivalent source surface (MXWF), Robin boundary surface (INF), attenuation coefficient (ATTN), viscous-thermal boundary layer (BLI) and port (PORT):

FLUID30 and FLUID220--
face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)
FLUID221 --
face 1 (J-I-K), face 2 (I-J-L), face 3 (J-K-L), face 4 (K-I-L)

Body Loads

Mass source or mass rate (JS), static pressure (CHRGD), impedance (IMPD), temperature (TEMP), velocity or acceleration (VELO)

Special Features

None

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(1)
Specific algorithm options:

1 --
FSIs present in the model for modal analysis (symmetric element matrix)

2 --
FSIs present in the model for full harmonic analysis (symmetric element matrix)

KEYOPT(2)
Acoustic element types:

0 --
Coupled acoustic element with FSIs

1 --
Uncoupled acoustic element without FSIs

KEYOPT(4)
PML absorbing condition:

0 --
Do not include PML absorbing condition

1 --
Include PML absorbing condition

6.3.5. 3-D Acoustic Element Item Output Data

The solution output associated with the elements consists of the following:

- Nodal displacements and pressures included in the overall nodal solution.
- Nodal velocity for modal and full harmonic or nodal pressure gradient for transient analysis is included in the element nodal solution via standard results output commands with Item = PG (for example, **PRNSOL**, **PLVECT**, **PRESOL**, and **PLESOL**).
- Nodal sound pressure level (SPL) for modal and harmonic analyses.
- Additional element output as shown in Table 6.1: 3-D Acoustic Element Output Definitions (p. 110).

A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 6.1: 3-D Acoustic Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	FLUID30: I, J, K, L, M, N, O, P FLUID220: I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B FLUID221: I, J, K, L, M, N, O, P, Q, R	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	1
TEMP	FLUID30 : T(I), T(J), …, T(P) FLUID220: T(I), T(J), …, T(B) FLUID221: T(I), T(J), …, T(R)	Y	Y
PRESSURE	Average pressure	Y	Y
PG(X,Y,Z)	Velocity components and vector sum for modal and harmonic analysis or pressure gradient components and vector sum for transient analysis	Y	Y
PL2	Square of the L2 norm of pressure over element volume	2	2
DENSRE	Real part of complex effective density	2	2
DENSIM	Imaginary part of complex effective density	3	3
SONCRE	Real part of complex effective sound velocity	3	3
SONCIM	Imaginary part of complex effective sound velocity	3	3
POUT	Output sound power	2	2
PINC	Input sound power	2	2

1. Available only at centroid as a ***GET** item.

2. Output only if **ANTYPE**,HARMIC or MODAL

3. Available only for equivalent fluid model.

Table 6.2: 3-D Acoustic Element Item and Sequence Numbers (p. 111) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) of this reference for more information. The following notation is used in Table 6.2: 3-D Acoustic Element Item and Sequence Numbers (p. 111):

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name
output quantity as defined in the Table 6.1: 3-D Acoustic Element Output Definitions (p. 110)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 6.2: 3-D Acoustic Element Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
PGX	SMISC	1
PGY	SMISC	2
PGZ	SMISC	3
PL2	SMICS	4
DENSRE	SMISC	5
DENSIM	SMISC	6
SONCRE	SMISC	7
SONCIM	SMICS	8
POUT	SMISC	9
PINC	SMICS	10
PRESSURE	NMISC	1
PGSUM	NMISC	2

6.3.6. 3-D Acoustic Element Assumptions and Restrictions

- The element must not have a zero volume.
- For FLUID30, element nodes may be numbered either as shown in Figure 30.1: FLUID30 Geometry (p. 185) or may have planes IJKL and MNOP interchanged. All elements must have 8 nodes. A prism-shaped element may be formed by defining duplicate K and L and duplicate O and P nodes. (See Degenerated Shape Elements (p. 39).) A tetrahedron or pyramid shape is also available.
- For FLUID220, the element nodes may be numbered either as shown in Figure 220.1: FLUID220 Geometry (p. 1149) or may have planes IJKL and MNOP interchanged. All elements must have 20 nodes. A prism-shaped element may be formed by defining duplicate L and S and duplicate P and W nodes. (See Degenerated Shape Elements (p. 39).) A pyramid shape is also available.
- For FLUID221, the element nodes may be numbered as shown in Figure 221.1: FLUID221 Geometry (p. 1151). All elements must have 10 nodes
- The element may not be twisted such that it has two separate volumes. Such a case typically occurs when the element nodes are not in the correct sequence.
- The acoustic pressure in the fluid medium is determined by the wave equation with the following assumptions:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The fluid is compressible (density changes due to pressure variations).
- There is no mean flow of the fluid.
- The acoustic pressure is considered to be the excess pressure from the mean pressure.
- Analyses are limited to relatively small acoustic pressures so that the changes in density are small compared with the mean density.

- The lumped mass matrix formulation (**LUMPM**,ON) is not valid for this element.
- The symmetric matrix formulation (KEYOPT(1) = 1) is not valid for coupled modal analysis with viscous material.

6.3.7. 3-D Acoustic Element Product Restrictions

No product-specific restrictions exist for 3-D acoustic elements.

6.4. Elements for Electric and Magnetic Analysis

Elements available for modeling electromagnetic phenomena are classified as follows:

- Continuum:

  Electrostatic
  Electric
  Magnetic
  Electromagnetic
  Infinite

- Source
- Circuit
- Contact

Use the following table to select an appropriate element for your application. For more details about the elements used for electromagnetic analyses, see the *Low-Frequency Electromagnetic Analysis Guide*.

Electric and Magnetic Analyses		
Electrostatic	**Degrees of Freedom**	**Application**
PLANE121 -- Higher-order quadrilateral	VOLT	Charge-based static and time-harmonic quasistatic electric field analyses
SOLID122 -- High-order hexahedral		
SOLID123 -- High-order tetrahedral		
Electric	**Degrees of Freedom**	**Application**
PLANE230 -- Higher-order quadrilateral	VOLT	Current-based static, time-transient and time-harmonic quasi-static electric field analyses

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Electric and Magnetic Analyses		
SOLID231 -- High-order hexahedral		
SOLID232 -- High-order tetrahedral		
Magnetic	**Degrees of Freedom**	**Application**
SOLID96 -- Low-order hexahedral	MAG	Magnetostatic analysis
Electromagnetic	**Degrees of Freedom**	**Application**
PLANE53 -- Higher-order quadrilateral	AZ, VOLT, CURR, EMF	Static, transient or harmonic electromagnetic, stranded coil, circuit-coupled stranded coil, circuit-coupled electromagnetic analyses
SOLID97 -- Lower-order hexahedral	AX, AY, AZ, VOLT, CURR, EMF	
PLANE233 -- Higher-order quadrilateral	AZ, VOLT, EMF	Static, transient or harmonic electromagnetic and stranded coil analyses
SOLID236 -- Edge-based mixed-order hexahedral		
SOLID237 -- Edge-based mixed-order tetrahedral		
Infinite	**Degrees of Freedom**	**Application**
INFIN110	AZ, VOLT	Open boundary for an unbounded electric or magnetic field problem
INFIN111	MAG, AX, AY, AZ, VOLT	
Source	**Degrees of Freedom**	**Application**
SOURC36	---	Current source primitive (coil, bar, arc) for magnetostatic analysis
Circuit	**Degrees of Freedom**	**Application**
CIRCU94	VOLT, CURR	Resistor, capacitor, inductor, current source, voltage source for a piezoelectric-circuit and charge-based transient and harmonic analyses
CIRCU124	VOLT, CURR, EMF	Resistor, capacitor, inductor, current source, voltage source, stranded coil, 2-D massive coil, 3-D massive coil, mutual inductor, controlled sources for a current-based static, transient and harmonic analyses

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CIRCU125	VOLT	Diode for electric circuit analysis
Contact	**Degrees of Freedom**	**Application**
CONTA171 -- 2-D lower-order surface-to-surface contact	AZ, VOLT	Electric and magnetic contact
CONTA172 -- 2-D higher-order surface-to-surface contact		
CONTA173 -- 3-D lower-order surface-to-surface contact	MAG, VOLT	
CONTA174 -- 3-D higher-order surface-to-surface contact		
CONTA175 -- 2-D / 3-D node-to-surface contact	AZ, MAG,VOLT	

6.5. Elements for Diffusion Analysis

The elements listed in this section are available for a diffusion analysis.

Use the following diffusion elements for 2-D simulation:

- PLANE238 - 2-D 8-Node Diffusion Solid

Use the following diffusion elements for 3-D simulation:

- SOLID239 - 3-D 20-Node Diffusion Solid
- SOLID240 - 3-D 10-Node Tetrahedral Diffusion Solid

For more information, see Diffusion in the *Mechanical APDL Theory Reference*.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 7: Element Library

Following are descriptions of each element, which appear in numerical order.

Descriptions common to several elements appear in separate sections of General Element Features (p. 43) and are referenced where applicable.

Please read About This Reference (p. 1) and General Element Features (p. 43) *before* examining the element descriptions in Part I: Element Library (p. 117).

More detailed information about each element is available in the *Mechanical APDL Theory Reference*, which describes how the element input items (such as the real constants, material properties, KEYOPT switches, etc.) are used to produce the element output.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Part I: Element Library

SOLID5

3-D Coupled-Field Solid

MP ME <> <> <> <> EM <> PP EME MFS
Product Restrictions

SOLID5 Element Description

Although this legacy element is available for use in your analysis, ANSYS, Inc. recommends using a current-technology element such as SOLID226.

SOLID5 has a 3-D magnetic, thermal, electric, piezoelectric, and structural field capability with limited coupling between the fields. The element has eight nodes with up to six degrees of freedom at each node. Scalar potential formulations (reduced RSP, difference DSP, or general GSP) are available for modeling magnetostatic fields in a static analysis. When used in structural and piezoelectric analyses, SOLID5 has large deflection and stress stiffening capabilities. See SOLID5 in the *Mechanical APDL Theory Reference* for more details about this element. Coupled field elements with similar field capabilities are PLANE13, and SOLID98.

Figure 5.1: SOLID5 Geometry

SOLID5 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 5.1: SOLID5 Geometry (p. 119). The element is defined by eight nodes and the material properties. The type of units (MKS or user defined) is specified through the **EMUNIT** command. **EMUNIT** also determines the value of MUZERO. The **EMUNIT** defaults are MKS units and MUZERO = $4 \pi \times 10^{-7}$ Henries/meter. In addition to MUZERO, orthotropic relative permeability is specified through the MURX, MURY, and MURZ material property labels.

MGXX, MGYY, and MGZZ represent vector components of the coercive force for permanent magnet materials. The magnitude of the coercive force is the square root of the sum of the squares of the components. The direction of polarization is determined by the components MGXX, MGYY, and MGZZ.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Permanent magnet polarization directions correspond to the element coordinate directions. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems (p. 55). Nonlinear magnetic, piezoelectric, and anisotropic elastic properties are entered with the **TB** command as described in Material Models in the *Material Reference*. Nonlinear orthotropic magnetic properties may be specified with a combination of a B-H curve and linear relative permeability. The B-H curve will be used in each element coordinate direction where a zero value of relative permeability is specified. Only one B-H curve may be specified per material.

Various combinations of nodal loading are available for this element (depending upon the KEYOPT(1) value). Nodal loads are defined with the **D** and the **F** commands. With the **D** command, the *Lab* variable corresponds to the degree of freedom (UX, UY, UZ, TEMP, VOLT, MAG) and *VALUE* corresponds to the value (displacements, temperature, voltage, scalar magnetic potential). With the **F** command, the *Lab* variable corresponds to the force (FX, FY, FZ, HEAT, AMPS, FLUX) and *VALUE* corresponds to the value (force, heat flow, current or charge, magnetic flux).

Element loads are described in Nodal Loading (p. 49). Pressure, convection or heat flux (but not both), radiation, and Maxwell force flags may be input on the element faces indicated by the circled numbers in Figure 5.1: SOLID5 Geometry (p. 119) using the **SF** and **SFE** commands. Positive pressures act into the element. Surfaces at which magnetic forces are to be calculated may be identified by using the MXWF label on the surface load commands (no value is required.) A maxwell stress tensor calculation is performed at these surfaces to obtain the magnetic forces. These forces are applied in solution as structural loads. The surface flag should be applied to "air" elements adjacent to the body for which forces are required. Deleting the MXWF specification removes the flag.

The body loads, temperature, heat generation rate and magnetic virtual displacement may be input based on their value at the element's nodes or as a single element value [**BF** and **BFE**]. When the temperature degree of freedom is active (KEYOPT(1) = 0,1 or 8), applied body force temperatures [**BF**, **BFE**] are ignored. In general, unspecified nodal values of temperature and heat generation rate default to the uniform value specified with the **BFUNIF** or **TUNIF** commands. Calculated Joule heating (JHEAT) is applied in subsequent iterations as heat generation rate.

If the temperature degree of freedom is present, the calculated temperatures override any input nodal temperatures.

Air elements in which Local Jacobian forces are to be calculated may be identified by using nodal values of 1 and 0 for the MVDI label [**BF**]. See the *Low-Frequency Electromagnetic Analysis Guide* for details. These forces are not applied in solution as structural loads.

Current for the scalar magnetic potential options is defined with the SOURC36 element the command macro RACE, or through electromagnetic coupling. The various types of scalar magnetic potential solution options are defined with the **MAGOPT** command.

A summary of the element input is given in "SOLID5 Input Summary" (p. 120). A general description of element input is given in Element Input (p. 43).

SOLID5 Input Summary

Nodes

I, J, K, L, M, N, O, P

Degrees of Freedom

UX, UY, UZ, TEMP, VOLT, MAG if KEYOPT (1) = 0

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TEMP, VOLT, MAG if KEYOPT (1) = 1
UX, UY, UZ if KEYOPT (1) = 2
UX, UY, UZ, VOLT if KEYOPT(1) = 3
TEMP if KEYOPT (1) = 8
VOLT if KEYOPT (1) = 9
MAG if KEYOPT (1) = 10

Real Constants

None

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, EY, EZ, (PRXY, PRYZ, PRXZ or NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ),
DENS, GXY, GYZ, GXZ, ALPD, BETD, KXX, KYY, KZZ, C,
ENTH, MUZERO, MURX, MURY, MURZ, RSVX, RSVY, RSVZ,
MGXX, MGYY, MGZZ, PERX, PERY, PERZ)

Surface Loads

Pressure, Convection or Heat Flux (but not both), Radiation (using Lab = RDSF), and Maxwell Force Flags --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

Heat Generations --
HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P)

Magnetic Virtual Displacements --
VD(I), VD(J), VD(K), VD(L), VD(M), VD(N), VD(O), VD(P)

Electric Field --
EFX, EFY, EFZ. See "SOLID5 Assumptions and Restrictions" (p. 126).

Special Features

Adaptive descent
Birth and death
Large deflection
Stress stiffening

KEYOPT(1)

Element degrees of freedom:

0 --
UX, UY, UZ, TEMP, VOLT, MAG

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
TEMP, VOLT, MAG

2 --
UX, UY, UZ

3 --
UX, UY, UZ, VOLT

8 --
TEMP

9 --
VOLT

10 --
MAG

KEYOPT(3)
Extra shapes:

0 --
Include extra shapes

1 --
Do not include extra shapes

KEYOPT(5)
Extra element output:

0 --
Basic element printout

2 --
Nodal stress or magnetic field printout

SOLID5 Output Data

The solution output associated with the element is in two forms

- Nodal degree of freedom results included in the overall nodal solution
- Additional element output as shown in Table 5.1: SOLID5 Element Output Definitions (p. 123).

Several items are illustrated in Figure 5.2: SOLID5 Element Output (p. 123). The element stress directions are parallel to the element coordinate system. The reaction forces, heat flow, current, and magnetic flux at the nodes can be printed with the **OUTPR** command. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.2: SOLID5 Element Output

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 5.1: SOLID5 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Element nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Element material number	Y	Y
VOLU:	Element volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	3
PRES	P1 at nodes J, I, L, K; P2 at I, J, N, M; P3 at J, K, O, N; P4 at K, L, P, O; P5 at L, I, M, P; P6 at M, N, O, P	Y	Y
TEMP	Input Temperatures: T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	Y	Y
HGEN	Input Heat Generations: HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P)	Y	Y
S:X, Y, Z, XY, YZ, XZ	Component stresses	1	1
S:1, 2, 3	Principal stresses	1	1
S:INT	Stress intensity	1	1
S:EQV	Equivalent stress	1	1
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	1	1
EPEL:1, 2, 3	Principal elastic strains	1	-
EPEL:EQV	Equivalent elastic strains [4]	1	1
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	1	1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
EPTH:EQV	Equivalent thermal strains [4]	1	1
LOC	Output location (X, Y, Z)	1	1
MUX, MUY, MUZ	Magnetic permeability	1	1
H: X, Y, Z	Magnetic field intensity components	1	1
H:SUM	Vector magnitude of H	1	1
B:X, Y, Z	Magnetic flux density components	1	1
B:SUM	Vector magnitude of B	1	1
FJB	Lorentz magnetic force components (X, Y, Z)	1	-
FMX	Maxwell magnetic force components (X, Y, Z)	1	-
FVW	Virtual work force components (X, Y, Z)	1	1
FMAG:X, Y, Z	Combined (FJB or FMX) force components	-	1
EF:X, Y, Z	Electric field components (X, Y, Z)	1	1
EF:SUM	Vector magnitude of EF	1	1
JS:X, Y, Z	Source current density components	1	1
JSSUM	Vector magnitude of JS	1	1
JHEAT:	Joule heat generation per unit volume	1	1
D:X, Y, Z	Electric flux density components	1	1
D:SUM	Vector magnitude of D	1	1
UE, UD, UM	Elastic (UE), dielectric (UD), and electromechanical coupled (UM) energies	1	1
TG:X, Y, Z	Thermal gradient components	1	1
TG:SUM	Vector magnitude of TG	1	1
TF:X, Y, Z	Thermal flux components	1	1
TF:SUM	Vector magnitude of TF (heat flow rate/unit cross-section area)	1	1
FACE	Face label	2	2
AREA	Face area	2	2
NODES	Face nodes	2	-
HFILM	Film coefficient at each node of face	2	-
TBULK	Bulk temperature at each node of face	2	-
TAVG	Average face temperature	2	2
HEAT RATE	Heat flow rate across face by convection	2	2
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	2	-
HFLUX	Heat flux at each node of face	2	-
HFAVG	Average film coefficient of the face	2	2
TBAVG	Average face bulk temperature	-	2
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	2

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Element solution at the centroid printed out only if calculated (based on input data).

2. Nodal stress or magnetic field solution (only if KEYOPT(5) = 2). The solution results are repeated at each node and only if a surface load is input.

3. Available only at centroid as a ***GET** item.

4. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY).

Table 5.2: SOLID5 Item and Sequence Numbers (p. 125) lists output available through the **ETABLE** command using the Sequence Number method. The following notation is used in Table 5.2: SOLID5 Item and Sequence Numbers (p. 125):

Name
output quantity as defined in the Table 5.1: SOLID5 Element Output Definitions (p. 123)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J,...,P
sequence number for data at nodes I,J,...,P

FC*n*
sequence number for solution items for element Face *n*

Table 5.2: SOLID5 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input									
	Item	**E**	**I**	**J**	**K**	**L**	**M**	**N**	**O**	**P**
P1	SMISC	-	2	1	4	3	-	-	-	-
P2	SMISC	-	5	6	-	-	8	7	-	-
P3	SMISC	-	-	9	10	-	-	12	11	-
P4	SMISC	-	-	-	13	14	-	-	16	15
P5	SMISC	-	18	-	-	17	19	-	-	20
P6	SMISC	-	-	-	-	-	21	22	23	24
MUX	NMISC	1	-	-	-	-	-	-	-	-
MUY	NMISC	2	-	-	-	-	-	-	-	-
MUZ	NMISC	3	-	-	-	-	-	-	-	-
FVWX	NMISC	4	-	-	-	-	-	-	-	-
FVWY	NMISC	5	-	-	-	-	-	-	-	-
FVWZ	NMISC	6	-	-	-	-	-	-	-	-
FVWSUM	NMISC	7	-	-	-	-	-	-	-	-
UE	NMISC	16	-	-	-	-	-	-	-	-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input									
	Item	E	I	J	K	L	M	N	O	P
UD	NMISC	17	-	-	-	-	-	-	-	-
UM	NMISC	18	-	-	-	-	-	-	-	-

Output Quantity Name	ETABLE and ESOL Command Input						
	Item	FC1	FC2	FC3	FC4	FC5	FC6
AREA	NMISC	19	25	31	37	43	49
HFAVG	NMISC	20	26	32	38	44	50
TAVG	NMISC	21	27	33	39	45	51
TBAVG	NMISC	22	28	34	40	46	52
HEAT RATE	NMISC	23	29	35	41	47	53
HFLXAVG	NMISC	24	30	36	42	48	54

SOLID5 Assumptions and Restrictions

- The element requires an iterative solution for field coupling (displacement, temperature, electric, magnetic, but not piezoelectric)
- When using SOLID5 with SOURC36 elements, the source elements must be placed so that the resulting Hs field fulfills boundary conditions for the total field.
- The element must not have a zero volume or a zero length side. This occurs most frequently when the element is not numbered properly.
- Elements may be numbered either as shown in Figure 5.1: SOLID5 Geometry (p. 119) or may have the planes IJKL and MNOP interchanged.
- A prism shaped element may be formed by defining duplicate node numbers as described in Degenerated Shape Elements (p. 39).
- The difference scalar magnetic potential option is restricted to singly-connected permeable regions, so that as $\mu \rightarrow \infty$ in these regions, the resulting field $H \rightarrow 0$. The reduced scalar and general scalar potential options do not have this restriction.
- At a free surface of the element (i.e., not adjacent to another element and not subjected to a boundary constraint), the normal component of magnetic flux density (B) is assumed to be zero.
- Temperatures and heat generation rates, if internally calculated, include any user defined heat generation rates.
- The thermal, electrical, magnetic, and structural terms are coupled through an iterative procedure.
- Large deflection capabilities available for KEYOPT(1) = 2 and 3 are not available for KEYOPT(1) = 0.
- Stress stiffening is available for KEYOPT(1) = 0, 2, and 3.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Do not constrain all VOLT DOFs to the same value in a piezoelectric analysis (KEYOPT(1) = 0 or 3). Perform a pure structural analysis instead (KEYOPT(1) = 2).
- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, the elements must have the same reaction solution for the VOLT DOF. Elements that have an electric charge reaction solution must all have the same electric charge reaction sign. For more information, see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*.
- The electric field body load is not used during solution and is applicable only to POST1 charged particle tracing.
- In an MSP analysis, avoid using a closed domain and use an open domain, closed with natural flux parallel boundary conditions on the MAG degree of freedom, or infinite elements. If you use a closed domain, you may see incorrect results when the formulation is applied using SOLID5, SOLID96, or SOLID98 elements and the boundary conditions are not satisfied by the Hs field load calculated by the Biot-Savart procedure based on SOURC36 current source primitive input.
- If you used the MAG degree of freedom, you cannot restart a job in ANSYS Mechanical using `Jobname.DB` and `Jobname.ESAV` files that were created by ANSYS Multiphysics.
- This element cannot be used in a distributed solution.

SOLID5 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Mechanical

Unless the Emag option is enabled, the following restrictions apply:

- This element does not have magnetic capability.
- The MAG degree of freedom is not active.
- KEYOPT(1) cannot be set to 10. If KEYOPT(1) = 0 (default) or 1, the MAG degree of freedom is inactive.
- KEYOPT(1) cannot be set to 9.
- The magnetic material properties (MUZERO, MUR_, MG__, and the BH data table) are not allowed.
- The Maxwell force flags and magnetic virtual displacements body loads are not applicable.

ANSYS Emag

- This element has only magnetic and electric field capability, and does not have structural, thermal, or piezoelectric capability.
- The only active degrees of freedom are MAG and VOLT.
- KEYOPT(1) settings of 0, 1, 2, 3 and 8 are invalid.
- The only allowable material properties are the magnetic and electric properties (MUZRO through PERZ, plus the BH data table).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The only applicable surface loads are Maxwell force flags. The only applicable body loads are temperatures (for material property evaluation only) and magnetic virtual displacements.
- The element does not have stress stiffening or birth and death features.
- KEYOPT(3) is not applicable.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

INFIN9

2-D Infinite Boundary

MP ME <> <> <> <> EM <> PP EME <>
Product Restrictions

INFIN9 Element Description

Although this legacy element is available for use in your analysis, ANSYS, Inc. recommends using a current-technology element such as INFIN110.

INFIN9 is used to model an open boundary of a 2-D planar unbounded field problem. The element has two nodes with a magnetic vector potential or temperature degree of freedom at each node. The enclosed element type can be the PLANE13 or PLANE53 magnetic elements, or the PLANE55, PLANE77, and PLANE35 thermal elements. With the magnetic degree of freedom (AZ) the analysis may be linear or nonlinear, static or dynamic. With the thermal degree of freedom steady-state or transient analysis (linear or nonlinear) may be done. See INFIN9 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 9.1: INFIN9 Geometry

INFIN9 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 9.1: INFIN9 Geometry (p. 129), and a typical application is shown in Figure 9.2: INFIN9 Element Usage (p. 131). The element is defined by two nodes and the material properties. Nonzero material properties must be defined. The element x-axis is oriented along the length of the element from node I toward node J.

The coefficient matrix of this boundary element is, in general, unsymmetric. The matrix is made symmetric by averaging the off-diagonal terms to take advantage of a symmetric solution with a slight decrease in accuracy. KEYOPT(2) can be used to prevent an unsymmetric matrix from being made symmetric.

A summary of the element input is given in "INFIN9 Input Summary" (p. 130). A general description of element input is given in Element Input (p. 43).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

INFIN9 Input Summary

Nodes
I, J

Degrees of Freedom

AZ if KEYOPT (1) = 0
TEMP if KEYOPT (1) = 1

Real Constants
None

Material Properties

EMUNIT command: MUZERO if KEYOPT (1) = 0 (has default value for MKS units or can be set manually), KXX if KEYOPT (1) = 1

Surface Loads
None

Body Loads
None

Special Features
None

KEYOPT(1)
Element degree of freedom:

0 --
Magnetic option (AZ degree of freedom)

1 --
Thermal option (TEMP degree of freedom)

KEYOPT(2)
Coefficient matrix formation:

0 --
Make the coefficient matrix symmetric

1 --
Coefficient matrix is used as generated (symmetric or unsymmetric, depending on the problem)

INFIN9 Output Data

The boundary element has no output of its own since it is used only to provide a semi-infinite boundary condition to a model consisting of other elements.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 9.2: INFIN9 Element Usage

INFIN9 Assumptions and Restrictions

- The boundary element assumes a straight line segment, the length of which must be nonzero.
- The semi-infinite sector is assumed to be bound on three sides by the boundary element, by a semi-infinite radial line from the global coordinate system origin through node I, and by a semi-infinite radial line from the global coordinate system origin through node J. The boundary element should be as normal as possible to the radial lines.
- Acute or wide intersection angles should be avoided by "filling-in" the model with the other elements so that the line of boundary elements around the model is smooth and concave when viewed from the global coordinate system origin.
- The boundary element must lie "against" an enclosed element (that is, share the same nodes).
- The origin of the global coordinate system must be inside the model and as centrally located as possible.
- The line of boundary elements should be located away from the region of interest of the enclosed elements for better accuracy.
- The line of boundary elements need not totally surround the model.
- The exterior semi-infinite domain is assumed to be homogeneous, isotropic, and linear without containing any sources or sinks.
- An axisymmetric option is not available.
- The element cannot be deactivated with the **EKILL** command.
- The element assumes that the degree of freedom (DOF) value at infinity is *always* zero (0.0). That is, the DOF value at infinity is *not* affected by **TUNIF**, **D**, or other load commands.
- When used in a model with the higher-order elements PLANE53, PLANE77, or PLANE35, the midside nodes of these elements must be removed at the interface with INFIN9 [**EMID**].

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If KEYOPT(2) = 1, the matrices are presumed to be unsymmetric.
- This element cannot be used in a distributed solution.

INFIN9 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Mechanical

Unless the Emag option is enabled, the following restrictions apply:

- This element does not have magnetic field capability.
- The AZ degree of freedom is not active.
- KEYOPT(1) defaults to 1 (TEMP) instead of 0 and cannot be changed.
- The material property MUZERO is not allowed.

ANSYS Emag

- This element has only magnetic field capability, and does not have thermal capability.
- The only active degree of freedom is AZ.
- The only allowable material property is MUZERO.
- KEYOPT(1) can only be set to 0 (default).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LINK11

Linear Actuator

MP ME ST <> <> <> <> <> PP EME MFS
Product Restrictions

LINK11 Element Description

LINK11 may be used to model hydraulic cylinders and other applications undergoing large rotations. The element is a uniaxial tension-compression element with three degrees of freedom at each node: translations in the nodal x, y, and z directions. No bending or twist loads are considered. See LINK11 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 11.1: LINK11 Geometry

LINK11 Input Data

The geometry and node locations for the element are shown in Figure 11.1: LINK11 Geometry (p. 133). The element is defined by two nodes, a stiffness, viscous damping, and mass. The element initial length L_o and orientation are determined from the node locations.

Element loads are described in Nodal Loading (p. 49). The stroke (length) is defined through the surface load input using the PRES label. The stroke is relative to the zero force position of the element. A force may be defined in the same manner as an alternate to the stroke.

A summary of the element input is given below. A general description of element input is given in Element Input (p. 43).

LINK11 Input Summary

Nodes

I, J

Degrees of Freedom

UX, UY, UZ

Real Constants

K - Stiffness (force/length)
C - Viscous damping coefficient (force*time/length)
M - Mass (force*time^2/length)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Material Properties

MP command: BETD, ALPD

Surface Loads

Pressures --

face 1 - Stroke
face 2 - Axial Force

Body Loads

None

Special Features

Birth and death
Large deflection
Stress stiffening

KEYOPTs

None

LINK11 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal displacement solution
- Additional element output as shown in Table 11.1: LINK11 Element Output Definitions (p. 134).

A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 11.1: LINK11 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
ILEN	Initial element length	Y	Y
CLEN	Current element length (this time step)	Y	Y
FORCE	Axial force (spring force)	Y	Y
DFORCE	Damping force	Y	Y
STROKE	Applied stroke (element load)	Y	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
MSTROKE	Measured stroke	Y	Y

Table 11.2: LINK11 Item and Sequence Numbers (p. 135) lists output available through the **ETABLE** command using the Sequence Number method. The following notation is used in Table 11.2: LINK11 Item and Sequence Numbers (p. 135):

Name
output quantity as defined in Table 11.1: LINK11 Element Output Definitions (p. 134)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 11.2: LINK11 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FORCE	SMISC	1
ILEN	NMISC	1
CLEN	NMISC	2
STROKE	NMISC	3
MSTROKE	NMISC	4
DFORCE	NMISC	5

LINK11 Assumptions and Restrictions

- The element must not have a zero length.
- The element assumes a straight line, axially loaded at the ends.
- A twist (torsion) about the element x-axis (defined from node I to node J) has no effect.
- No bending of the element is considered, as in a pin-jointed structure.
- The mass is equally divided between the nodes.
- Only the lumped mass matrix is available.
- Surface load pressure indicators are not displayed for element or node plots.

LINK11 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

PLANE13

2-D Coupled-Field Solid

MP ME <> <> <> <> EM <> PP EME MFS
Product Restrictions

PLANE13 Element Description

Although this legacy element is available for use in your analysis, ANSYS, Inc. recommends using a current-technology element such as PLANE223.

PLANE13 has a 2-D magnetic, thermal, electrical, piezoelectric, and structural field capability with limited coupling between the fields. PLANE13 is defined by four nodes with up to four degrees of freedom per node. The element has nonlinear magnetic capability for modeling B-H curves or permanent magnet demagnetization curves. PLANE13 has large deflection and stress stiffening capabilities. When used in purely structural analyses, PLANE13 also has large strain capabilities. See PLANE13 in the *Mechanical APDL Theory Reference* for more details about this element. Other coupled-field elements are SOLID5, and SOLID98.

Figure 13.1: PLANE13 Geometry

PLANE13 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 13.1: PLANE13 Geometry (p. 137). The element input data includes four nodes and magnetic, thermal, electrical, and structural material properties. The type of units (MKS or user defined) is specified through the **EMUNIT** command. **EMUNIT** also determines the value of MUZERO. The **EMUNIT** defaults are MKS units and MUZRO = $4 \pi \times 10^{-7}$ henries/meter. In addition to MUZERO, orthotropic relative permeability is specified through the MURX and MURY material property labels.

MGXX and MGYY represent vector components of the coercive force for permanent magnet materials. The magnitude of the coercive force is the square root of the sum of the squares of the components. The direction of polarization is determined by the components MGXX and MGYY. Permanent magnet polarization and orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems (p. 55). Nonlinear magnetic B-H, piezoelectric, and anisotropic elastic properties are entered with the **TB** command as described in Material Models in the *Material Reference*. Nonlinear orthotropic magnetic properties may be specified with a combination of a B-H curve and linear relative permeability. The B-H curve will be used in each

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

element coordinate direction where a zero value of relative permeability is specified. Only one B-H curve may be specified per material.

Various combinations of nodal loading are available for this element (depending upon the KEYOPT(1) value). Nodal loads are defined with the **D** and the **F** commands. Nodal forces, if any, should be input per unit of depth for a plane analysis and on a full 360° basis for an axisymmetric analysis.

Element loads are described in Nodal Loading (p. 49). Pressure, convection or heat flux (but not both), radiation, and Maxwell force flags may be input on the element faces indicated by the circled numbers in Figure 13.1: PLANE13 Geometry (p. 137) using the **SF** and **SFE** commands. Positive pressures act into the element. Surfaces at which magnetic forces are to be calculated are identified by using the MXWF label on the surface load commands (no value is required). A maxwell stress tensor calculation is performed at these surfaces to obtain the magnetic forces. These forces are applied in solution as structural loads. The surface flag should be applied to "air" elements adjacent to the body for which forces are required. Deleting the MXWF specification removes the flag.

Body loads - temperature, heat generation rate, and magnetic virtual displacement - may be input at the element's nodes or as a single element value [**BF**, **BFE**]. Source current density loads may be applied to an area [**BFA**] or input as an element value [**BFE**]. When the temperature degree of freedom is active (KEYOPT(1) = 2 or 4), applied body force temperatures [**BF**, **BFE**] are ignored. In general, unspecified nodal temperatures and heat generation rates default to the uniform value specified with the **BFUNIF** or **TUNIF** command. Heat generation from Joule heating is applied in Solution as thermal loading for static and transient analyses.

If the temperature degree of freedom is present, the calculated temperatures override any input nodal temperatures.

Air elements in which local Jacobian forces are to be calculated may be identified by using nodal values of 1 and 0 for the MVDI label [**BF**]. See the *Low-Frequency Electromagnetic Analysis Guide* for details. These forces are not applied in solution as structural loads.

A summary of the element input is given in "PLANE13 Input Summary" (p. 138). A general description of element input is given in Element Input (p. 43). For axisymmetric applications see Harmonic Axisymmetric Elements (p. 83).

PLANE13 Input Summary

Nodes

I, J, K, L

Degrees of Freedom

AZ if KEYOPT (1) = 0
TEMP if KEYOPT (1) = 2
UX, UY if KEYOPT (1) = 3
UX, UY, TEMP, AZ if KEYOPT (1) = 4
VOLT, AZ if KEYOPT (1) = 6
UX, UY, VOLT if KEYOPT (1) = 7

Real Constants

None

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, EY, EZ, (PRXY, PRYZ, PRXZ or NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ, (or CTEX, CTEY,CTEZ *or* THSX, THSY,THSZ),
DENS, GXY, BETD, ALPD,
KXX, KYY, C, ENTH, MUZERO, MURX,
MURY, RSVZ, MGXX, MGYY, PERX, PERY

Surface Loads

Pressure, Convection or Heat Flux (but not both), Radiation (using Lab = RDSF), and Maxwell Force Flags--
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L)

Heat Generations --
HG(I), HG(J), HG(K), HG(L)

Magnetic Virtual Displacements --
VD(I), VD(J), VD(K), VD(L)

Source Current Density --

spare, spare, JSZ(I), PHASE(I), spare, spare,
JSZ(J), PHASE(J), spare, spare, JSZ(K), PHASE(K),
spare, spare, JSZ(L), PHASE(L)

Special Features

Adaptive descent
Birth and death
Large deflection
Large strain
Stress stiffening

KEYOPT(1)

Element degrees of freedom:

0 --
AZ degree of freedom

2 --
TEMP degree of freedom

3 --
UX, UY degrees of freedom

4 --
UX, UY, TEMP, AZ degrees of freedom

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6 --
VOLT, AZ degrees of freedom

7 --
UX, UY, VOLT degrees of freedom

KEYOPT(2)
Extra shapes:

0 --
Include extra shapes

1 --
Do not include extra shapes

KEYOPT(3)
Element behavior:

0 --
Plane strain (with structural degrees of freedom)

1 --
Axisymmetric

2 --
Plane stress (with structural degrees of freedom)

KEYOPT(4)
Element coordinate system defined:

0 --
Element coordinate system is parallel to the global coordinate system

1 --
Element coordinate system is based on the element I-J side

KEYOPT(5)
Extra element output:

0 --
Basic element printout

1 --
Repeat basic solution for all integration points

2 --
Nodal stress printout

PLANE13 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in Table 13.1: PLANE13 Element Output Definitions (p. 141).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Several items are illustrated in Figure 13.2: PLANE13 Element Output (p. 141). The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

Figure 13.2: PLANE13 Element Output

Because of different sign conventions for Cartesian and polar coordinate systems, magnetic flux density vectors point in opposite directions for planar (KEYOPT(3) = 0) and axisymmetric (KEYOPT(3) =1) analyses.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 13.1: PLANE13 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	3
PRES	P1 at nodes J, I; P2 at K, J; P3 at L, K; P4 at I, L	Y	Y
TEMP	Input temperatures T(I), T(J), T(K), T(L)	Y	Y
HGEN	Input heat generations HG(I), HG(J), HG(K), HG(L)	Y	-
S:X, Y, Z, XY	Stresses (SZ = 0.0 for plane stress elements)	1	1
S:1, 2, 3	Principal stresses	1	1
S:INT	Stress intensity	1	1
S:EQV	Equivalent stress	1	1
EPEL:X, Y, Z, XY	Elastic strains	1	1
EPEL:1, 2, 3	Principal elastic strains	1	-
EPEL:EQV	Equivalent elastic strain [4]	-	1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
EPTH:X, Y, Z, XY	Average thermal strains	1	1
EPTH:EQV	Equivalent thermal strain [4]	-	1
TG:X, Y, SUM	Thermal gradient components and vector sum	1	1
TF:X, Y, SUM	Thermal flux (heat flow rate/cross-sectional area) components and vector sum	1	1
EF:X, Y	Electric field components (X, Y)	1	1
EF:SUM	Vector magnitude of EF	1	1
D:X, Y	Electric flux density components (X, Y)	1	1
D:SUM	Vector magnitude of D	1	1
UE, UD, UM	Elastic (UE), dielectric (UD), and electromechanical coupled (UM) energies	1	1
LOC	Output location (X, Y)	1	-
MUX, MUY	Magnetic permeability	1	1
H:X, Y	Magnetic field intensity components	1	1
H:SUM	Vector magnitude of H	1	1
B:X, Y	Magnetic flux density components	1	1
B:SUM	Vector magnitude of B	1	1
JSZ	Source current density, available for static analysis only	1	1
JTZ	Total current density	1	1
JHEAT:	Joule heat generation per unit volume	1	1
FJB(X, Y)	Lorentz force components	1	1
FMX(X, Y)	Maxwell force components	1	1
FVW(X, Y)	Virtual work force components	1	1
FMAG:X, Y	Combined (FJB and FMX) force components	-	1
FACE	Face label	2	2
AREA	Face area	2	2
NODES	Face nodes	2	-
HFILM	Film coefficient at each node of face	2	-
TBULK	Bulk temperature at each node of face	2	-
TAVG	Average face temperature	2	2
HEAT RATE	Heat flow rate across face by convection	2	2
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	2	-
HFLUX	Heat flux at each node of face	2	-
HFAVG	Average film coefficient of the face	2	2
TBAVG	Average face bulk temperature	-	2
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	2
TJB(Z)	Lorentz torque about global Cartesian +Z axis	1	1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
TMX(Z)	Maxwell torque about global Cartesian +Z axis	1	1
TVW(Z)	Virtual work torque about global Cartesian +Z axis	1	1

1. Solution values are output only if calculated (based on input data).

   For harmonic analysis, joule losses (JHEAT), forces (FJB(X, Y), FMX(X, Y), FVW(X, Y)), and torque (TJB(Z), TMX(Z), TVW(Z)) represent time-average values. These values are stored in the "Real" data set. The macros **POWERH**, **FMAGSUM**, and **TORQSUM** can be used to retrieve this data.

2. Available only if a surface load is input.

3. Available only at centroid as a ***GET** item.

4. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY).

Table 13.2: PLANE13 Miscellaneous Element Output

Description	Names of Items Output	O	R
Integration Pt. Solution	SINT, SEQV, EPEL, S, MUX, MUY, H, HSUM, B, BSUM	1	-
Nodal Solution	SINT, SEQV, S, H, HSUM, B, BSUM	2	-

1. Output at each integration point, if KEYOPT(5) = 1.

2. Output at each node, if KEYOPT(5) = 2.

Note

JT represents the total measurable current density in a conductor, including eddy current effects, and velocity effects if calculated.

For axisymmetric solutions with KEYOPT(4) = 0, the X and Y directions correspond to the radial and axial directions, respectively. The X, Y, Z, and XY stress output correspond to the radial, axial, hoop, and in-plane shear stresses, respectively.

For harmonic analysis, joule losses (JHEAT), forces (FJB(X, Y), FMX(X, Y), FVW(X, Y)), and torque (TJB(Z), TMX(Z), TVW(Z)) represent time-average values. These values are stored in the "Real" data set. The macros POWERH, FMAGSUM, and TORQSUM can be used to retrieve this data.

Table 13.3: PLANE13 Item and Sequence Numbers (p. 144) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) of the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) of this manual for more information. The following notation is used in Table 13.3: PLANE13 Item and Sequence Numbers (p. 144):

Name
output quantity as defined in the Table 13.1: PLANE13 Element Output Definitions (p. 141)

Item
predetermined Item label for **ETABLE** command

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

E
sequence number for single-valued or constant element data

I,J,K,L
sequence number for data at nodes I, J, K, L

FC*N*
sequence number for solution items for element Face *N*

Table 13.3: PLANE13 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
JSZ	SMISC	1	-	-	-	-
P1	SMISC	-	4	3	-	-
P2	SMISC	-	-	6	5	-
P3	SMISC	-	-	-	8	7
P4	SMISC	-	9	-	-	10
MUX	NMISC	1	-	-	-	-
MUY	NMISC	2	-	-	-	-
FVWX	NMISC	3	-	-	-	-
FVWY	NMISC	4	-	-	-	-
FVWSUM	NMISC	5	-	-	-	-
JTZ	NMISC	7	-	-	-	-
UE	NMISC	8	-	-	-	-
UD	NMISC	9	-	-	-	-
UM	NMISC	10	-	-	-	-
TJB(Z)	NMISC	35	-	-	-	-
TMX(Z)	NMISC	36	-	-	-	-
TVW(Z)	NMISC	37	-	-	-	-

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	FC1	FC2	FC3	FC4
AREA	NMISC	11	17	23	29
HFAVG	NMISC	12	18	24	30
TAVG	NMISC	13	19	25	31
TBAVG	NMISC	14	20	26	32
HEAT RATE	NMISC	15	21	27	33
HFLXAVG	NMISC	16	22	28	34

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PLANE13 Assumptions and Restrictions

- The element requires an iterative solution for field coupling (displacement, temperature, electric, magnetic, but not piezoelectric)
- The area of the element must be positive.
- The element must lie in a global X-Y plane as shown in Figure 13.1: PLANE13 Geometry (p. 137) and the Y-axis must be the axis of symmetry for axisymmetric analyses.
- An axisymmetric structure should be modeled in the +X quadrants.
- For structural and piezoelectric problems, the extra displacement and VOLT shapes are automatically deleted for triangular elements so that a constant strain element results.
- Transient magnetic analyses should be performed in a nonlinear transient dynamic analysis.
- A skin-effect analysis (where eddy current formation is permitted in conducting regions with impressed current loading) is performed by using KEYOPT(1) = 6, specifying a resistivity, and coupling all VOLT degrees of freedom for elements in each of such regions. This is valid for both planar and axisymmetric models.
- Current density loading (**BFE**,,JS) is only valid for the AZ option (KEYOPT(1) = 0). For the VOLT, AZ option (KEYOPT(1) = 6) use **F**,,AMPS.
- When this element does not have the VOLT degree of freedom (KEYOPT(1) = 4), for a harmonic or transient analysis, its behavior depends on the applied load. For a **BFE**,,JS load, the element acts as a stranded conductor. Without **BFE**,,JS loads, it acts as a solid conductor modeling eddy current effects.

  In this respect, PLANE13 (and PLANE53) are not like the 3-D elements SOLID97, SOLID236, and SOLID237. When SOLID97, SOLID236, and SOLID237 do not have the VOLT degree of freedom, they act as stranded conductors.
- Do not constrain all VOLT DOFs to the same value in a piezoelectric analysis (KEYOPT(1) = 7). Perform a pure structural analysis instead (KEYOPT(1) = 3).
- Permanent magnets are not permitted in a harmonic analysis.
- If a model has at least one element with piezoelectric degrees of freedom (displacements and VOLT) activated, then all elements where a VOLT degree of freedom is needed must be one of the piezoelectric types, and they must all have the piezoelectric degrees of freedom activated. If the piezoelectric effect is not desired in these elements, simply define very small piezoelectric material properties for them.
- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, the elements must have the same reaction solution for the VOLT DOF. Elements that have an electric charge reaction solution must all have the same electric charge reaction sign. For more information, see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*.
- This element cannot be used in a distributed solution.

PLANE13 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Mechanical

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Unless the Emag option is enabled, the following restrictions apply:

- This element has only structural, thermal, or piezoelectric capability, and does not have magnetic capability.
- The AZ degree of freedom is not active.
- KEYOPT(1) defaults to 4 (UX, UY, TEMP) instead of 0, and cannot be set to 0. If set to 4 or 6, the AZ degree of freedom is not active.
- The magnetic and electric material properties (MUZERO, MUR_, MG__, and the BH data table) are not allowed.
- The Maxwell force flags surface loads are not applicable.

ANSYS Emag

- This element has only magnetic and electric field capability, and does not have structural, thermal, or piezoelectric capability.
- The only active degrees of freedom are AZ and VOLT.
- The only allowable material properties are the magnetic and electric properties (MUZRO through PERY, plus the BH data table).
- The only applicable surface loads are Maxwell force flags. The heat generation body loads are not applicable. The temperature body load is only used for material property evaluation.
- The element does not allow any special features.
- KEYOPT(1) can only be set to 0 (default) or 6. KEYOPT(3) = 2 is not applicable.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

COMBIN14

Spring-Damper

MP ME ST PR PRN DS <> <> PP EME MFS
Product Restrictions

COMBIN14 Element Description

COMBIN14 has longitudinal or torsional capability in 1-D, 2-D, or 3-D applications. The longitudinal spring-damper option is a uniaxial tension-compression element with up to three degrees of freedom at each node: translations in the nodal x, y, and z directions. No bending or torsion is considered. The torsional spring-damper option is a purely rotational element with three degrees of freedom at each node: rotations about the nodal x, y, and z axes. No bending or axial loads are considered.

The spring-damper element has no mass. Masses can be added by using the appropriate mass element (see MASS21). The spring or the damping capability may be removed from the element. See COMBIN14 in the *Mechanical APDL Theory Reference* for more details about this element. A general spring or damper is also available in the stiffness matrix element (MATRIX27). Another spring-damper element (having its direction of action determined by the nodal coordinate directions) is COMBIN40.

Figure 14.1: COMBIN14 Geometry

2-D elements must lie in a z = constant plane

COMBIN14 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 14.1: COMBIN14 Geometry (p. 147). The element is defined by two nodes, a spring constant (k) and damping coefficients $(c_v)_1$ and $(c_v)_2$. The damping capability is not used for static or undamped modal analyses. The longitudinal spring constant should have units of Force / Length, the damping coefficient units are Force * Time / Length. The torsional spring constant and damping coefficient have units of Force * Length / Radian and Force * Length * Time / Radian, respectively. For a 2-D axisymmetric analysis, these values should be on a full 360° basis.

The damping portion of the element contributes only damping coefficients to the structural damping matrix. The damping force (F) or torque (T) is computed as:

$F_x = - c_v du_x/dt$ or $T_\theta = - c_v d\,\theta/dt$

where c_v is the damping coefficient given by $c_v = (c_v)_1 + (c_v)_2 v$.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

v is the velocity calculated in the previous substep. The second damping coefficient $(c_v)_2$ is available to produce a nonlinear damping effect characteristic of some fluid environments. If $(c_v)_2$ is input (as real constant CV2), KEYOPT(1) must be set to 1.

KEYOPT(2) = 1 through 6 is used for defining the element as a one-dimensional element. With these options, the element operates in the nodal coordinate system (see Elements That Operate in the Nodal Coordinate System (p. 57)). The KEYOPT(2) = 7 and 8 options allow the element to be used in a thermal or pressure analysis.

A preload in the spring may be specified in one of two ways, either through an initial (force-free) length (ILENGTH) or an initial force (IFORCE) input. Only one of the input options may be used to define the preload. If the initial length is different than the input length defined by the nodal coordinates, a preload is presumed to exist. If an initial force is given, a negative value indicates the spring is initially in compression and a positive value indicates tension. For the 3-D torsional spring option (KEYOPT(3) = 1), ILENGTH is interpreted as the initial number of turns (rotations) in the spring (the spring is pre-wound) and IFORCE is the torque preload in the spring. The right-hand rule from node I to node J is used to define positive and negative turns as well as positive and negative torque. In a nonlinear analysis, the preload is ramped in the first load step if **KBC**,0 is set.

A summary of the element input is given in "COMBIN14 Input Summary" (p. 148). A general description of element input is given in Element Input (p. 43).

COMBIN14 Input Summary

Nodes

I, J

Degrees of Freedom

UX, UY, UZ if KEYOPT (3) = 0
ROTX, ROTY, ROTZ if KEYOPT (3) = 1
UX, UY if KEYOPT (3) = 2
see list below if KEYOPT(2) > 0

Real Constants

K, CV1, CV2, (Blank), (Blank), ILENGTH, IFORCE

See Table 14.1: COMBIN14 Real Constants (p. 150) for a description of the real constants.

Note

In a full harmonic analysis, real constants K and CV1 can be defined as table parameters using the frequency as primary variable (*FREQ* on the ***DIM** command).

Material Properties

MP command: BETD

Surface Loads

None

Body Loads

None

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Special Features

Birth and death
Large deflection
Linear perturbation
Nonlinearity (if CV2 is not zero)
Stress stiffening

KEYOPT(1)
Solution type:

0 --
Linear Solution (default)

1 --
Nonlinear solution (required if CV2 is nonzero)

KEYOPT(2)
Degree-of-freedom selection for 1-D behavior:

0 --
Use KEYOPT(3) options

1 --
1-D longitudinal spring-damper (UX degree of freedom)

2 --
1-D longitudinal spring-damper (UY degree of freedom)

3 --
1-D longitudinal spring-damper (UZ degree of freedom)

4 --
1-D Torsional spring-damper (ROTX degree of freedom)

5 --
1-D Torsional spring-damper (ROTY degree of freedom)

6 --
1-D Torsional spring-damper (ROTZ degree of freedom)

7 --
Pressure degree of freedom element

8 --
Temperature degree of freedom element

Note

KEYOPT(2) overrides KEYOPT(3)

KEYOPT(3)
Degree-of-freedom selection for 2-D and 3-D behavior:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0 --
3-D longitudinal spring-damper

1 --
3-D torsional spring-damper

2 --
2-D longitudinal spring-damper (2-D elements must lie in an X-Y plane)

Table 14.1: COMBIN14 Real Constants

No.	Name	Description
1	K	Spring constant
2	CV1	Damping coefficient
3	CV2	Damping coefficient (KEYOPT(1) must be set to 1)
4, 5	(Blank)	- -
6	ILENGTH	Initial force-free length (Initial number of turns if torsional spring (KEYOPT(3) = 1))
7	IFORCE	Initial force (or torque if torsional spring (KEYOPT(3) = 1))

COMBIN14 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 14.2: COMBIN14 Element Output Definitions (p. 151).

Several items are illustrated in Figure 14.2: COMBIN14 Stress Output (p. 150). A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

Figure 14.2: COMBIN14 Stress Output

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 14.2: COMBIN14 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
XC, YC, ZC	Location where results are reported	Y	1
FORC or TORQ	Spring force or moment	Y	Y
STRETCH or TWIST	Stretch of spring or twist of spring (radians)	Y	Y
RATE	Spring constant	Y	Y
VELOCITY	Velocity	-	Y
DAMPING FORCE or TORQUE	Damping force or moment (zero unless **ANTYPE**,TRANS and damping present)	Y	Y

1. Available only at centroid as a ***GET** item.

Table 14.3: COMBIN14 Item and Sequence Numbers (p. 151) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) of the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) of this manual for more information. The following notation is used in Table 14.3: COMBIN14 Item and Sequence Numbers (p. 151):

Name
output quantity as defined in the Table 14.2: COMBIN14 Element Output Definitions (p. 151)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 14.3: COMBIN14 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FORC	SMISC	1
STRETCH	NMISC	1
VELOCITY	NMISC	2
DAMPING FORCE	NMISC	3

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

COMBIN14 Assumptions and Restrictions

- If KEYOPT(2) is zero, the length of the spring-damper element must not be zero (that is, nodes I and J should not be coincident, since the node locations determine the spring orientation).
- The longitudinal spring element stiffness acts only along its length. The torsion spring element stiffness acts only about its length, as in a torsion bar.
- The element allows only a uniform stress in the spring.
- In a thermal analysis, the temperature or pressure degree of freedom acts in a manner analogous to the displacement.
- Only the KEYOPT(2) = 0 option supports stress stiffening or large deflection. Also, if KEYOPT(3) = 1 (torsion) is used with large deflection, the coordinates will not be updated.
- The spring or the damping capability may be deleted from the element by setting K or CV equal to zero, respectively.
- If CV2 is not zero, the element is nonlinear and requires an iterative solution (KEYOPT(1) = 1).
- The preload may not change after the first load step. Any changes are ignored.
- The preload is ignored in modal and harmonic analyses.

The restrictions described below only apply if KEYOPT(2) is greater than zero.

- If KEYOPT(2) is greater than zero, the element has only one degree of freedom. This degree of freedom is specified in the nodal coordinate system and is the same for both nodes (see Elements That Operate in the Nodal Coordinate System (p. 57)). If the nodal coordinate systems are rotated relative to each other, the same degree of freedom may be in different directions (thereby giving possibly unexpected results). The element, however, assumes only a 1-D action. Nodes I and J, then, may be anywhere in space (preferably coincident).
- For noncoincident nodes and KEYOPT(2) = 1, 2, or 3, no moment effects are included. That is, if the nodes are offset from the line of action, moment equilibrium may not be satisfied.
- The element is defined such that a positive displacement of node J relative to node I tends to stretch the spring. If, for a given set of conditions, nodes I and J are interchanged, a positive displacement of node J relative to node I tends to compress the spring.

COMBIN14 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

Structural Analysis:

- No damping capability; CV1 and CV2 are not allowed.
- Only stress stiffening and large deflections are allowed.
- KEYOPT(2) = 7 or 8 is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The BETD material property is not allowed.

ANSYS Professional

Thermal Analysis:

- KEYOPT(2) defaults to 8.
- KEYOPT(3) is not applicable.

DesignSpace

- KEYOPT(2) = 7 is not applicable.
- In the Mechanical application, KEYOPT(2) = 7 *and* KEYOPT(2) = 8 are not applicable .

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

MASS21

Structural Mass

MP ME ST PR PRN DS <> <> PP EME MFS
Product Restrictions

MASS21 Element Description

MASS21 is a point element having up to six degrees of freedom: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z axes. A different mass and rotary inertia may be assigned to each coordinate direction. See MASS21 in the *Mechanical APDL Theory Reference* for more details about this element.

Another element with a full mass matrix capability (off-diagonal terms) is MATRIX27.

Figure 21.1: MASS21 Geometry

MASS21 Input Data

The mass element is defined by a single node, concentrated mass components (Force*Time^2/Length) in the element coordinate directions, and rotary inertias (Force*Length*Time^2) about the element coordinate axes. The element coordinate system may be initially parallel to the global Cartesian coordinate system or to the nodal coordinate system (KEYOPT(2)). The element coordinate system rotates with the nodal coordinate rotations during a large deflection analysis. Options are available to exclude the rotary inertia effects and to reduce the element to a 2-D capability (KEYOPT(3)). If the element requires only one mass input, it is assumed to act in all appropriate coordinate directions. The coordinate system for this element is shown in Figure 21.1: MASS21 Geometry (p. 155).

KEYOPT(1) = 1 defines the mass in volume*density form, which allows plotting of the mass using **/ESHAPE**, as well as the use of a temperature-dependent density.

A summary of the element input is given in "MASS21 Input Summary" (p. 155). Element Input (p. 43) gives a general description of element input.

MASS21 Input Summary

Nodes

I

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ if KEYOPT (3) = 0
UX, UY, UZ if KEYOPT (3) = 2
UX, UY, ROTZ if KEYOPT (3) = 3

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UX, UY if KEYOPT (3) = 4
(degrees of freedom are in the nodal coordinate system)

Real Constants

MASSX, MASSY, MASSZ, IXX, IYY, IZZ, if KEYOPT (3) = 0
MASS, if KEYOPT (3) = 2
MASS, IZZ, if KEYOPT (3) = 3
MASS, if KEYOPT (3) = 4
(MASSX, MASSY, and MASSZ are concentrated mass components in the element coordinate directions. IXX, IYY, and IZZ are rotary inertias about the element coordinate axes. See also KEYOPT(2)).

Material Properties

MP command: DENS (if KEYOPT(1) = 1), ALPD

Surface Loads

None

Body Loads

None

Special Features

Birth and death
Large deflection
Linear perturbation

KEYOPT(1)

Real constant interpretation (mass/volume or rotary inertia/density):

0 --
Interpret real constants as masses and rotary inertias

1 --
Interpret real constants as volumes and rotary inertias/density (Density must be input as a material property)

KEYOPT(2)

Initial element coordinate system:

0 --
Element coordinate system is initially parallel to the global Cartesian coordinate system

1 --
Element coordinate system is initially parallel to the nodal coordinate system

KEYOPT(3)

Rotary inertia options:

0 --
3-D mass with rotary inertia

2 --
3-D mass without rotary inertia

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3 --
2-D mass with rotary inertia

4 --
2-D mass without rotary inertia

MASS21 Output Data

Nodal displacements are included in the overall displacement solution. There is no element solution output associated with the element unless element reaction forces and/or energies are requested.

MASS21 Assumptions and Restrictions

- 2-D elements are assumed to be in a global Cartesian Z = constant plane.
- The mass element has no effect on the static analysis solution unless acceleration or rotation is present, or inertial relief is selected [**IRLF**].
- The standard mass summary printout is based on the average of MASSX, MASSY, and MASSZ if (KEYOPT(3) = 0). Only the precise mass summary includes the directional masses.
- In an inertial relief analysis, the full matrix is used. All terms are used during the analysis.

MASS21 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The only special feature allowed is large deflection.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

PLANE25

Axisymmetric-Harmonic 4-Node Structural Solid

MP ME ST <> <> <> <> <> PP EME MFS
Product Restrictions

PLANE25 Element Description

Although this legacy element is available for use in your analysis, ANSYS, Inc. recommends using a current-technology element such as SOLID272 (KEYOPT(6) = 0).

PLANE25 is used for 2-D modeling of axisymmetric structures with nonaxisymmetric loading. Examples of such loading are bending, shear, or torsion. The element is defined by four nodes having three degrees of freedom per node: translations in the nodal x, y, and z direction. For unrotated nodal coordinates, these directions correspond to the radial, axial, and tangential directions, respectively.

See Harmonic Axisymmetric Elements with Nonaxisymmetric Loads (p. 84) for a description of various loading cases. See PLANE25 in the *Mechanical APDL Theory Reference* for more details about this element. See PLANE83 for a multi-node version of this element.

Figure 25.1: PLANE25 Geometry

PLANE25 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 25.1: PLANE25 Geometry (p. 159). The element input data includes four nodes, the number of harmonic waves (MODE on the **MODE** command), the symmetry condition (ISYM on the **MODE** command) and the orthotropic material properties. The MODE and ISYM parameters are discussed in detail in Harmonic Axisymmetric Elements with Nonaxisymmetric Loads (p. 84).

The material may be orthotropic, with directions corresponding to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems (p. 55). Harmonically varying nodal forces, if any, should be input on a full 360° basis.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Element loads are described in Nodal Loading (p. 49). Harmonically varying pressures may be input as surface loads on the element faces as shown by the circled numbers on Figure 25.1: PLANE25 Geometry (p. 159). Positive pressures act into the element.

Harmonically varying temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

KEYOPT(2) is used to include or suppress the extra displacement shapes.

KEYOPT(3) is used for temperature loading with MODE > 0 and temperature-dependent material properties. Material properties may only be evaluated at a constant (nonharmonically varying) temperature. If MODE = 0, the material properties are always evaluated at the average element temperature. If MODE > 0, TREF must be input as zero.

KEYOPT(4), (5), and (6) provide various element printout options (see Element Solution (p. 51)).

A summary of the element input is given in "PLANE25 Input Summary" (p. 160). Element Input (p. 43) gives a general description of element input.

PLANE25 Input Summary

Nodes
I, J, K, L

Degrees of Freedom
UX, UY, UZ

Real Constants
None

Material Properties

MP command: EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ), DENS, GXY, GYZ, GXZ, BETD, ALPD

Surface Loads

Pressures --
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L)

Mode Number
Number of harmonic waves around the circumference (**MODE**)

Loading Condition
Symmetry condition (**MODE**)

Special Features

Birth and death

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stress stiffening

KEYOPT(1)
Element coordinate system:

0 --
Element coordinate system is parallel to the global coordinate system

1 --
Element coordinate system is based on the element I-J side.

KEYOPT(2)
Extra displacement shapes:

0 --
Include extra displacement shapes

1 --
Suppress extra displacement shapes

KEYOPT(3)
If MODE is greater than zero, use temperatures for:

0 --
Use temperatures only for thermal bending (evaluate material properties at TREF)

1 --
Use temperatures only for material property evaluation (thermal strains are not computed)

KEYOPT(4)
Extra stress output:

0 --
Basic element solution

1 --
Repeat basic solution for all integration points

2 --
Nodal Stress Solution

KEYOPT(5)
Combined stress output:

0 --
No combined stress solution

1 --
Combined stress solution at centroid and nodes

KEYOPT(6)
Include extra surface output (surface solution valid only for isotropic materials):

0 --
Basic element solution

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
Surface solution for face I-J also

2 --
Surface solution for both faces I-J and K-L also

PLANE25 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 25.1: PLANE25 Element Output Definitions (p. 163).

Several items are illustrated in Figure 25.2: PLANE25 Stress Output (p. 162).

In the displacement printout, the UZ component is out-of-phase with the UX and UY components. For example, in the MODE = 1, ISYM = 1 loading case, UX and UY are the peak values at $\theta = 0°$ and UZ is the peak value at $\theta = 90°$. The same occurs for the reaction forces (FX, FY, etc.). The element stress directions are parallel to the element coordinate system. We recommend that you always use the *angle* field on the **SET** command when postprocessing the results. For more information about harmonic elements, see Harmonic Axisymmetric Elements with Nonaxisymmetric Loads (p. 84)

The sign convention on the surface shears is such that for a rectangular element that is lined up parallel to the axes with node J in the positive Y direction from node I, the shear stresses on surfaces I-J and K-L are analogous to the centroidal SYZ in both definition and sign. Stress components which are inherently zero for a load case are printed for clarity. Solution Output (p. 50) gives a general description of solution output. See the *Basic Analysis Guide* for ways to view results.

Figure 25.2: PLANE25 Stress Output

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 25.1: PLANE25 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
ISYM	Loading key: 1 = symmetric, -1 = antisymmetric	Y	-
MODE	Number of waves in loading	Y	-
VOLU:	Volume	Y	Y
PRES	Pressure P1 at nodes J,I; P2 at K,J; P3 at L,K; P4 at I,L	Y	Y
TEMP	Temperatures T(I), T(J), T(K), T(L)	Y	Y
PK ANG	Angle where component stresses have peak values: 0 and 90/MODE degrees. Blank if MODE = 0.	Y	Y
XC, YC	Location where results are reported	Y	3
S:X, Y, Z	Direct stresses (radial, axial, hoop) at *PK ANG* locations	Y	Y
S:XY, YZ, XZ	Shear stresses (radial-axial, axial-hoop, radial-hoop) at *PK ANG* locations	Y	Y
S:1, 2, 3	Principal stresses at both *PK ANG* locations as well as where extreme occurs (EXTR); if MODE = 0, only one location is given.	1	1
S:INT	Stress intensity at both *PK ANG* locations as well as where extreme occurs (EXTR); if MODE = 0, only one location is given.	1	1
S:EQV	Equivalent stress at both *PK ANG* locations as well as where extreme occurs (EXTR); if MODE = 0, only one location is given.	1	1
EPEL:X, Y, Z, XY	Elastic strain	Y	Y
EPEL:EQV	Equivalent elastic strain [4]	-	Y
EPTH:X, Y, Z, XY	Average thermal strains	1	1
EPTH:EQV	Equivalent thermal strain [4]	-	1
FACE	Face label	2	2
TEMP	Surface average temperature	2	2
EPEL(PAR, PER, Z, SH)	Surface strains (parallel, perpendicular, hoop, shear) at *PK ANG* locations and where extreme occurs (EXTR)	2	2
S(PAR, PER, Z, SH)	Surface stresses (parallel, perpendicular, hoop, shear) at *PK ANG* locations and where extreme occurs (EXTR)	2	2

1. These items are output only if KEYOPT(5) = 1.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. These items are printed only if KEYOPT(6) is greater than zero.

3. Available only at centroid as a ***GET** item.

4. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY).

Table 25.2: PLANE25 Item and Sequence Numbers (p. 164) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) of the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) for more information. The following notation is used in Table 25.2: PLANE25 Item and Sequence Numbers (p. 164):

Name
output quantity as defined in the Table 25.1: PLANE25 Element Output Definitions (p. 163)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J,K,L
sequence number for data at nodes I,J,K,L

Table 25.2: PLANE25 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	I	J	K	L
P1	SMISC	2	1	-	-
P2	SMISC	-	4	3	-
P3	SMISC	-	-	6	5
P4	SMISC	7	-	-	8
THETA = 0					
S1	NMISC	1	16	31	46
S2	NMISC	2	17	32	47
S3	NMISC	3	18	33	48
SINT	NMISC	4	19	34	49
SEQV	NMISC	5	20	35	50
THETA = 90/MODE					
S1	NMISC	6	21	36	51
S2	NMISC	7	22	37	52
S3	NMISC	8	23	38	53
SINT	NMISC	9	24	39	54
SEQV	NMISC	10	25	40	55
EXTR Values					
S1	NMISC	11	26	41	56

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	I	J	K	L
S2	NMISC	12	27	42	57
S3	NMISC	13	28	43	58
SINT	NMISC	14	29	44	59
SEQV	NMISC	15	30	45	60

Note

The NMISC items (1 thru 60) in the above table represent the combined stress solution, KEYOPT(5) = 1. If MODE = 0, their values are zero at THETA = 90/MODE and at EXTR.

See Surface Solution for the item and sequence numbers for surface output for the **ETABLE** command.

PLANE25 Assumptions and Restrictions

- The area of the element must be positive.
- The element must be defined in the global X-Y plane as shown in Figure 25.1: PLANE25 Geometry (p. 159) and the global X-axis must be the radial direction. Negative X coordinates should not be used.
- The element assumes a linear elastic material. Post-analysis superposition of results is valid only with other linear elastic solutions. The element should not be used with the large deflection option.
- A triangular element may be formed by defining duplicate K and L node numbers (see Degenerated Shape Elements (p. 39)). The extra shapes are automatically deleted for triangular elements so that a constant strain element results.
- Surface stress printout is valid only if the conditions described in Element Solution (p. 51) are met.
- You can use only axisymmetric (**MODE**,0) loads without significant torsional stresses to generate the stress state used for stress stiffened modal analyses using this element.
- This element does not support spectrum analysis.
- Modeling hint: If shear effects are important in a shell-like structure, you should use at least two elements through the thickness.

PLANE25 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

MATRIX27

Stiffness, Damping, or Mass Matrix

MP ME ST PR PRN <> <> <> PP EME MFS
Product Restrictions

MATRIX27 Element Description

MATRIX27 represents an arbitrary element whose geometry is undefined but whose elastic kinematic response can be specified by stiffness, damping, or mass coefficients in matrix form. The matrix is assumed to relate two nodes, each with six degrees of freedom per node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z axes. See MATRIX27 in the *Mechanical APDL Theory Reference* for more details about this element. Other similar, but less general, elements are the spring-damper element (COMBIN14), and the mass element (MASS21).

Figure 27.1: MATRIX27 Schematic

MATRIX27 Input Data

The node locations and the coordinate system for this element are shown in Figure 27.1: MATRIX27 Schematic (p. 167). The element is defined by two nodes and the matrix coefficients. The stiffness, damping, or mass matrix constants are input as real constants. The units of the stiffness constants are Force/Length or Force*Length/Radian and the damping constants, Force*Time/Length and Force*Length*Time/Radian. The mass constants should have units of Force*Time^2/Length or Force*Time^2*Length/Radian.

All matrices generated by this element are 12 by 12. The degrees of freedom are ordered as UX, UY, UZ, ROTX, ROTY, ROTZ for node I followed by the same for node J. If one node is not used, simply let all rows and columns relating to that node default to zero.

A structural matrix that combines the effects of many elements is normally positive or zero definite, as are the element matrices that contribute to it. There may be unusual circumstances where an element matrix is negative definite, and this is okay if there are other matrices connected to the same nodes that are positive definite, resulting in a final system of equations is still positive or zero definite. A simple example of such a circumstance is a beam element loaded with half of the buckling load. The stress stiffness matrix is negative definite, but the combined regular and stress stiffness matrix is positive definite.

When using MATRIX27 with symmetric element matrices (KEYOPT(2) = 0), positive or zero definite matrices should be input using KEYOPT(1) = 0, the default. If it is desired to input a negative definite element matrix, the user should set KEYOPT(1) = 1 so that the negative definite checking will be bypassed. However, when using an unsymmetric or skew-symmetric element matrix, (KEYOPT(2) = 2 or 3), there are no limits on the form of the matrix, as the unsymmetric solvers are designed to solve any system of equations, as long as it is not singular.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The matrix constants should be input according to the matrix diagrams shown in "MATRIX27 Output Data" (p. 169). For example, if a simple spring of stiffness K in the nodal x direction is desired, the input constants would be $C_1 = C_{58} = K$ and $C_7 = -K$ for KEYOPT(2) = 0 and KEYOPT(3) = 4.

A summary of the element input is given in "MATRIX27 Input Summary" (p. 168). Element Input (p. 43) gives a general description of element input.

MATRIX27 Input Summary

Nodes

I, J

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants

C1, C2, ... C78 - Define the upper triangular portion of the symmetric matrix (if KEYOPT(2) = 0)
C1, C2, ... C144 - Define all terms of the unsymmetric matrix (if KEYOPT(2) = 2)
C1, C2, ... C66 - Define upper triangular portion (less diagonal terms) if skew symmetric (KEYOPT(2) = 3)

Material Properties

MP command: BETD, ALPD

Surface Loads

None

Body Loads

None

Special Features

Birth and death
Linear perturbation

KEYOPT(1)

Matrix form (only works with KEYOPT(2) = 0):

0 --

Input positive or zero definite matrices only

1 --

Input positive, zero, or negative definite matrices

KEYOPT(2)

Matrix formulation:

0 --

Symmetric matrices (78 constants required)

2 --

Unsymmetric matrices(144 constants required)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3 --
Skew symmetric matrices (66 constants required)

KEYOPT(3)
Real constant input data:

2 --
Defines a 12 x 12 mass matrix

4 --
Defines a 12 x 12 stiffness matrix

5 --
Defines a 12 x 12 damping matrix

KEYOPT(4)
Element matrix output:

0 --
Do not print element matrix

1 --
Print element matrix at beginning of solution phase

MATRIX27 Output Data

The solution output associated with the element consists of node displacements included in the overall nodal solution. There is no element solution output associated with the element unless element reaction forces and/or energies are requested. KEYOPT(4) = 1 causes the element matrix to be printed (for the first substep of the first load step only). Solution Output (p. 50) gives a general description of solution output.

For KEYOPT(2) = 0, the symmetric matrix has the form:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$
\left[\begin{array}{cccccc|cccccc}
C_1 & C_2 & C_3 & . & . & . & . & . & . & . & . & C_{12} \\
 & C_{13} & C_{14} & . & . & . & . & . & . & . & . & C_{23} \\
 & & C_{24} & . & . & . & . & . & . & . & . & . \\
 & & & C_{34} & . & . & . & . & . & . & . & . \\
 & & & & C_{43} & . & . & . & . & . & . & . \\
 & & & & & C_{51} & . & . & . & . & . & . \\
\hline
 & & & & & & C_{58} & . & . & . & . & . \\
 & & & & & & & C_{64} & . & . & . & . \\
 & & & & & & & & C_{69} & . & . & . \\
 & & \text{Symmetric} & & & & & & & C_{73} & . & . \\
 & & & & & & & & & & C_{76} & . \\
 & & & & & & & & & & & C_{78}
\end{array}\right]
$$

For KEYOPT(2) = 2, the unsymmetric matrix has the form:

$$
\left[\begin{array}{cccccc|cccccc}
C_1 & C_2 & C_3 & . & . & . & . & . & . & . & . & C_{12} \\
C_{13} & C_{14} & C_{15} & . & . & . & . & . & . & . & . & C_{24} \\
C_{25} & C_{26} & C_{27} & . & . & . & . & . & . & . & . & C_{36} \\
. & . & . & . & . & . & . & . & . & . & . & . \\
. & . & . & . & . & . & . & . & . & . & . & . \\
. & . & . & . & . & . & . & . & . & . & . & . \\
\hline
. & . & . & . & . & . & . & . & . & . & . & . \\
. & . & . & . & . & . & . & . & . & . & . & . \\
. & . & . & . & . & . & . & . & . & . & . & . \\
. & . & . & . & . & . & . & . & . & . & . & . \\
. & . & . & . & . & . & . & . & . & . & . & . \\
C_{133} & C_{134} & C_{135} & . & . & . & . & . & . & . & . & C_{144}
\end{array}\right]
$$

For KEYOPT(2) = 3, the skew symmetric matrix has the form:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$
\left[\begin{array}{cccccc|cccccc}
0 & C_1 & C_2 & C_3 & . & . & . & . & . & . & . & C_{11} \\
 & 0 & C_{12} & C_{13} & . & . & . & . & . & . & . & C_{21} \\
 & & 0 & C_{22} & . & . & . & . & . & . & . & C_{30} \\
 & & & 0 & . & . & . & . & . & . & . & . \\
 & & & & 0 & . & . & . & . & . & . & . \\
 & & & & & 0 & . & . & . & . & . & . \\
\hline
 & & & & & & 0 & . & . & . & . & . \\
 & & & & & & & 0 & . & . & . & . \\
 & & & & & & & & 0 & . & . & . \\
 & & \text{Skew Symmetric} & & & & & & & 0 & C_{64} & C_{65} \\
 & & & & & & & & & & 0 & C_{66} \\
 & & & & & & & & & & & 0
\end{array}\right]
$$

MATRIX27 Assumptions and Restrictions

- Nodes may be coincident or noncoincident.
- Since element matrices should normally not be negative definite, a note is printed for those cases where this can be easily detected.
- With a lumped mass matrix [**LUMPM**,ON] all off-diagonal terms must be zero.
- The matrix terms are associated with the nodal degrees of freedom and are assumed to act in the nodal coordinate directions (see Elements That Operate in the Nodal Coordinate System (p. 57)).

MATRIX27 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- Damping and unsymmetric matrices are not allowed.
- Real constants C79 through C144, for unsymmetric matrices, are not applicable.
- The birth and death special feature is not allowed.
- KEYOPT(2) can only be set to 0 (default). KEYOPT(3) = 5 is not allowed.
- The BETD material property is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

SHELL28

Shear/Twist Panel

MP ME ST PR PRN <> <> <> PP EME MFS
Product Restrictions

SHELL28 Element Description

SHELL28 is used to carry shear load in a frame structure. The element has three degrees of freedom at each node: translations in the nodal x, y, and z directions, or rotations about the nodal x, y, and z axes. See SHELL28 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 28.1: SHELL28 Geometry

SHELL28 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 28.1: SHELL28 Geometry (p. 173). The element is defined by four nodes, a thickness, and material properties. The only material properties actually used are GXY and DENS. GXY may be entered directly or calculated from EX and either NUXY or PRXY. Also, EX must be input, whether or not GXY is entered. The element coordinate system orientation is as described in Coordinate Systems (p. 55). Real constant SULT is the ultimate shear stress used for the margin of safety calculation. ADMSUA is the added mass per unit area. KEYOPT(1) is used to select whether the element should be used as a shear panel or as a twist panel. Only the lumped mass matrix is available.

Element loads are described in Nodal Loading (p. 49). Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF. Temperatures are used only for material property evaluation.

A summary of the element input is given in "SHELL28 Input Summary" (p. 173). Element Input (p. 43) gives a general description of element input.

SHELL28 Input Summary

Nodes

I, J, K, L

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Degrees of Freedom

UX, UY, UZ if KEYOPT(1) = 0
ROTX, ROTY, ROTZ if KEYOPT(1) = 1

Real Constants

THCK - Panel thickness
SULT - Ultimate shear stress
ADMSUA - Added mass/unit area

Material Properties
MP command: EX, PRXY (or NUXY), GXY, DENS, BETD, ALPD

Surface Loads
None

Body Loads

Temperatures --
T(I), T(J), T(K), T(L)

Special Features
Stress stiffening

KEYOPT(1)
Element behavior:

0 --
Shear panel

1 --
Twist panel

SHELL28 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 28.1: SHELL28 Element Output Definitions (p. 175)

Several items are illustrated in Figure 28.2: SHELL28 Stress Output (p. 175). Solution Output (p. 50) gives a general description of solution output. See the *Basic Analysis Guide* for ways to view results.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28.2: SHELL28 Stress Output

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 28.1: SHELL28 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
SXY	Average of four corner shear stresses	Y	Y
XC, YC, ZC	Location where results are reported	Y	3
TEMP	Temperatures T(I), T(J), T(K), T(L)	Y	Y
SXY(I,J,K,L)	Shear stresses at corner nodes	Y	Y
SXY(MAX)	Maximum of four corner shear stresses	Y	Y
SMARGN	Margin of safety on shear	Y	Y
FDIK, FDJL	Forces along diagonals I-K and J-L	1	1
FLI, FJI	Forces at node I from node L and node J	1	1
FIJ, FKJ	Forces at node J from node I and node K	1	1
FJK, FLK	Forces at node K from node J and node L	1	1
FKL, FIL	Forces at node L from node K and node I	1	1
SFLIJ	Shear flow on edge I - J	1	1
SFLJK	Shear flow on edge J - K	1	1
SFLKL	Shear flow on edge K - L	1	1
SFLLI	Shear flow on edge L - I	1	1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
FZI	Z - Force at node I	1	1
FZJ	Z - Force at node J	1	1
FZK	Z - Force at node K	1	1
FZL	Z - Force at node L	1	1
MDIK, MDJL	Moments about diagonals I-K and J-L	2	2

1. The values are output only if KEYOPT(1) = 0

2. The values are output in place of FDIK and FDJL only if KEYOPT(1) = 1

3. Available only at centroid as a ***GET** item.

Table 28.2: SHELL28 Item and Sequence Numbers (p. 176) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) of this manual for more information. The following notation is used in Table 28.2: SHELL28 Item and Sequence Numbers (p. 176):

Name
output quantity as defined in the Table 28.1: SHELL28 Element Output Definitions (p. 175)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 28.2: SHELL28 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FDIK (MDIK)	SMISC	1
FDJL (MDJL)	SMISC	2
FLI	SMISC	3
FJI	SMISC	4
FIJ	SMISC	5
FKJ	SMISC	6
FJK	SMISC	7
FLK	SMISC	8
FKL	SMISC	9
FIL	SMISC	10
FZI	SMISC	11
FZJ	SMISC	12
FZK	SMISC	13
FZL	SMISC	14

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
SXY	SMISC	15
SXYI	SMISC	16
SXYJ	SMISC	17
SXYK	SMISC	18
SXYL	SMISC	19
SXYMAX	SMISC	20
SMARGN	SMISC	21
SFLIJ	SMISC	22
SFLJK	SMISC	23
SFLKL	SMISC	24
SFLLI	SMISC	25

SHELL28 Assumptions and Restrictions

- Zero area elements are not allowed. This condition occurs most often when the elements are numbered improperly.
- This element is most often used with a latticework of beam or spar elements. If this element is used alone it is almost always unstable, because it carries only shear (and not tension or compression) loading.
- This element is not recommended for general use. Its use should be restricted to applications which have historically used such an element. For all other applications, you should use other shell elements such as SHELL41, SHELL181, and SHELL281, which automatically combine tension, compression, bending, shear, and twisting effects.
- This element is based on the premise of having only shear, but no normal stress along the edges. Since this is possible only for rectangles, you can expect the accuracy of the element to degrade if nonrectangular shapes are used.

SHELL28 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The BETD material property is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

FLUID29

2-D Axisymmetric Harmonic Acoustic Fluid

MP ME <> <> <> <> <> <> PP EME MFS
Product Restrictions

FLUID29 Element Description

FLUID29 is used for modeling the fluid medium and the interface in fluid/structure interaction problems. Typical applications include sound wave propagation and submerged structure dynamics. The governing equation for acoustics, namely the 2-D wave equation, has been discretized taking into account the coupling of acoustic pressure and structural motion at the interface. The element has four corner nodes with three degrees of freedom per node: translations in the nodal x and y directions and pressure. The translations, however, are applicable only at nodes that are on the interface. Acceleration effects, such as in sloshing problems, may be included.

The element has the capability to include damping of sound absorbing material at the interface. The element can be used with other 2-D structural elements to perform unsymmetric or damped modal, full harmonic and full transient method analyses (see the description of the **TRNOPT** command). When there is no structural motion, the element is also applicable to static and modal analyses. See FLUID29 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 29.1: FLUID29 Geometry

FLUID29 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 29.1: FLUID29 Geometry (p. 179). The element is defined by four nodes, the number of harmonic waves (MODE on the **MODE** command), the symmetry condition (ISYM on the **MODE** command), a reference pressure, and the isotropic material properties. The MODE and ISYM parameters are discussed in detail in Harmonic Axisymmetric Elements with Nonaxisymmetric Loads (p. 84). The reference pressure (PREF) is used to calculate the element sound pressure level (defaults to 20×10^{-6} N/m^2). The speed of sound ($\sqrt{k / \rho_o}$) in the fluid is input by SONC where k is the bulk modulus of the fluid (Force/Area) and ρ_o is the mean fluid density (Mass/Volume) (input as DENS). The dissipative effect due to fluid viscosity is neglected, but absorption of sound at the interface is accounted for by generating a damping matrix using the surface area and boundary admittance at the interface. Experimentally measured values of the boundary admittance for the sound absorbing material may be input as material property MU. We recommend MU values from 0.0 to 1.0; however, values greater than 1.0 are allowed. MU = 0.0 represents no sound

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

absorption and MU = 1.0 represents full sound absorption. DENS, SONC and MU are evaluated at the average of the nodal temperatures.

Nodal flow rates, if any, may be specified using the **F** command where both the real and imaginary components may be applied. Nodal flow rates should be input per unit of depth for a plane analysis and on a 360° basis for an axisymmetric analysis.

Element loads are described in Nodal Loading (p. 49). Fluid-structure interfaces (FSI) can be flagged by surface loads at the element faces as shown by the circled numbers on Figure 29.1: FLUID29 Geometry (p. 179). Specifying the FSI label (without a value) (**SF**, **SFA**, **SFE**) couples the structural motion and fluid pressure at the interface. Deleting the FSI specification (**SFDELE**, **SFADELE**, **SFEDELE**) removes the flag. The flag specification should be on the fluid elements at the interface. The surface load label IMPD with a value of unity should be used to include damping that may be present at a structural boundary with a sound absorption lining. A zero value of IMPD removes the damping calculation. The displacement degrees of freedom (UX and UY) at the element nodes not on the interface should be set to zero to avoid zero-pivot warning messages.

Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

KEYOPT(2) is used to specify the absence of a structure at the interface and, therefore, the absence of coupling between the fluid and structure. Since the absence of coupling produces symmetric element matrices, a symmetric eigensolver (**MODOPT**) may be used within the modal analysis. However, for the coupled (unsymmetric) problem, a corresponding unsymmetric eigensolver (**MODOPT**) must be used.

Vertical acceleration (ACELY on the **ACEL** command) is needed for the gravity regardless of the value of MODE, even for a modal analysis.

A summary of the element input is given in "FLUID29 Input Summary" (p. 180). A general description of element input is given in Element Input (p. 43). For axisymmetric applications see Harmonic Axisymmetric Elements (p. 83).

FLUID29 Input Summary

Nodes

I, J, K, L

Degrees of Freedom

UX, UY, PRES if KEYOPT (2) = 0
PRES if KEYOPT (2) = 1

Real Constants

PREF (reference pressure)

Material Properties

MP command: DENS, SONC, MU

Surface Loads

Fluid-structure Interface Flag --

face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Impedance --
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Mode Number
Input mode number on **MODE** command

Loading Condition
Input for ISYM on **MODE** command

1 --
Symmetric loading

-1 --
Antisymmetric loading

Special Features
None

KEYOPT(2)
Structure at element interface:

0 --
Structure present at interface (unsymmetric element matrix)

1 --
No structure at interface (symmetric element matrix)

KEYOPT(3)
Element behavior:

0 --
Planar

1 --
Axisymmetric

2 --
Axiharmonic

KEYOPT(7)
Free surface effect:

0 --
Do not include sloshing effect

1 --
Include sloshing effect on face of elements located on Y = 0.0 plane (elements must not have positive Y coordinates)

FLUID29 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements and pressures included in the overall nodal solution

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Additional element output as shown in Table 29.1: FLUID29 Element Output Definitions (p. 182).

Solution Output (p. 50) gives a general description of solution output. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 29.1: FLUID29 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	2
TEMP	Temperatures T(I), T(J), T(K), T(L)	Y	Y
PRESSURE	Average pressure	Y	Y
PG(X, Y, SUM)	Pressure gradient components and vector sum	Y	Y
VL(X, Y, SUM)	Fluid velocity components and vector sum	1	1
SOUND PR.LEVEL	Sound pressure level (in decibels)	1	1

1. Output only if **ANTYPE**,HARMIC

2. Available only at centroid as a ***GET** item.

Table 29.2: FLUID29 Item and Sequence Numbers (p. 183) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) of this manual for more information. The following notation is used in Table 29.2: FLUID29 Item and Sequence Numbers (p. 183):

Name
output quantity as defined in the Table 29.1: FLUID29 Element Output Definitions (p. 182)

Item
predetermined Item label for **ETABLE** command

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

E

sequence number for single-valued or constant element data

Table 29.2: FLUID29 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
PGX	SMISC	1
PGY	SMISC	2
VLX	SMISC	3
VLY	SMISC	4
PRESSURE	NMISC	1
PGSUM	NMISC	2
VLSUM	NMISC	3
SOUND PR. LEVEL	NMISC	4

FLUID29 Assumptions and Restrictions

- The area of the element must be positive.
- The element must lie in a global X-Y plane as shown in Figure 29.1: FLUID29 Geometry (p. 179).
- All elements must have 4 nodes. A triangular element may be formed by defining duplicate K and L nodes (see Degenerated Shape Elements (p. 39)).
- The acoustic pressure in the fluid medium is determined by the wave equation with the following assumptions:
  - The fluid is compressible (density changes due to pressure variations).
  - Inviscid fluid (no dissipative effect due to viscosity).
  - There is no mean flow of the fluid.
  - The mean density and pressure are uniform throughout the fluid. Note that the acoustic pressure is the excess pressure from the mean pressure.
  - Analyses are limited to relatively small acoustic pressures so that the changes in density are small compared with the mean density.

The lumped mass matrix formulation (**LUMPM**,ON) is not allowed for this element.

FLUID29 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

FLUID30

3-D Acoustic Fluid

MP ME <> <> <> <> <> <> PP EME MFS
Product Restrictions

FLUID30 Element Description

Use FLUID30 for modeling the fluid medium and the interface in fluid-structure interaction problems.

For detailed information about this element, see Elements for Acoustic Analysis (p. 104). For further information, see FLUID30 in the *Mechanical APDL Theory Reference*.

Figure 30.1: FLUID30 Geometry

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

LINK31

Radiation Link

MP ME <> PR PRN <> <> <> PP EME MFS
Product Restrictions

LINK31 Element Description

LINK31 is a uniaxial element which models the radiation heat flow rate between two points in space. The link has a single degree of freedom, temperature, at each node. The radiation element is applicable to a 2-D (plane or axisymmetric) or 3-D, steady-state or transient thermal analysis.

An empirical relationship allowing the form factor and area to multiply the temperatures independently is also available. The emissivity may be temperature dependent. If the model containing the radiation element is also to be analyzed structurally, the radiation element should be replaced by an equivalent (or null) structural element. See LINK31 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 31.1: LINK31 Geometry

LINK31 Input Data

The geometry, node locations, and the coordinate system for this radiation element are shown in Figure 31.1: LINK31 Geometry (p. 187). The element is defined by two nodes, a radiating surface area, a geometric form factor, the emissivity, and the Stefan-Boltzmann constant (SBC). For axisymmetric problems, the radiation area should be input on a full 360° basis.

The emissivity may be constant or temperature (absolute) dependent. If it is constant, the value is input as a real constant. If it is temperature dependent, the values are input for the material property EMIS and the real constant value is used only to identify the material property number. In this case the MAT value associated with element is not used. EMIS defaults to 1.0.

The standard radiation function is defined as follows:

$$q = \sigma \varepsilon FA(T(I)^4 - T(J)^4)$$

where:

σ = Stefan-Boltzmann Constant (SBC)
(defaults to 0.119×10^{-10} (BTU/Hr*in^2* °R^4)
ε = emissivity
F = geometric form factor
A = area (Length)^2
q = heat flow rate (Heat/Time)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The nonlinear temperature equation is solved by a Newton-Raphson iterative solution based on the form:

$$[(T(I)^2 + T(J)^2)(T(I) + T(J))]_p(T(I) - T(J))$$

where the $[\ ]_p$ term is evaluated at the temperature of the previous substep. The initial temperature should be near the anticipated solution and should not be zero (i.e., both TUNIF and TOFFST should not be zero).

An empirical radiation function of the following form may also be selected with KEYOPT(3):

$$q = \sigma \varepsilon(FT(I)^4 - AT(J)^4)$$

where F and A are arbitrary input constants.

A summary of the element input is given in "LINK31 Input Summary" (p. 188). A general description of element input is given in Element Input (p. 43).

LINK31 Input Summary

Nodes
I, J

Degrees of Freedom
TEMP

Real Constants

AREA - Radiating surface area
FORM FACTOR - Geometric form factor
EMISSIVITY - Emissivity (If EMISSIVITY = -*n*, use material *n* for emissivity vs. temperature definition)
SBC - Stefan-Boltzmann constant

Material Properties
MP command: EMIS (required only if EMISSIVITY = -N)

Surface Loads
None

Body Loads
None

Special Features

Birth and death
Nonlinearity

KEYOPT(3)
Radiation equation:

0 --
Use standard radiation equation

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
Use empirical radiation equation

The Stefan-Boltzmann constant (SBC) defaults to 0.1190E-10 with units of Btu, hr, in, °R (or °F if **TOFFST** is used)

LINK31 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output as shown in Table 31.1: LINK31 Element Output Definitions (p. 189)

The heat flow rate is positive from node I to node J. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 31.1: LINK31 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
MAT	Material number	Y	Y
AREA	AREA	Y	Y
XC, YC, ZC	Location where results are reported	Y	1
EMIS(I, J)	Emissivity - I, J	Y	Y
TEMP(I, J)	Temperatures - I, J	Y	Y
HEAT RATE	Heat flow rate from node I to node J	Y	Y

1. Available only at centroid as a ***GET** item.

Table 31.2: LINK31 Item and Sequence Numbers (p. 190) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) of this manual for more information. The following notation is used in Table 31.2: LINK31 Item and Sequence Numbers (p. 190):

Name
output quantity as defined in the Table 31.1: LINK31 Element Output Definitions (p. 189)

Item
predetermined Item label for **ETABLE** command

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

E

sequence number for single-valued or constant element data

Table 31.2: LINK31 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
HEAT RATE	SMISC	1
TEMPI	SMISC	2
TEMPJ	SMISC	3
EMISI	NMISC	1
EMISJ	NMISC	2
AREA	NMISC	3
FORM FACTOR	NMISC	4

LINK31 Assumptions and Restrictions

- If the default Stefan-Boltzmann constant is used, the units associated with this element are Btu, inches, hours and °R (or °F + TOFFST). Other data input for this analysis must be consistent with this set of units or an appropriate conversion factor should be included in the radiation element's real constants.
- Nodes may or may not be coincident.
- An iterative solution is required with this element.

LINK31 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LINK33

3-D Conduction Bar

MP ME <> PR PRN DS <> <> PP EME MFS
Product Restrictions

LINK33 Element Description

LINK33 is a uniaxial element with the ability to conduct heat between its nodes. The element has a single degree of freedom, temperature, at each node point. The conducting bar is applicable to a steady-state or transient thermal analysis.

If the model containing the conducting bar element is also to be analyzed structurally, the bar element should be replaced by an equivalent structural element. See LINK33 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 33.1: LINK33 Geometry

LINK33 Input Data

The geometry, node locations, and the coordinate system for this conducting bar are shown in Figure 33.1: LINK33 Geometry (p. 191). The element is defined by two nodes, a cross-sectional area, and the material properties. Specific heat and density are ignored for steady-state solutions. The thermal conductivity is in the element longitudinal direction.

Element loads are described in Nodal Loading (p. 49). Heat generation rates may be input as element body loads at the nodes. The node J heat generation rate HG(J) defaults to the node I heat generation rate HG(I).

A summary of the element input is given in "LINK33 Input Summary" (p. 191). A general description of element input is given in Element Input (p. 43).

LINK33 Input Summary

Nodes

I, J

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Degrees of Freedom
TEMP

Real Constants
AREA - Cross-sectional area

Material Properties
MP command: KXX, DENS, C, ENTH

Surface Loads
None

Body Loads

Heat Generation --
HG(I), HG(J)

Special Features
Birth and death

KEYOPTS
None

LINK33 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output as shown in Table 33.1: LINK33 Element Output Definitions (p. 192)

The heat flow rate is in units of Heat/Time and is positive from node I to node J. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 33.1: LINK33 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
LENGTH	Length	Y	Y
AREA	Input area	Y	Y
TEMP(I, J)	Temperatures - I, J	Y	Y
HEAT RATE	Heat flow rate from node I to node J	Y	Y
THERMAL FLUX	Thermal flux (heat flow rate/cross-sectional area)	Y	Y

1. Available only at centroid as a ***GET** item.

Table 33.2: LINK33 Item and Sequence Numbers (p. 193) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) of this manual for more information. The following notation is used in Table 33.2: LINK33 Item and Sequence Numbers (p. 193):

Name
output quantity as defined in the Table 33.1: LINK33 Element Output Definitions (p. 192)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 33.2: LINK33 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
HEAT RATE	SMISC	1
TEMPI	SMISC	2
TEMPJ	SMISC	3
THERMAL FLUX	SMISC	4
LENGTH	NMISC	1
AREA	NMISC	2

LINK33 Assumptions and Restrictions

- Heat is assumed to flow only in the longitudinal element direction.
- The element must not have a zero length, so nodes I and J must not be coincident.
- A free end of the element (i.e., not adjacent to another element and not subjected to a boundary constraint) is assumed to be adiabatic.

LINK33 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS Professional

- The birth and death special feature is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LINK34

Convection Link

MP ME <> PR PRN <> <> <> PP EME MFS
Product Restrictions

LINK34 Element Description

LINK34 is a uniaxial element with the ability to convect heat between its nodes. The element has a single degree of freedom, temperature, at each node point. The convection element is applicable to a 2-D (plane or axisymmetric) or 3-D, steady-state or transient thermal analysis.

If the model containing the convection element is also to be analyzed structurally, the convection element should be replaced by an equivalent (or null) structural element. The element may have a nonlinear film coefficient which may also be a function of temperature or time. See LINK34 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 34.1: LINK34 Geometry

LINK34 Input Data

The geometry and node locations for this convection element are shown in Figure 34.1: LINK34 Geometry (p. 195). The element is defined by two nodes, a convection surface area, two empirical terms, and a film coefficient. In an axisymmetric analysis the convection area must be expressed on a full 360° basis. The empirical terms n (input as EN) and CC determine the form of the convection equation in conjunction with KEYOPT(3).

The convection function is defined as follows:

$$q = h_f*A*E*(T(I) - T(J))$$

where:

q = heat flow rate (Heat/Time)

h_f = film coefficient (Heat/Length^2*Time*Deg)

A = area (Length^2)

T = temperature (this substep) (Deg)

E = empirical convection term = $F*IT_p(I) - T_p(J)I^n + CC/h_f$

T_p = temperature (previous substep) (Deg)

n = empirical coefficient (EN)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CC = input constant

Note

E = F if n and CC = 0.0. F = 1.0 unless KEYOPT(3) = 2. If KEYOPT(3) = 3, E equals the larger of $IT_p(I) - T_p(J)I^n$ or CC/h_f.

A special option obtained with KEYOPT(3) = 2 allows an alternate input for h_f and an input scale factor (F). This option uses the *VAL1* field of the **SFE** command with *KVAL* = 0 for the h_f value and *KVAL* = 2 for the F value. If the h_f value is zero (or blank), the HF material property is used for h_f. If the F value is zero (or blank) or negative, a value of 1.0 is assumed for F. Note, the F value input in this field will ramp within a load step if KBC = 0. An **SFE** command must be included (even if the values are left blank) for all LINK34 elements having KEYOPT(3) = 2.

Heat generation rates may be input as element body loads at the nodes. The node J heat generation rate HG(J) defaults to the node I heat generation rate HG(I).

A summary of the element input is given in "LINK34 Input Summary" (p. 196). A general description of element input is given in Element Input (p. 43).

LINK34 Input Summary

Nodes
I, J

Degrees of Freedom
TEMP

Real Constants

AREA - Convection surface area
EN - Empirical coefficient
CC - Input constant

Material Properties
MP command: HF

Surface Loads

Convections --
1 - Alternate input of HF and F if KEYOPT(3) = 2 (see text above)

Body Loads

Heat Generation --
HG(I), HG(J)

Special Features

Birth and death
Nonlinearity if real constant EN is not equal to zero or if KEYOPT(3) = 3

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(2)
Evaluation of film coefficient:

0 --
Use average of T(I) and T(J) to evaluate HF

1 --
Use greater of T(I) or T(J) to evaluate HF

2 --
Use lesser of T(I) or T(J) to evaluate HF

3 --
Use differential |T(I) - T(J)| to evaluate HF

KEYOPT(3)
Film coefficient and scale factor:

0 --
Standard element input and empirical term

2 --
Use alternate input for HF and F (input with **SFE** command)

3 --
Use discontinuous empirical term

LINK34 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output as shown in Table 34.1: LINK34 Element Output Definitions (p. 197)

The heat flow rate is in units of Heat/Time and is positive from node I to node J. In an axisymmetric analysis, the heat flow is on a full 360° basis. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 34.1: LINK34 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
XC, YC	Location where results are reported	Y	1
H	Film coefficient (includes empirical term)	Y	Y
AREA	Input area	Y	Y
TEMP	Temperature at node I and node J	Y	Y
HEAT RATE	Heat flow rate from node I to node J	Y	Y

1. Available only at centroid as a ***GET** item.

Table 34.2: LINK34 Item and Sequence Numbers (p. 198) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) of this manual for more information. The following notation is used in Table 34.2: LINK34 Item and Sequence Numbers (p. 198):

Name
output quantity as defined in the Table 34.1: LINK34 Element Output Definitions (p. 197)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J
sequence number for data at nodes I and J

Table 34.2: LINK34 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
HEAT RATE	SMISC	1	-	-
TEMP	SMISC	-	2	3
H	NMISC	1	-	-
AREA	NMISC	2	-	-

LINK34 Assumptions and Restrictions

- If $T_p(I) = T_p(J)$ and n are nonzero, the first term of E is defined to be zero.
- Since all unspecified nodal temperatures are initially set to the uniform temperature, a nonzero value of n may result in no heat flowing through the element in the first substep of a thermal solution.
- Nodes may or may not be coincident.
- The element is nonlinear if n is nonzero or KEYOPT(3) = 3. However, the solver always assumes the element is nonlinear and, therefore, always performs an iterative solution. (Only 2 iterations are performed if the element is linear.)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LINK34 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The birth and death special feature is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

PLANE35

2-D 6-Node Triangular Thermal Solid

MP ME <> PR PRN <> <> <> PP EME MFS
Product Restrictions

PLANE35 Element Description

PLANE35 is a 6-node triangular element compatible with the 8-node PLANE77 element. The triangular shape makes it well suited to model irregular meshes (such as produced from various CAD/CAM systems). The element has one degree of freedom, temperature, at each node.

The 6-node thermal element is applicable to a 2-D, steady-state or transient thermal analysis. If the model containing this element is also to be analyzed structurally, the element should be replaced by an equivalent structural element (such as PLANE183). The element may be used as a plane element or as an axisymmetric ring element. See PLANE35 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 35.1: PLANE35 Geometry

PLANE35 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 35.1: PLANE35 Geometry (p. 201).

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems (p. 55). Specific heat and density are ignored for steady-state solutions. Properties not input default as described in Linear Material Properties in the *Material Reference*.

Element loads are described in Nodal Loading (p. 49). Convection or heat flux (but not both) and radiation may be input as surface loads at the element faces as shown by the circled numbers on Figure 35.1: PLANE35 Geometry (p. 201). Heat generation rates may be input as element body loads at the nodes. If the node I heat generation rate HG(I) is input, and all others are unspecified, they default to HG(I). If all corner node heat generation rates are specified, each midside node heat generation rate defaults to the average heat generation rate of its adjacent corner nodes. An edge with a removed midside node implies that the temperature varies linearly, rather than parabolically, along that edge.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.

A summary of the element input is given in "PLANE35 Input Summary" (p. 202). A general description of element input is given in Element Input (p. 43). For axisymmetric applications see Harmonic Axisymmetric Elements (p. 83).

PLANE35 Input Summary

Nodes
I, J, K, L, M, N

Degrees of Freedom
TEMP

Real Constants
None

Material Properties
MP command: KXX, KYY, DENS, C, ENTH

Surface Loads

Convection or Heat Flux (but not both) and Radiation (using Lab = RDSF) --
face 1 (J-I), face 2 (K-J), face 3 (I-K)

Body Loads

Heat Generations --
HG(I), HG(J), HG(K), HG(L), HG(M), HG(N)

Special Features
Birth and death

KEYOPT(1)
Specific heat matrix:

0 --
Consistent specific heat matrix

1 --
Diagonalized specific heat matrix

KEYOPT(3)
Element behavior:

0 --
Plane

1 --
Axisymmetric

PLANE35 Output Data

The solution output associated with the element is in two forms:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Nodal temperatures included in the overall nodal solution
- Additional element output as shown in Table 35.1: PLANE35 Element Output Definitions (p. 203)

For an axisymmetric analysis the face area and the heat flow rate are on a full 360° basis. Convection heat flux is positive out of the element; applied heat flux is positive into the element. The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 35.1: PLANE35 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	2
HGEN	Heat generations HG(I), HG(J), HG(K), HG(L), HG(M), HG(N)	Y	-
TG:X, Y, SUM	Thermal gradient components and vector sum at centroid	Y	Y
TF:X, Y, SUM	Thermal flux (heat flow rate/cross-sectional area) components and vector sum at centroid	Y	Y
FACE	Face label	1	1
AREA	Face area	1	1
NODES	Face nodes	1	1
HFILM	Film coefficient	1	1
TAVG	Average face temperature	1	1
TBULK	Fluid bulk temperature	1	-
HEAT RATE	Heat flow rate across face by convection	1	1
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	1	-
HFLUX	Heat flux at each node of face	1	-
HFAVG	Average film coefficient of the face	-	1
TBAVG	Average face bulk temperature	-	1
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. If a surface load has been input

2. Available only at centroid as a ***GET** item.

Table 35.2: PLANE35 Item and Sequence Numbers (p. 204) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) of this manual for more information. The following notation is used in Table 35.2: PLANE35 Item and Sequence Numbers (p. 204):

Name
output quantity as defined in the Table 35.1: PLANE35 Element Output Definitions (p. 203)

Item
predetermined Item label for **ETABLE** command

FC*N*
sequence number for solution items for element Face *N*

Table 35.2: PLANE35 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	FC1	FC2	FC3
AREA	NMISC	1	7	13
HFAVG	NMISC	2	8	14
TAVG	NMISC	3	9	15
TBAVG	NMISC	4	10	16
HEAT RATE	NMISC	5	11	17
HFLXAVG	NMISC	6	12	18

PLANE35 Assumptions and Restrictions

- The area of the element must be positive.
- The element must lie in an X-Y plane as shown in Figure 35.1: PLANE35 Geometry (p. 201) and the Y-axis must be the axis of symmetry for axisymmetric analyses.
- An axisymmetric structure should be modeled in the +X quadrants.
- A face with a removed midside node implies that the temperature varies linearly, rather than parabolically, along that face.
- The specific heat and enthalpy are evaluated at each integration point to allow for abrupt changes (such as melting) within a coarse grid of elements.
- A free surface of the element (i.e., not adjacent to another element and not subjected to a boundary constraint) is assumed to be adiabatic.
- Thermal transients having a fine integration time step and a severe thermal gradient at the surface will require a fine mesh at the surface.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PLANE35 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The birth and death special feature is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

SOURC36

Current Source

MP <> <> <> <> <> EM <> PP EME <>
Product Restrictions

SOURC36 Element Description

SOURC36 is a primitive (consisting of predefined geometries) used to supply current source data to magnetic field problems. The element represents a distribution of current in a model employing a scalar potential formulation (degree of freedom MAG). The currents are used to calculate a source magnetic field intensity (H_s) using a numerical integration technique involving the Biot-Savart law. The H_s term is used in the formulation as a magnetic load on the model. See SOURC36 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 36.1: SOURC36 Geometry

SOURC36 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 36.1: SOURC36 Geometry (p. 207). The element input data includes three nodes and the following real constants (see "SOURC36 Input Summary" (p. 208)):

TYPE

Source type - use 1 for Coil, 2 for Bar, 3 for Arc.

CUR

Total current flowing through source (i.e., number of turns times current per turn).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DY
Characteristic y dimension for source type.

DZ
Characteristic z dimension for source type.

EPS
Convergence criterion for source field (H_s) calculations for arc and coils. Defaults to 0.001. EPS represents the relative maximum difference in the field H_s calculated at any node during the iterative calculation of the source field. EPS does not apply for bar sources.

Characteristic dimensions described above are in the element coordinate system. In the case of circular sources (coils, arcs) the radius is determined from the first and third nodes (I, K). For bar sources, the length is determined from the first two nodes (I, J).

As a modeling aid, a magnetic command macro, **RACE**, is available within the ANSYS command set. This macro enables the user to build a racetrack conductor from SOURC36 primitives. The macro is discussed in further detail in the *Command Reference* and in the *Low-Frequency Electromagnetic Analysis Guide*.

A summary of the element input is given in "SOURC36 Input Summary" (p. 208). A general description of element input is given in Element Input (p. 43).

SOURC36 Input Summary

Nodes
I, J, K (nodes I, J and K define the characteristic length, current flow direction, and orient the source)

Degrees of Freedom
None

Real Constants

TYPE, CUR, DY, DZ, (Blank), (Blank),
(Blank), (Blank), EPS
See Table 36.1: SOURC36 Real Constants (p. 209) for a description of the real constants.

Material Properties
None

Surface Loads
None

Body Loads
None

Special Features
None

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPTS

None

Table 36.1: SOURC36 Real Constants

No.	Name	Description
1	TYPE	Source type
2	CUR	Total current through source
3	DY	Characteristic Y dimension
4	DZ	Characteristic Z dimension
5 ... 8	(Blank)	
9	EPS	Convergence criteria for H_s calculations

As a modeling aid, a magnetic command macro, **RACE**, is available within the ANSYS command set. This macro enables the user to build a racetrack conductor from SOURC36 primitives. The macro is discussed in further detail in the *Command Reference* and in the *Low-Frequency Electromagnetic Analysis Guide*.

SOURC36 Output Data

The source element has no output of its own since it is used only to supply current source data to magnetic field problems.

SOURC36 Assumptions and Restrictions

- The source element must have characteristic DY or DZ values that are greater than zero.
- The third node must not be colinear with the first two nodes.
- The nodes for this element need not be attached to any other elements.
- For the coil and the arc (types 1 and 3), the K-I line determines the radius (and the x axis) and the J node orients the x-y plane.
- For the arc (type 3) the subtended angle must be less than 180°. When you specify an arc using three points, ANSYS will always use the angle that is less than 180°.
- All source element nodes should be located a least 1E-6 units apart.
- Source element cannot have a zero inside radius (Radius $\neq$ DY/2 for types 1 and 3).
- The EPS convergence criterion is a measure of the relative difference in the calculated H_s field used during an iterative numerical integration procedure for coil and arc source primitives. The default value (.001) provides for good accuracy in regions outside of the source primitive location. For highly accurate calculations within the source primitive domain, the criteria may have to be tightened (i.e., a factor of 20 increase would be represented by EPS = .00005).
- Tightening the convergence criteria will significantly increase the solution run time.
- Users concerned with accurate calculations within the coil and arc source primitive domain should experiment with the criteria until satisfied with the degree of accuracy obtained.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- All currents for a magnetostatic model employing the scalar potential formulation must be specified. Whereas symmetry conditions on the finite element model may be employed, no symmetry may be employed on the current source elements.

SOURC36 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

COMBIN37

Control

MP ME ST <> <> <> <> <> PP EME MFS
Product Restrictions

COMBIN37 Element Description

COMBIN37 is a unidirectional element with the capability of turning on and off during an analysis. The element has one degree of freedom at each node, either a translation in a nodal coordinate direction, rotation about a nodal coordinate axis, pressure, or temperature. Similar unidirectional elements (without remote control capability) are COMBIN14, COMBIN39, and COMBIN40. The element has many applications, such as controlling heat flow as a function of temperature (thermostat), controlling damping as a function of velocity (mechanical snubber), controlling flow resistance as a function of pressure (relief valve), controlling friction as a function of displacement (friction clutch), etc. See COMBIN37 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 37.1: COMBIN37 Geometry

COMBIN37 Input Data

The functioning of this element is shown in Figure 37.1: COMBIN37 Geometry (p. 211). The element is defined by two pairs of nodes, these being active nodes (I, J) and optional control nodes (K, L). Generally in the cases using UX, UY, or UZ as the active degrees of freedom, it is recommended to have the active nodes be coincident as this eliminates the possibility of moment disequilibrium. However, for visualization purposes, it may be useful to give node J a slightly greater coordinate value than node I. The element is defined such that a positive displacement of node J relative to node I will stretch the spring. Thus, if nodes I and J are interchanged, the same nodal motions will compress the spring.

Certain parameters associated with the control nodes are used to determine whether the control element is part of the structure (on) or not (off) and, thus, can be used to disconnect regions of the model during time dependent or iterative analyses. Other input values are stiffness (STIF), damping coefficient (DAMP), concentrated nodal masses (MASI, MASJ), on/off control values (ONVAL, OFFVAL), element load (AFORCE: positive pulls node I in the positive nodal coordinate direction, and pulls node J in the negative nodal coordinate direction), initial on/off element status (START: -1 if explicitly off, 0 if determined from starting value of control parameter, 1 if explicitly on), several nonlinear constants (C1, C2, C3, C4), and a limiting sliding force (FSLIDE).

The FSLIDE value represents the absolute value of the spring force that must be exceeded before sliding occurs. If FSLIDE is 0.0, the sliding capability of the element is removed, that is, a rigid connection is

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

assumed. For structural analyses, units are force/length or moment/rotation for stiffness, force*time/length or moment*time/rotation for damping, force*time^2/length or moment*time^2/rotation for mass, and force or moment for element load. For thermal analyses with temperature degrees of freedom, stiffness represents conductance and has units of heat/time*degrees, mass represents thermal capacitance with units of heat/degrees, and element load represents heat flow with units of heat/time. Also, in analyses with pressure degrees of freedom, stiffness represents flow conductance with units of length^2/time. Stiffness, damping, mass, and element load should be defined on a full 360° basis for axisymmetric analyses.

The active nodes (I, J) have only one degree of freedom each, selected with the KEYOPT(3) option. The control nodes (K, L) can have the same, or a different, degree of freedom as specified with KEYOPT(2). The KEYOPT(1) option assigns to the parameters of the control nodes either the value of the degree of freedom, the first or second derivative of the value, the integral of the value, or time, for example:

$$CPAR = UX_K - UX_L$$
$$CPAR = d(T_K - T_L)/dt$$
$$CPAR = d^2(ROTZ_K - ROTZ_L)/dt^2$$
$$CPAR = \int_o^t (UY_K - UY_L)dt$$
$$CPAR = t$$

Control nodes need not be connected to any other element. If node L is not defined, the control parameter is based only upon node K. If time is the control parameter (KEYOPT(1)), control nodes K and L need not be defined.

When the element is active and used in structural analyses, the element acts like any other spring/damper/mass element (such as COMBIN14, MASS21, and COMBIN40). In addition, the element can exhibit nonlinear behavior according to the function: RVMOD = RVAL + C1$|$CPAR$|^{C2}$ + C3$|$CPAR$|^{C4}$, where RVMOD is the modified value of an input real constant value RVAL (identified by KEYOPT(6)), C1 through C4 are other real constants, and CPAR is the control parameter (see KEYOPT(1)). RVMOD may also be defined by user subroutine USERRC and is accessed by KEYOPT(9) = 1. Note, FSLIDE modified to a negative value is set to zero. In a field analysis, the temperature or pressure degree of freedom acts in a manner analogous to the displacement.

As illustrated in Figure 37.2: COMBIN37 Behavior as a Function of Control Parameter (p. 216), the KEYOPT(4) and KEYOPT(5) options, when used in combination with ONVAL and OFFVAL, set the control behavior of the element. The element is either on or off depending on the position of the control parameter with respect to the values of ONVAL and OFFVAL. Also, note that when KEYOPT(4) = 0 and the control parameter (CPAR) is within the ONVAL/OFFVAL interval, the element's status depends on the direction of the CPAR (i.e., on going from on to off, and vice-versa). If ONVAL = OFFVAL = 0.0 (or blank), the on/off capability is ignored and the element is always active.

A summary of the element input is given in "COMBIN37 Input Summary" (p. 212). A general description of element input is given in Element Input (p. 43).

COMBIN37 Input Summary

Nodes

I, J, K, L (or I, J, K or I, J)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ, PRESS, or TEMP (depending on KEYOPT(2) and KEYOPT (3) below)

Real Constants

STIF, DAMP, MASJ, ONVAL, OFFVAL, AFORCE,
MASI, START, C1, C2, C3, C4,
FSLIDE
See Table 37.1: COMBIN37 Real Constants (p. 215) for a description of the real constants

Material Properties

MP command: ALPD, BETD

Surface Loads

None

Body Loads

None

Special Features

Adaptive descent
Nonlinearity

KEYOPT(1)

Control parameter:

0, 1 --
Control on value (UK-UL) (or UK if L not defined)

2 --
Control on first derivative of value with respect to time

3 --
Control on second derivative of value with respect to time

4 --
Control on integral of value with respect to time (zero initial condition assumed)

5 --
Control on time value (KEYOPT(2) and nodes K and L ignored)

KEYOPT(2)

Degree of freedom for control nodes (K and L):

N --
Use degree of freedom N as listed for KEYOPT(3) (defaults to KEYOPT(3))

KEYOPT(3)

Degree of freedom for active nodes (I and J):

0, 1 --
UX (Displacement along nodal X axes)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2 --
UY (along nodal Y)

3 --
UZ (along nodal Z)

4 --
ROTX (rotation about nodal X axes)

5 --
ROTY (about nodal Y)

6 --
ROTZ (about nodal Z)

7 --
PRESS

8 --
TEMP

KEYOPT(4)
ON-OFF range behavior (see Figure 37.2: COMBIN37 Behavior as a Function of Control Parameter (p. 216)):

0 --
Overlapping ranges

1 --
Unique ranges

KEYOPT(5)
ON-OFF position behavior (see Figure 37.2: COMBIN37 Behavior as a Function of Control Parameter (p. 216)):

0 --
OFF-either-ON (or OFF-ON-OFF if unique)

1 --
ON-either-OFF (or ON-OFF-ON if unique)

KEYOPT(6)
Real constants used for RVMOD function (used if C1 or C3 is not equal to zero; see "COMBIN37 Input Data" (p. 211)):

0, 1 --
Use STIF for nonlinear function. (Both STIF and FSLIDE cannot be zero).

2 --
Use DAMP

3 --
Use MASJ

4 --
Use ONVAL

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5 --
Use OFFVAL

6 --
Use AFORCE

7 --
Use MASI

8 --
Use FSLIDE

KEYOPT(9)
Method to define nonlinear behavior:

0 --
Use RVMOD expression for real constant modifications

1 --
Real constants modified by user subroutine USERRC

Note

See the *Guide to User-Programmable Features* information about user written subroutines

---

Table 37.1: COMBIN37 Real Constants

No.	Name	Description
1	STIF	Spring stiffness
2	DAMP	Damping coefficient
3	MASJ	Nodal mass at node J
4	ONVAL	"ON" control value
5	OFFVAL	"OFF" control value
6	AFORCE	Element load
7	MASI	Nodal mass at node I
8	START	Initial on/off element status
9	C1	First scalar in RVMOD equation
10	C2	First exponent in RVMOD equation
11	C3	Second scalar in RVMOD equation
12	C4	Second exponent in RVMOD equation
13	FSLIDE	Limiting sliding force

COMBIN37 Output Data

The solution output associated with the element is in two forms:

- Nodal degree of freedom results included in the overall nodal solution
- Additional element output as shown in Table 37.2: COMBIN37 Element Output Definitions (p. 217).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The active nodal displacements and forces correspond to the degree of freedom selected with the KEYOPT(3) option. For axisymmetric analysis, the element forces are expressed on a full 360° basis.

The element value STRETCH is the relative deflection at the end of the substep less the amount of sliding (e.g., UJ-UI-SLIDE). STATUS and OLDST indicate if the element is on or off at the end of the current and previous substeps, respectively. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

Figure 37.2: COMBIN37 Behavior as a Function of Control Parameter

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 37.2: COMBIN37 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
ACTIVE NODES	Nodes - I, J	Y	Y
CONTROL NODES	Nodes - K, L	Y	Y
XC, YC, ZC	Location where results are reported	Y	5
CONTROL PARAM	CPAR value (see KEYOPT(1)) of the control nodes	Y	Y
STAT	Element status	1	1
OLDST	STAT value of the previous time step	1	1
UI	Displacement of node I	2	2
UJ	Displacement of node J	2	2
UK	Displacement of node K	2	2
UL	Displacement of node L	2	2
STRETCH	Relative displacement	2	2
SFORCE	Spring force in element	2	2
AFORCE	Applied force in the element	2	2
SLSTAT	Sliding status	3	3
OLDSLS	Sliding status value of the previous time step	3	3
SLIDE	Amount of sliding	4	4

1. If the value of the element status is:

   0 - OFF

   1 - ON

2. For the thermal and fluid options, analogous items are output. Thermal option output items TEMPI, TEMPJ, TEMPK, TEMPL, DELTEMP, SHEAT, and AHEAT and fluid option output items PRESI, PRESJ, PRESK, PRESL, DELPRES, SFLOW, and AFLOW are respectively analogous to output items UI, UJ, UK, UL, STRETCH, SFORCE, and AFORCE.

3. Output only if FSLIDE is greater than zero. If the value of the sliding status is:

   0 - No sliding

   1 - Sliding right (node J moving to right of node I)

   -1- Sliding left (node J moving to left of node I)

4. If FSLIDE is greater than zero

5. Available only at centroid as a ***GET** item.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Table 37.3: COMBIN37 Item and Sequence Numbers (p. 218) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) of this manual for more information. The following notation is used in Table 37.3: COMBIN37 Item and Sequence Numbers (p. 218):

Name
output quantity as defined in the Table 37.2: COMBIN37 Element Output Definitions (p. 217)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 37.3: COMBIN37 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
SFORCE	SMISC	1
AFORCE	SMISC	2
STAT	NMISC	1
OLDST	NMISC	2
SLSTAT	NMISC	3
OLDSLS	NMISC	4
STRETCH	NMISC	5
UI	NMISC	6
UJ	NMISC	7
UK	NMISC	8
UL	NMISC	9
CPAR	NMISC	10
SLIDE	NMISC	11

Analogous thermal and fluid option output items use the same item and sequence numbers. See footnote 2 of Table 37.2: COMBIN37 Element Output Definitions (p. 217).

COMBIN37 Assumptions and Restrictions

- The element may have only one degree of freedom per node which is specified in the nodal coordinate system (see Elements That Operate in the Nodal Coordinate System (p. 57)).
- The element assumes only a one-dimensional action.
- Nodes I and J may be anywhere in space (preferably coincident).
- No moment effects are included due to noncoincident nodes. That is, if the nodes are offset from the line of action, moment equilibrium may not be satisfied.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The nonlinear capabilities of the element operate only in static and nonlinear transient dynamic analyses.
- If used in other analysis types, the element maintains its initial status (on or off), throughout the analysis.
- The real constants for this element are not allowed to be changed from their initial values.
- The element can not be deactivated with the **EKILL** command.
- Only the lumped mass matrix is available.

COMBIN37 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Structural

- KEYOPT(2) = 8 is not allowed.
- KEYOPT(3) = 8 is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

FLUID38

Dynamic Fluid Coupling

MP ME ST <> <> <> <> <> PP EME MFS
Product Restrictions

FLUID38 Element Description

FLUID38 is used to represent a dynamic coupling between two points of a structure. The coupling is based on the dynamic response of two points connected by a constrained mass of fluid. The points represent the centerlines of concentric cylinders. The fluid is contained in the annular space between the cylinders. The cylinders may be circular or have an arbitrary cross-section. The element has two degrees of freedom per node: for example, translations in the nodal x and z directions. The axes of the cylinders are then assumed to be in the nodal y directions. The element may be used in any structural dynamic analysis. See FLUID38 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 38.1: FLUID38 Geometry

FLUID38 Input Data

The node locations and the coordinate system for this element are shown in Figure 38.1: FLUID38 Geometry (p. 221). The element is defined by two nodes and several real constants. The real constants are defined in Table 38.1: FLUID38 Real Constants (p. 223).

KEYOPT(3) is used to select the form of the fluid coupling element. The form of the element determines the real constants required, the material properties (if any), and the matrices calculated. The density is input as material property DENS and is evaluated at the average of the two node temperatures. The damping matrix is calculated only if F is nonzero. KEYOPT(6) is used to select the direction of operation for the element. If KEYOPT(6) = 1, the X and Y labels used in this description should be interchanged. Similarly, if KEYOPT(6) = 3, interchange the Z and Y labels.

A summary of the element input is given in "FLUID38 Input Summary" (p. 222). A general description of element input is given in Element Input (p. 43).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FLUID38 Input Summary

Nodes

I, J

Degrees of Freedom

UX, UZ if KEYOPT(6) = 0 or 2, or
UY, UZ if KEYOPT(6) = 1, or
UX, UY if KEYOPT(6) = 3

Real Constants

If KEYOPT(3) = 0:

R2, R1, L, F, DX, DZ,
WX, WZ

If KEYOPT(3) = 2:

M2, M1, MHX, MHZ, DX, DZ,
WX, WZ, CX, CZ
See Table 38.1: FLUID38 Real Constants (p. 223) for a description of the real constants

Material Properties

MP command: DENS if KEYOPT (3) = 0
None if KEYOPT (3) = 2

Surface Loads

None

Body Loads

Temperature --
T(I), T(J)

Special Features

None

KEYOPT(3)

Cross-section of cylinders:

0 --
Concentric circular cylinders

2 --
Concentric arbitrary cylinders

KEYOPT(6)

Flow axis parallel to:

0, 2 --
Nodal Y axis (UX, UZ degrees of freedom)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
Nodal X axis (UX, UZ degrees of freedom)

3 --
Flow axis parallel to nodal Z axis (UX, UY degrees of freedom)

Table 38.1: FLUID38 Real Constants

No.	Name	Description
Concentric Circular Cylinders: KEYOPT(3) = 0		
1	R2	Radius of outer cylinder (length); node J refers to outer boundary
2	R1	Radius of inner cylinder (length); node I refers to inner boundary
3	L	Length of cylinders
4	F	Darcy friction factor for turbulent flow
5	DX	Estimate of maximum relative amplitude DX
6	DZ	Estimate of maximum relative amplitude DZ
7	WX	Estimate of resonant X frequency (Rad/Time)
8	WZ	Estimate of resonant Z frequency (Rad/Time)
Concentric Arbitrary Cylinders: KEYOPT(3) = 2		
1	M2	Mass of fluid that could be contained within the outer boundary (Boundary 2) in absence of inner boundary.
2	M1	Mass of fluid displaced by the inner boundary (Boundary 1)
3	MHX	Hydrodynamic mass in X direction
4	MHZ	Hydrodynamic mass in Z direction
5	DX	Estimate of maximum relative amplitude DX
6	DZ	Estimate of maximum relative amplitude DZ
7	WX	Estimate of resonant X frequency (Rad/Time)
8	WZ	Estimate of resonant Z frequency (Rad/Time)
9	CX	Flow and geometry constant for X motion (mass/lenght)
10	CZ	Flow and geometry constant for Z motion (mass/length)

FLUID38 Output Data

There is no element solution output associated with the element unless element reaction forces and/or energies are requested.

FLUID38 Assumptions and Restrictions

- The element operates in the nodal coordinate system (see Elements That Operate in the Nodal Coordinate System (p. 57)).
- No fluid coupling exists in the flow axis direction.
- The element has no nodal coordinate system transformation to account for nonparallel nodal coordinate systems.
- Nodes I and J may be located anywhere in space (preferably coincident).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The lumped mass option [**LUMPM**] is not available with this element.

FLUID38 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

COMBIN39

Nonlinear Spring

MP ME ST <> <> <> <> <> PP EME MFS
Product Restrictions

COMBIN39 Element Description

COMBIN39 is a unidirectional element with nonlinear generalized force-deflection capability that can be used in any analysis. The element has longitudinal or torsional capability in 1-D, 2-D, or 3-D applications. The longitudinal option is a uniaxial tension-compression element with up to three degrees of freedom at each node: translations in the nodal x, y, and z directions. No bending or torsion is considered. The torsional option is a purely rotational element with three degrees of freedom at each node: rotations about the nodal x, y, and z axes. No bending or axial loads are considered.

The element has large displacement capability for which there can be two or three degrees of freedom at each node.

See COMBIN39 in the *Mechanical APDL Theory Reference* for more details about this element. The element has no mass or thermal capacitance. These may be added by using the appropriate elements (see MASS21 and MASS71). A bilinear force-deflection element with damping and gaps is also available (COMBIN40).

Figure 39.1: COMBIN39 Geometry

COMBIN39 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 39.1: COMBIN39 Geometry (p. 225). The element is defined by two (preferably coincident) node points and a generalized force-deflection curve. The points on this curve (D1, F1, etc.) represent force (or moment) versus relative translation (or rotation) for structural analyses, and heat (or flow) rate versus temperature (or pressure) difference for a thermal analyses. The loads should be defined on a full 360° basis for an axisymmetric analysis.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The force-deflection curve should be input such that deflections are increasing from the third (compression) to the first (tension) quadrants. Adjacent deflections should not be nearer than 1E-7 times total input deflection range. The last input deflection must be positive. Segments tending towards vertical should be avoided. If the force-deflection curve is exceeded, the last defined slope is maintained, and the status remains equal to the last segment number. If the compressive region of the force-deflection curve is explicitly defined (and not reflected), then at least one point should also be at the origin (0,0) and one point in the first (tension) quadrant. If KEYOPT(2) = 1 (no compressive resistance), the force-deflection curve should not extend into the third quadrant. Note that this tension-only behavior can cause convergence difficulties similar to those that can be experienced by contact elements. See the *Contact Technology Guide*, as well as various contact element descriptions, for guidelines on overcoming convergence difficulties. Note that the number of points defining the loading curve (20 points) can be effectively doubled by using the reflective option.

Slopes of segments may be either positive or negative, except that the slopes at the origin must be positive and, if KEYOPT(1) = 1, slopes at the ends may not be negative. Also, if KEYOPT(1) = 1, force-deflection points may not be defined in the second or fourth quadrants and the slope of any segment may not be greater than the slope of the segment at the origin in that quadrant.

The KEYOPT(1) option allows either unloading along the same loading curve or unloading along the line parallel to the slope at the origin of the curve. This second option allows modeling of hysteretic effects. As illustrated in Figure 39.2: COMBIN39 Force-Deflection Curves (p. 229), the KEYOPT(2) option provides several loading curve capabilities.

The KEYOPT(3) option selects one degree of freedom. This may be a translation, a rotation, a pressure or a temperature.

Alternately, the element may have more than one type of degree of freedom (KEYOPT(4) > 0). The two nodes defining the element should not be coincident, since the load direction is colinear with the line joining the nodes. The longitudinal option (KEYOPT(4) = 1 or 3) creates a uniaxial tension-compression element with two or three translational degrees of freedom at each node. No bending or torsion is considered. The torsional option (KEYOPT(4) = 2) creates a purely rotational element with three rotational degrees of freedom at each node. No bending or axial loads are considered. The stress stiffening capability is applicable when forces are applied, but not when torsional loads are applied.

The element has large displacement capability with two or three degrees of freedom for each node when you use KEYOPT(4) = 1 or 3 in combination with **NLGEOM**,ON.

Convergence difficulties caused by moving through rapid changes of the slope (tangent) of the force-deflection diagram are sometimes helped by use of line search (**LNSRCH**,ON).

A summary of the element input is given in "COMBIN39 Input Summary" (p. 226). A general description of element input is given in Element Input (p. 43).

COMBIN39 Input Summary

Nodes

I, J

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ, PRES, or TEMP.
Make 1-D choices with KEYOPT(3).
Make limited 2- or 3-D choices with KEYOPT(4).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Real Constants

D1, F1, D2, F2, D3, F3,
D4, F4, ..., D20, F20
See Table 39.1: COMBIN39 Real Constants (p. 228) for a description of the real constants

Material Properties
MP command: BETD

Surface Loads
None

Body Loads
None

Special Features

Large displacement
Linear perturbation
Nonlinearity
Stress stiffening

KEYOPT(1)
Unloading path:

0 --
Unload along same loading curve

1 --
Unload along line parallel to slope at origin of loading curve

KEYOPT(2)
Element behavior under compressive load:

0 --
Compressive loading follows defined compressive curve (or reflected tensile curve if not defined)

1 --
Element offers no resistance to compressive loading

2 --
Loading initially follows tensile curve then follows compressive curve after buckling (zero or negative stiffness)

KEYOPT(3)
Element degrees of freedom (1-D) (KEYOPT(4) overrides KEYOPT(3)):

0, 1 --
UX (Displacement along nodal X axes)

2 --
UY (Displacement along nodal Y axes)

3 --
UZ (Displacement along nodal Z axes)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4 --
ROTX (Rotation about nodal X axes)

5 --
ROTY (Rotation about nodal Y axes)

6 --
ROTZ (Rotation about nodal Z axes)

7 --
PRES

8 --
TEMP

KEYOPT(4)
Element degrees of freedom (2-D or 3-D):

0 --
Use any KEYOPT(3) option

1 --
3-D longitudinal element (UX, UY and UZ)

2 --
3-D torsional element (ROTX, ROTY and ROTZ)

3 --
2-D longitudinal element. (UX and UY) Element must lie in an X-Y plane

KEYOPT(6)
Element output:

0 --
Basic element printout

1 --
Also print force-deflection table for each element (only at first iteration of problem)

Table 39.1: COMBIN39 Real Constants

No.	Name	Description
1	D1	D value for the first point on force-deflection curve
2	F1	F value for the first point on force-deflection curve
3	D2	D value for the second point on force-deflection curve
4	F2	F value for the second point on force-deflection curve
5, ... 40	D3, F3, etc.	Continue input of D and F values up to a maximum of 20 points on the force-deflection curve

COMBIN39 Output Data

The solution output associated with the element is in two forms:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Nodal degree of freedom results included in the overall nodal solution
- Additional element output as shown in Table 39.2: COMBIN39 Element Output Definitions (p. 230)

The nodal displacements and forces correspond to the degrees of freedom selected with KEYOPT(3). For an axisymmetric analysis, the element forces are expressed on a full 360° basis. The element value STRETCH is the relative deflection at the end of the substep (e.g., UX(J) - UX(I) - UORIG, etc.). STAT and OLDST describe the curve segment number at the end of the current and previous substeps, respectively. STAT or OLDST = 0 indicates nonconservative unloading (KEYOPT(1) = 1). A status of 99 or -99 (as shown in Figure 39.1: COMBIN39 Geometry (p. 225)) indicates that the active load point on the curve is outside of the supplied data. The slope of the last segment that is provided is simply continued beyond the last data point.

A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

Figure 39.2: COMBIN39 Force-Deflection Curves

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 39.2: COMBIN39 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
XC, YC, ZC	Location where results are reported	Y	4
UORIG	Origin shift upon reversed loading	1	1
FORCE	Force in element	Y	Y
STRETCH	Relative displacement (includes origin shift)	Y	Y
STAT	Status at end of this time step	2	2
OLDST	Same as STAT except status assumed at beginning of this time step	2	2
UI	Displacement of node I	Y	Y
UJ	Displacement of node J	Y	Y
CRUSH	Status of the force deflection curve after buckling	3	-
SLOPE	Current slope	Y	-

1. If KEYOPT(1) = 1

2. If the value of STAT is:

   0 - Indicates nonconservative unloading

   1-20 - Curve segment number at end of time step

   99 - Beyond last segment (last segment is extrapolated) (negative STAT values indicate compressive segments)

3. If KEYOPT(2) = 2 and if the value of CRUSH is:

   0 - Use defined tensile curve

   1 - Use reflected compressive curve in tension (element has been compressed)

4. Available only at centroid as a ***GET** item.

Table 39.3: COMBIN39 Item and Sequence Numbers (p. 231) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) of this manual for more information. The following notation is used in Table 39.3: COMBIN39 Item and Sequence Numbers (p. 231):

Name
output quantity as defined in the Table 39.2: COMBIN39 Element Output Definitions (p. 230)

Item
predetermined Item label for **ETABLE** command

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

E

sequence number for single-valued or constant element data

Table 39.3: COMBIN39 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FORCE	SMISC	1
STRETCH	NMISC	1
UI	NMISC	2
UJ	NMISC	3
UORIG	NMISC	4
STAT	NMISC	5
OLDST	NMISC	6

COMBIN39 Assumptions and Restrictions

- If you specify KEYOPT(4) = 0, the element has only one degree of freedom per node. This degree of freedom defined by KEYOPT(3), is specified in the nodal coordinate system and is the same for both nodes (see Elements That Operate in the Nodal Coordinate System (p. 57)). KEYOPT(3) also defines the direction of the force. Nodes I and J may be anywhere in space (preferably coincident).
- If you specify KEYOPT(4) $\neq$ 0, the element has two or three displacement degrees of freedom per node. Nodes I and J should not be coincident, since the line joining the nodes defines the direction of the force.
- The element is defined such that a positive displacement of node J relative to node I tends to put the element in tension.
- The element is nonlinear and requires an iterative solution.
- The nonlinear behavior of the element operates only in static and nonlinear transient dynamic analyses.
- As with most nonlinear elements, loading and unloading should occur gradually.
- When the element is also nonconservative, loads should be applied along the actual load history path and in the proper sequence.
- The element can not be deactivated with the **EKILL** command.
- The real constants for this element can not be changed from their initial values.
- Whenever the force that the element carries changes sign, UORIG is reset, and the origin of the force-deflection curve effectively shifts over to the point where the force changed sign. If KEYOPT(2) = 1 and the force tends to become negative, the element "breaks" and no force is transmitted until the force tends to become positive again.
- When KEYOPT(1) = 1, the element is both nonlinear and nonconservative.
- In a thermal analysis, the temperature or pressure degree of freedom acts in a manner analogous to the displacement.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

COMBIN39 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Structural

- KEYOPT(3) = 8 (temperature DOF) is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

COMBIN40

Combination

MP ME ST PR PRN DS <> <> PP EME MFS
Product Restrictions

COMBIN40 Element Description

COMBIN40 is a combination of a spring-slider and damper in parallel, coupled to a gap in series. A mass can be associated with one or both nodal points. The element has one degree of freedom at each node, either a nodal translation, rotation, pressure, or temperature. The mass, springs, slider, damper, and/or the gap may be removed from the element. The element may be used in any analysis. See COMBIN40 in the *Mechanical APDL Theory Reference* for more details about this element. Other elements having damper, slider, or gap capabilities are COMBIN14, MATRIX27, COMBIN37, and COMBIN39.

Figure 40.1: COMBIN40 Geometry

COMBIN40 Input Data

The combination element is shown in Figure 40.1: COMBIN40 Geometry (p. 233). The element is defined by two nodes, two spring constants K1 and K2 (Force/Length), a damping coefficient C (Force*Time/Length), a mass M (Force*Time^2/Length), a gap size GAP (Length), and a limiting sliding force FSLIDE (Force). (Units listed here apply only to KEYOPT(3) = 0, 1, 2, or 3.)

If the element is used in an axisymmetric analysis, these values (except GAP) should be on a full 360° basis. A spring constant of 0.0 (for either K1 or K2, but not both) or a damping coefficient of 0.0 will remove these capabilities from the element. The mass, if any, may be applied at node I or node J or it may be equally distributed between the nodes.

The gap size is defined by the fourth element real constant. If positive, a gap of this size exists. If negative, an initial interference of this amount exists. If GAP = 0.0, the gap capability is removed from the element. The FSLIDE value represents the absolute value of the spring force that must be exceeded before sliding occurs. If FSLIDE is 0.0, the sliding capability of the element is removed, that is, a rigid connection is assumed.

A "breakaway" feature is available to allow the element stiffness (K1) to drop to zero once a limiting force |FSLIDE| has been reached. The limit is input as -|FSLIDE| and is applicable to both tensile breaking and compressive crushing. A "lockup" feature may be selected with KEYOPT(1). This feature removes the gap opening capability once the gap has closed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The force-deflection relationship for the combination element is as shown in Figure 40.2: COMBIN40 Behavior (p. 236) (for no damping). If the initial gap is identically zero, the element responds as a spring-damper-slider element having both tension and compression capability. If the gap is not initially zero, the element responds as follows: when the spring force (F1+F2) is negative (compression), the gap remains closed and the element responds as a spring-damper parallel combination. As the spring force (F1) increases beyond the FSLIDE value, the element slides and the F1 component of the spring force remains constant. If FSLIDE is input with a negative sign, the stiffness drops to zero and the element moves with no resisting F1 spring force. If the spring force becomes positive (tension), the gap opens and no force is transmitted. In a thermal analysis, the temperature or pressure degree of freedom acts in a manner analogous to the displacement.

The element has only the degrees of freedom selected with KEYOPT(3). The KEYOPT(3) = 7 and 8 options (pressure and temperature DOFs) allow the element to be used in a thermal analysis (with thermal equivalent real constants).

A summary of the element input is given in "COMBIN40 Input Summary" (p. 234). A general description of element input is given in Element Input (p. 43).

COMBIN40 Input Summary

Nodes

I, J

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ, PRES, or TEMP (depending on KEYOPT(3) below)

Real Constants

Units for real constants will depend on the KEYOPT(3) setting.

K1 - Spring constant
C - Damping coefficient
M - Mass
GAP - Gap size
FSLIDE - Limiting sliding force
K2 - Spring constant (par to slide)

Note

If GAP is exactly zero, the interface cannot open. If GAP is negative, there is an initial interference. If FSLIDE is exactly zero, the sliding capability is removed. If FSLIDE is negative, the "breakaway" feature is used.

Material Properties

MP command: ALPD, BETD

Surface Loads

None

Body Loads

None

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Special Features

Adaptive descent
Nonlinearity (unless both GAP and FSLIDE equal zero)

KEYOPT(1)
Gap behavior:

0 --
Standard gap capability

1 --
Gap remains closed after initial contact ("lockup")

KEYOPT(3)
Element degrees of freedom:

0, 1 --
UX (Displacement along nodal X axes)

2 --
UY (Displacement along nodal Y axes)

3 --
UZ (Displacement along nodal Z axes)

4 --
ROTX (Rotation about nodal X axes)

5 --
ROTY (Rotation about nodal Y axes)

6 --
ROTZ (Rotation about nodal Z axes)

7 --
PRES

8 --
TEMP

KEYOPT(4)
Element output:

0 --
Produce element printout for all status conditions

1 --
Suppress element printout if gap is open (STAT = 3)

KEYOPT(6)
Mass location:

0 --
Mass at node I

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
Mass equally distributed between nodes I and J

2 --
Mass at node J

COMBIN40 Output Data

The solution output associated with the element is in two forms:

- Nodal degree of freedom results included in the overall nodal solution
- Additional element output as shown in Figure 40.2: COMBIN40 Behavior (p. 236)

Several items are illustrated in Figure 40.2: COMBIN40 Behavior (p. 236). The displacement direction corresponds to the nodal coordinate direction selected with KEYOPT(3). The value STR is the spring displacement at the end of this substep, STR = U(J)-U(I)+GAP-SLIDE. This value is used in determining the spring force. For an axisymmetric analysis, the element forces are expressed on a full 360° basis. The value SLIDE is the accumulated amount of sliding at the end of this substep relative to the starting location.

STAT describes the status of the element at the end of this substep for use in the next substep. If STAT = 1, the gap is closed and no sliding occurs. If STAT = 3, the gap is open. If STAT = 3 at the end of a substep, an element stiffness of zero is being used. A value of STAT = +2 indicates that node J moves to the right of node I. STAT = -2 indicates a negative slide. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

Figure 40.2: COMBIN40 Behavior

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 40.1: COMBIN40 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
XC, YC, ZC	Location where results are reported	Y	2
SLIDE	Amount of sliding	Y	Y
F1	Force in spring 1	Y	Y
STR1	Relative displacement of spring 1	Y	Y
STAT	Element status	1	1
OLDST	STAT value of the previous time step	1	1
UI	Displacement of node I	Y	Y
UJ	Displacement of node J	Y	Y
F2	Force in spring 2	Y	Y
STR2	Relative displacement of spring 2	Y	Y

1. If the value of STAT is:

   1 - Gap closed (no sliding)

   2 - Sliding right (node J moving to right of node I)

   -2 - Sliding left (node J moving to left of node I)

   3 - Gap open

2. Available only at centroid as a ***GET** item.

Table 40.2: COMBIN40 Item and Sequence Numbers (p. 238) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) of this manual for more information. The following notation is used in Table 40.2: COMBIN40 Item and Sequence Numbers (p. 238):

Name
output quantity as defined in the Table 40.1: COMBIN40 Element Output Definitions (p. 237)

Item
predetermined Item label for **ETABLE** command

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

E

sequence number for single-valued or constant element data

Table 40.2: COMBIN40 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
F1	SMISC	1
F2	SMISC	2
STAT	NMISC	1
OLDST	NMISC	2
STR1	NMISC	3
STR2	NMISC	4
UI	NMISC	5
UJ	NMISC	6
SLIDE	NMISC	7

COMBIN40 Assumptions and Restrictions

- The element has only one degree of freedom per node which is specified in the nodal coordinate system (see Elements That Operate in the Nodal Coordinate System (p. 57)).
- The element assumes only a 1-D action.
- Nodes I and J may be anywhere in space (preferably coincident).
- The element is defined such that a positive displacement of node J relative to node I tends to open the gap. If, for a given set of conditions, nodes I and J are interchanged, the gap element acts as a hook element, i.e., the gap closes as the nodes separate.
- The real constants for this element can not be changed from their initial values.
- The element can not be deactivated with the **EKILL** command.
- The nonlinear options of the element operate only in static and nonlinear transient dynamic (**TRNOPT**,FULL) analyses.
- If used in other analysis types, the element maintains its initial status throughout the analysis.
- A 0.0 value for GAP or FSLIDE removes the gap or sliding capability, respectively, from the element.
- The mass, if any, is 1-D.
- The element requires an iterative solution if GAP and/or FSLIDE are nonzero.
- A stiffness (K1 or K2) must be defined if the gap capability is used. Unreasonably high stiffness values should be avoided.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The rate of convergence may decrease as the stiffness increases. If FSLIDE is not equal to zero, the element is nonconservative as well as nonlinear. Nonconservative elements require that the load be applied very gradually, along the actual load history path, and in the proper sequence (if multiple loadings exist).
- Only the lumped mass matrix is available.

COMBIN40 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

Structural Analysis:

- No damping capability; CV1 and CV2 are not allowed.
- Only stress stiffening and large deflections are allowed.
- KEYOPT(3) = 7 or 8 is not allowed.
- The BETD material property is not allowed.
- FSLIDE and K2 not allowed.

ANSYS Structural

- KEYOPT(3) = 7 and 8 (pressure and temperature DOFs) are not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

SHELL41

Membrane Shell

MP ME ST PR PRN <> <> <> PP EME MFS
Product Restrictions

SHELL41 Element Description

Although this legacy element is available for use in your analysis, ANSYS, Inc. recommends using a current-technology element such as SHELL181 (KEYOPT(1) = 1 or KEYOPT(3) = 2).

SHELL41 is a 3-D element having membrane (in-plane) stiffness but no bending (out-of-plane) stiffness. It is intended for shell structures where bending of the elements is of secondary importance. The element has three degrees of freedom at each node: translations in the nodal x, y, and z directions.

The element has variable thickness, stress stiffening, large deflection, and a cloth option. See SHELL41 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 41.1: SHELL41 Geometry

x_{IJ} = Element x-axis if ESYS is not supplied.

x = Element x-axis if ESYS is supplied.

SHELL41 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 41.1: SHELL41 Geometry (p. 241). The element is defined by four nodes, four thicknesses, a material direction angle and the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems (p. 55). The element x-axis may be rotated by an angle THETA (in degrees).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The element may have variable thickness. The thickness is assumed to vary smoothly over the area of the element, with the thickness input at the four nodes. If the element has a constant thickness, only TK(I) need be input. If the thickness is not constant, all four thicknesses must be input. The elastic foundation stiffness (EFS) is defined as the pressure required to produce a unit normal deflection of the foundation. The elastic foundation capability is bypassed if EFS is less than, or equal to, zero. ADMSUA is the added mass per unit area.

Element loads are described in Nodal Loading (p. 49). Pressures may be input as surface loads on the element faces as shown by the circled numbers on Figure 41.1: SHELL41 Geometry (p. 241). Positive pressures act into the element. Because shell edge pressures are input on a per-unit-length basis, per-unit-area quantities must be multiplied by the shell thickness. The pressure loading is converted to equivalent element loads applied at the nodes. Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

Use KEYOPT(1) for a tension-only option. This nonlinear option acts like a cloth in that tension loads will be supported but compression loads will cause the element to wrinkle.

You should *not* use this "cloth" option to model cloth materials, since real cloth materials do contain some bending stiffness. You can use the cloth option to efficiently model regions where wrinkling is to be approximated, such as for shear panels in aircraft structures. Wrinkling for this type of application may be in one (or both) orthogonal directions. If you do need to model a real cloth material, you can use the cloth option to simulate the tension part of the loading, but you will need to superimpose a very thin regular shell element to include a bending stiffness for the material. Superimposing a thin shell may also aid solution stability.

Any out-of-planeness within the element or round off-error in nodal location may cause an instability in the displacement solution. To counteract this, a slight normal stiffness may be added to the element with the EFS real constant. KEYOPT(2) is used to include or suppress the extra displacement shapes. KEYOPT(4) provides various element printout options (see Element Solution (p. 51)).

A summary of the element input is given in "SHELL41 Input Summary" (p. 242). A general description of element input is given in Element Input (p. 43).

SHELL41 Input Summary

Nodes

I, J, K, L

Degrees of Freedom

UX, UY, UZ

Real Constants

TK(I), TK(J), TK(K), TK(L), THETA, EFS,
ADMSUA
See Table 41.1: SHELL41 Real Constants (p. 244) for a description of the real constants

Material Properties

MP command: EX, EY, PRXY or NUXY, ALPX, ALPY (or CTEX, CTEY *or* THSX,THSY), DENS, GXY, ALPD, BETD (X-direction defined by THETA real constant)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface Loads

Pressures --

face 1 (I-J-K-L) (bottom, in +Z direction), face 2 (I-J-K-L) (top, in -Z direction), face 3 (J-I), face 4 (K-J), face 5 (L-K), face 6 (I-L)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L)

Special Features

Adaptive descent
Birth and death
Large deflection
Nonlinearity (if KEYOPT(1) = 2)
Stress stiffening

KEYOPT(1)
Element stiffness behavior:

0 --
Stiffness acts in both tension and compression

2 --
Stiffness acts in tension, collapses in compression ("cloth" option)

KEYOPT(2)
Extra displacement shapes:

0 --
Include extra displacement shapes

1 --
Suppress extra displacement shape

KEYOPT(4)
Extra stress output:

0 --
Basic element printout

1 --
Repeat basic printout at integration points

2 --
Nodal stress printout

KEYOPT(5)
Member force output:

0 --
No member force printout

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
Print member forces in the element coordinate system

KEYOPT(6)
Edge output (isotropic material):

0 --
No edge printout

1 --
Edge printout for midpoint of side I-J

2 --
Edge printout for midpoints of both sides I-J and K-L

Note

Edge printout valid only for isotropic materials

Table 41.1: SHELL41 Real Constants

No.	Name	Description
1	TK(I)	Shell thickness at node I
2	TK(J)	Shell thickness at node J (defaults to TK(I))
3	TK(K)	Shell thickness at node K (defaults to TK(I))
4	TK(L)	Shell thickness at node L (defaults to TK(I))
5	THETA	Element x-axis rotation
6	EFS	Elastic foundation stiffness
7	ADMSUA	Added mass/unit area

SHELL41 Output Data

The solution output associated with the element is in two forms:

- Nodal degree of freedom results included in the overall nodal solution
- Additional element output as shown in Table 41.2: SHELL41 Element Output Definitions (p. 245)

Several items are illustrated in Figure 41.2: SHELL41 Stress Output (p. 245). The element stress directions correspond to the element coordinate directions. Edge stresses are defined parallel and perpendicular to the IJ edge (and the KL edge). A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 41.2: SHELL41 Stress Output

x_{IJ} = Element x-axis if ESYS is not supplied.

x = Element x-axis if ESYS is supplied.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 41.2: SHELL41 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
AREA	Surface area	Y	Y
XC, YC, ZC	Location where results are reported	Y	4
PRES	Pressures P1 at nodes I, J, K, L; P2 at I, J, K, L; P3 at J, I; P4 at K, J; P5 at L, K; P6 at I, L	Y	Y
TEMP	Temperatures T(I), T(J), T(K), T(L)	Y	Y
S:X, Y, Z, XY	Stresses	Y	Y
S:1, 2, 3	Principal stress	Y	Y
S:INT	Stress intensity	Y	Y
S:EQV	Equivalent stress	Y	Y
EPEL:X, Y, Z, XY	Average elastic strain	Y	Y
EPEL:EQV	Equivalent elastic strain	Y	Y
EPTH:X, Y, Z, XY	Average thermal strain	Y	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
EPTH:EQV	Equivalent thermal strain	Y	Y
ANGLES	Diagonal tension angles (degrees) between element x-axis and tensile stress directions	1	1
CURRENT STATS.	Element statuses at end of this time step	2	2
OLD STATUSES	Element statuses at end of previous time step	2	2
TEMP	Edge average temperature	3	3
EPEL(PAR, PER, Z)	Edge elastic strains (parallel, perpendicular, Z)	3	3
S(PAR, PER, Z)	Edge stresses (parallel, perpendicular, Z)	3	3
SINT	Edge stress intensity	3	3
SEQV	Edge equivalent stress	3	3
FX, FY, FZ	Nodal forces	-	Y

1. Output at the integration points only if KEYOPT(1) = 2 (meaningful only if STAT = 1)
2. Output at the integration points only if KEYOPT(1) = 2. The element status is given by the following values:

   0 - Tension in both (orthogonal) directions

   1 - Tension in one direction, collapse in other direction

   2 - Collapse in both directions
3. Edge I-J output, if KEYOPT(6) is greater than zero.
4. Available only at centroid as a ***GET** item.

Table 41.3: SHELL41 Miscellaneous Element Output

Description	Names of Items Output	O	R
Integration Point Stress Solution	TEMP, S(X, Y, Z, XY), SINT, SEQV	1	-
Nodal Stress Solution	TEMP, S(X, Y, Z, XY), SINT, SEQV	2	-
Edge K-L	TEMP, EPEL(PAR, PER, Z), S(PAR, PER, Z), SINT, SEQV	3	-
Member Forces	FX, FY, FZ	4	-

1. Output at each integration point, if KEYOPT(4) = 1
2. Output at each node, if KEYOPT(4) = 2
3. Output if KEYOPT(6) = 2
4. Output at each node (in the element coordinate system) if KEYOPT(5) = 1

Table 41.4: SHELL41 Item and Sequence Numbers (p. 247) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic*

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Analysis Guide and The Item and Sequence Number Table (p. 52) of this manual for more information. The following notation is used in Table 41.4: SHELL41 Item and Sequence Numbers (p. 247):

Name
output quantity as defined in the Table 41.2: SHELL41 Element Output Definitions (p. 245)

Item
predetermined Item label for **ETABLE** command

I,J,K,L
sequence number for data at nodes I,J,K,L

Table 41.4: SHELL41 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	I	J	K	L
FX	SMISC	1	4	7	10
FY	SMISC	2	5	8	11
FZ	SMISC	3	6	9	12
P1	SMISC	13	14	15	16
P2	SMISC	17	18	19	20
P3	SMISC	22	21	-	-
P4	SMISC	-	24	23	-
P5	SMISC	-	-	26	25
P6	SMISC	27	-	-	28
S:1	NMISC	1	6	11	16
S:2	NMISC	2	7	12	17
S:3	NMISC	3	8	13	18
S:INT	NMISC	4	9	14	19
S:EQV	NMISC	5	10	15	20

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	Corner Location			
		1	2	3	4
ANGLE	NMISC	21	23	25	27
STAT	NMISC	22	24	26	28

SHELL41 Assumptions and Restrictions

- The four nodes defining the element should lie in an exact flat plane; however, a small out-of-plane tolerance is permitted so that the element may have a slightly warped shape.
- A slightly warped element will produce a warning message. If the warping is too severe, a fatal message results and a triangular element should be used (see Degenerated Shape Elements (p. 39)).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Zero area elements are not allowed.
- TK(I) must not be zero.
- The element must not taper down to a zero thickness at any corner.
- A triangular element may be formed by defining duplicate K and L node numbers (see Degenerated Shape Elements (p. 39)).
- The extra shapes are automatically deleted for triangular elements so that a constant strain element results.
- The triangular shape is required for large deflection analyses since a four-node element may warp during deflection.
- Edge stress printout is valid only if the conditions described in Element Solution (p. 51) are met.

Modeling hints:

- An assembly of SHELL41 elements describing a flat plane should be *exactly* flat; otherwise singularities may develop in the direction perpendicular to the plane.
- Stress stiffening will help stabilize the solution after the first substep if the membrane element is in a tension field.
- An assemblage of flat elements can produce an approximation to a curved surface, but each flat element should not extend over more than a 15° arc.

SHELL41 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

INFIN47

3-D Infinite Boundary

MP ME ST <> <> <> EM <> PP EME MFS
Product Restrictions

INFIN47 Element Description

Although this legacy element is available for use in your analysis, ANSYS, Inc. recommends using a current-technology element such as INFIN111.

INFIN47 is used to model an open boundary of a 3-D unbounded field problem. The element may be a 4-node quadrilateral or a 3-node triangle with a magnetic potential or temperature degree of freedom at each node. The enveloped (or enclosed) element types may be the SOLID5, SOLID96, or SOLID98 magnetic elements or the SOLID70, SOLID90 or SOLID87 thermal solid elements. With the magnetic degree of freedom the analysis may be linear or nonlinear static. With the thermal degree of freedom steady-state or transient analyses (linear or nonlinear) may be done. See INFIN47 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 47.1: INFIN47 Geometry

INFIN47 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 47.1: INFIN47 Geometry (p. 249). The element is defined by 4 nodes, and the material properties. Nonzero material properties must be defined. A triangular element may be formed by defining duplicate K and L node numbers as described in Degenerated Shape Elements (p. 39). The element x-axis is parallel to the I-J side of the element.

The coefficient matrix of this boundary element is, in general, unsymmetric. The matrix is made symmetric by averaging the off-diagonal terms to take advantage of a symmetric solution with a slight decrease in accuracy. KEYOPT(2) can be used to keep an unsymmetric matrix from being made symmetric.

A summary of the element input is given in "INFIN47 Input Summary" (p. 250). A general description of element input is given in Element Input (p. 43).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

INFIN47 Input Summary

Nodes
I, J, K, L

Degrees of Freedom

MAG if KEYOPT(1) = 0
TEMP if KEYOPT(1) = 1

Real Constants
None

Material Properties

EMUNIT command: MUZERO if KEYOPT (1) = 0 (has default value for MKS units or can be set manually), KXX if KEYOPT(1) = 1

Surface Loads
None

Body Loads
None

Element Printout
None

Special Features
None

KEYOPT(1)
Element degree(s) of freedom:

0 --
Magnetic option

1 --
Thermal option

KEYOPT(2)
Coefficient matrix:

0 --
Make the coefficient matrix symmetric

1 --
Coefficient matrix is used as generated (symmetric or unsymmetric, depending on the problem)

INFIN47 Output Data

The boundary element has no output of its own since it is used only to provide a semi-infinite boundary condition to a model consisting of other elements.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

INFIN47 Assumptions and Restrictions

- The 4 nodes defining the element should lie as close as possible to a flat plane; however, a moderate out-of-plane tolerance is permitted so that the element may have a somewhat warped shape.
- An excessively warped element will produce a warning message. In the case of warping errors, triangular elements should be used (see Degenerated Shape Elements (p. 39)).
- Shell element warping is described in detail in Warping Factor in *Mechanical APDL Theory Reference*.
- Zero area elements are not allowed.
- The semi-infinite volume is assumed to be bound on five sides (four, if triangular) by the boundary element and by four semi-infinite radial surfaces (three, if triangular) defined from the global coordinate system origin through nodes I and J, J and K, K and L, and L and I (nodes I and J, J and K, and K and I if triangular).
- The boundary element should be as normal as possible to the radial surfaces.
- Acute or wide intersection angles should be avoided by "filling-in" the model with the other elements so that the line of boundary elements around the model is smooth and concave when viewed from the global coordinate system origin.
- The element assumes that the degree of freedom (DOF) value at infinity is *always* zero (0.0). That is, the DOF value at infinity is *not* affected by **TUNIF**, **D**, or other load commands.
- The boundary element must lie "against" an enclosed element (that is, share the same nodes).
- The exterior semi-infinite domain is assumed to be homogeneous, isotropic, and linear without containing any sources or sinks.
- The origin of the global coordinate system must be inside the model and as centrally located as possible.
- The surface of boundary elements should be located away from the region of interest of the enclosed elements for better accuracy. The surface of boundary elements need not totally surround the model.
- The element may not be deactivated with the **EKILL** command.
- When used in a model with higher order elements SOLID90, SOLID87, and SOLID98, the midside nodes of these elements must be removed at the interface with INFIN47 [**EMID**].
- If KEYOPT(2) = 1, the matrices are presumed to be unsymmetric.
- This element cannot be used in a distributed solution.

INFIN47 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Mechanical

Unless the Emag option is enabled, the following restrictions apply:

- This element does not have magnetic field capability.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The MAG degree of freedom is not active.
- KEYOPT(1) defaults to 1 (TEMP) instead of 0 and cannot be changed.
- The material property MUZRO is not allowed.

ANSYS Emag

- This element has only magnetic field capability, and does not have thermal capability.
- The only active degree of freedom is MAG.
- The only allowable material property is MUZERO.
- KEYOPT(1) can only be set to 0 (default).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MATRIX50

Superelement (or Substructure)

MP ME ST PR PRN <> EM <> PP EME MFS
Product Restrictions

MATRIX50 Element Description

MATRIX50 is a group of previously assembled ANSYS elements that is treated as a single element. The superelement, once generated, may be included in any ANSYS model and used in any analysis type for which it is applicable. The superelement can greatly decrease the cost of many analyses. Once the superelement matrices have been formed, they are stored in a file and can be used in other analyses the same way any other ANSYS elements are used. Multiple load vectors may also be stored with the superelement matrices, thereby allowing various loading options. See MATRIX50 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 50.1: MATRIX50 Schematic

MATRIX50 Input Data

The superelement, which is a mathematical matrix representation of an arbitrary structure, has no fixed geometrical identity and is conceptually shown in Figure 50.1: MATRIX50 Schematic (p. 253). Any analysis using a superelement as one of its element types is called a superelement use pass (or run). The degrees of freedom are the master degrees of freedom specified during the generation pass.

The element name is MATRIX50 (the number 50 or the name MATRIX50 should be input for the variable *ENAME* on the **ET** command). The **SE** command is used to define a superelement. **SE** reads the superelement from `Jobname`.SUB (defaults to `File`.SUB) in the working directory. The material number [**MAT**] is only used when material dependent damping [**MP**,BETD] or electrical permittivity [**MP**,PERX] is an input. The real constant table number [**REAL**] is not used. However, the appropriate element type number [**TYPE**] must be entered.

An element load vector is generated along with the element at each load step of the superelement generation pass. Up to 31 load vectors may be generated. Load vectors may be proportionately scaled in the use pass. The scale factor is input on the element surface load command [**SFE**]. The load label is input as SELV, the load key is the load vector number, *KVAL* determines whether the load vector is real or imaginary, and the load value is the scale factor. The load vector number is determined from the load step number associated with the superelement generation. If a superelement load vector has a zero scale factor (or is not scaled at all), this load vector is not included in the analysis. Any number of load vector-scale factor combinations may be used in the use pass.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In a large rotation analysis (**NLGEOM**,ON), you can use KEYOPT(3) to specify whether the load vectors associated with this element type rotates with the element (as you would for a pressure load) or remains in the original (unrotated) direction (as you would for a non-follower force load); all load vectors (if multiple load vectors) are rotated or left unrotated. You can use KEYOPT(4) to indicate that the superelement was generated with constraints (**D**) so that it cannot translate or rotate freely in the use pass as expected (although you can apply constraints in the use pass to the master degrees of freedom to prevent such motion.)

The KEYOPT(1) option is for the special case where the superelement is to be used with a T^4 nonlinearity, such as for radiation. The `File.SUB` for this case may be constructed directly by the user or may be generated by AUX12, the radiation matrix generator.

A summary of the element input is given in "MATRIX50 Input Summary" (p. 254). A general description of element input is given in Element Input (p. 43).

MATRIX50 Input Summary

Nodes

None input (supplied by element)

Degrees of Freedom

As determined from the included element types (a mixture of multi-field degrees of freedom is not allowed)

Real Constants

None

Material Properties

MP command: ALPD, BETD, PERX

Surface Loads

Surface load effects may be applied through a generated load vector and scale factors. Use the **SFE** command to supply scale factors with *LAB* = SELV, *LKEY* = load vector number (31 maximum), *KVAL* = real or imaginary, and *VAL1* = scale factor.

Body Loads

Body loads may be applied through a generated load vector and scale factors as described for surface loads.

Special Features

Large rotation
Radiation (if KEYOPT(1) = 1)

KEYOPT(1)

Element behavior:

0 --

Normal substructure

1 --

Special radiation substructure

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(3)

Load vector update with large rotations (**NLGEOM**,ON):

0 --

Load vector(s) rotate with the substructure as it rotates

1 --

Load vector(s) do not rotate and remain in their original direction

KEYOPT(4)

Constrained substructure with large rotations (**NLGEOM**,ON):

0 --

Substructure was unconstrained in the generation pass

1 --

Substructure was constrained in the generation pass

KEYOPT(6)

Nodal force output:

0 --

Do not print nodal forces

1 --

Print nodal forces

MATRIX50 Output Data

Displacements and forces may be printed for each (master) degree of freedom in a structural superelement in the "use" pass. Energies are also available when requested. The nodal forces may be output if KEYOPT(6) = 1. The stress distribution within the superelement and the expanded nodal displacements can be obtained from a subsequent stress pass. In addition to the database and substructure files from the generation run, `File.DSUB` must be saved from the superelement "use" pass and input to the expansion pass (if an expansion pass is desired). A general description of solution output is given in Solution Output (p. 50).

MATRIX50 Assumptions and Restrictions

- A superelement may contain elements of any type except Lagrange multiplier-based elements (such as MPC184, PLANE182 with KEYOPT(6) = 1, and CONTA171 with KEYOPT(2) = 3).
- See the **D** command for degree of freedom field groups.
- Superelements of different field types may be mixed within the use run.
- The nonlinear portion of any element included in a superelement will be ignored and any bilinear element will maintain its initial status throughout the analysis.
- Superelements may contain other superelements.
- The relative locations of the superelement attachment points in the nonsuperelement portion of the model (if any) should match the initial superelement geometry.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If the superelement contains a mass matrix, acceleration [**ACEL**] defined in the use run will be applied to the superelement.
- If a load vector containing acceleration effects is also applied in the use run, *both* accelerations (the **ACEL** command and the load vector) will be applied to the superelement.
- Similarly, if the superelement contains a damping matrix (as specified in the generation run) and α and β damping multipliers [**ALPHA** and **BETA**] are defined in the use run, additional damping effects will be applied to the superelement.
- You should be careful to avoid duplicating acceleration and damping effects.
- Pressure and thermal effects may be included in a superelement only through its load vectors.
- The dimensionality of the superelement corresponds to the maximum dimensionality of any element used in its generation. A 2-D superelement should only be used in 2-D analyses, and 3-D superelements in 3-D analyses.
- See Substructuring Analysis in the *Mechanical APDL Theory Reference* for a discussion of the substructure matrix procedure.

MATRIX50 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Structural

- KEYOPT(1) = 0
- The PERX material property is not applicable.

ANSYS Professional

- This element may be used as a radiation substructure only. KEYOPT(1) defaults to 1 instead of 0 and cannot be changed.
- The BETD material property, PERX material property, surface loads, and body loads are not applicable.
- The large rotation special feature is not applicable.

ANSYS Emag

- The BETD material property is not applicable.
- The large rotation special feature is not applicable.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PLANE53

2-D 8-Node Magnetic Solid

MP <> <> <> <> <> EM <> PP EME <>
Product Restrictions

PLANE53 Element Description

Although this legacy element is available for use in your analysis, ANSYS, Inc. recommends using a current-technology element such as PLANE233.

PLANE53 models 2-D (planar and axisymmetric) magnetic fields. The element is defined by 8 nodes and has up to 4 degrees of freedom per node: z component of the magnetic vector potential (AZ), time-integrated electric scalar potential (VOLT), electric current (CURR), and electromotive force (EMF). PLANE53 is based on the magnetic vector potential formulation and is applicable to the following low-frequency magnetic field analyses: magnetostatics, eddy currents (AC time harmonic and transient analyses), voltage forced magnetic fields (static, AC time harmonic and transient analyses), and electromagnetic-circuit coupled fields (static, AC time harmonic and transient analyses). The element has nonlinear magnetic capability for modeling B-H curves or permanent magnet demagnetization curves. See PLANE53 in the *Mechanical APDL Theory Reference* for more details about this element. A similar 4 node element (without voltage forced and magnetic-circuit coupled capability) is PLANE13.

Degree of freedom and force labels for this element are shown in the following table.

Table 53.1: DOF and Force Labels

Degree of Freedom		Force (reaction solution)	
Label	**Label Description**	**Label**	**Label Description**
AZ [1]	Magnetic Vector Potential	CSGZ [2]	Magnetic Current Segment
VOLT	Voltage	AMPS	Current Flow
CURR	Current Flow	VLTG	Voltage Drop
EMF	Electromotive Force	CURT	Current Flow

1. The label used with the CNVTOL command is A.
2. The label used with the CNVTOL command is CSG.

Figure 53.1: PLANE53 Geometry

PLANE53 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 53.1: PLANE53 Geometry (p. 257). The element input data includes 8 nodes and the magnetic material properties. The type of units (MKS or user defined) is specified through the **EMUNIT** command. **EMUNIT** also determines the value of MUZRO. The **EMUNIT** defaults are MKS units and MUZRO = 4 π x 10^{-7} henries/meter. In addition to MUZRO, orthotropic relative permeability is specified through the MURX and MURY material property labels.

MGXX and MGYY represent vector components of the coercive force for permanent magnet materials. The magnitude of the coercive force is the square root of the sum of the squares of the components. The direction of polarization is determined by the components MGXX and MGYY. Permanent magnet polarization and orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems (p. 55). Properties not input default as described in Linear Material Properties in the *Material Reference*. Nonlinear magnetic B-H properties are entered with the **TB** command as described in Material Models in the *Material Reference*. Nonlinear orthotropic magnetic properties may be specified with a combination of a B-H curve and linear relative permeability. The B-H curve will be used in each element coordinate direction where a zero value of relative permeability is specified. Only one B-H curve may be specified per material.

Various combinations of nodal loading are available for this element, depending upon the KEYOPT(1) value. Nodal loads are defined with the **D** and the **F** commands. With the **D** command, the *Lab* variable corresponds to the degree of freedom (VOLT or AZ) and *VALUE* corresponds to the value (time-integrated electric scalar potential or vector magnetic potential). With the **F** command, the *Lab* variable corresponds to the force (AMPS or CSGZ) and *VALUE* corresponds to the value (current or magnetic current segment). The nodal forces, if any, should be input per unit of depth for a plane analysis and on a full 360° basis for an axisymmetric analysis.

Element loads are described in Nodal Loading (p. 49). Maxwell force flags may be input on the element faces indicated by the circled numbers in Figure 53.1: PLANE53 Geometry (p. 257) using the **SF** and **SFE** commands. Surfaces at which magnetic forces are to be calculated may be identified by using the MXWF label on the surface load commands (no value is required.) A Maxwell stress tensor calculation is performed at these surfaces to obtain the magnetic forces. The surface flag should be applied to "air" elements adjacent to the body for which forces are required. Deleting the MXWF specification removes the flag. Lorentz and Maxwell forces may be made available for a subsequent structural analysis with companion elements [**LDREAD**].

The temperature (for material property evaluation only) and magnetic virtual displacement body loads may be input based on their value at the element's nodes or as a single element value [**BF**, **BFE**]. Source current density and voltage body loads may be applied to an area [**BFA**] or input as an element value [**BFE**]. In general, unspecified nodal values of temperatures default to the uniform value specified with the **BFUNIF** or **TUNIF** commands. Calculated Joule heating (JHEAT) may be made available for a subsequent thermal analysis with companion elements [**LDREAD**].

Air elements in which local Jacobian forces are to be calculated may be identified by using nodal values of 1 and 0 for the MVDI label [**BF**]. See the *Low-Frequency Electromagnetic Analysis Guide* for details.

A summary of the element input is given in "PLANE53 Input Summary" (p. 259). A general description of element input is given in Element Input (p. 43). For axisymmetric applications see Harmonic Axisymmetric Elements (p. 83).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PLANE53 Input Summary

Nodes

I, J, K, L, M, N, O, P

Degrees of Freedom

AZ if KEYOPT(1) = 0
AZ, VOLT if KEYOPT(1) = 1
AZ, CURR if KEYOPT(1) = 2
AZ, CURR, EMF if KEYOPT(1) = 3 or 4

Real Constants

CARE, TURN, LENG, DIRZ, FILL, VELOX,
VELOY, OMEGAZ, XLOC, YLOC

See Table 53.2: PLANE53 Real Constants (p. 261) for descriptions of the real constants.

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.

MP command: MURX, MURY, RSVX, MGXX, MGYY

EMUNIT command: MUZERO

Surface Loads

Maxwell Force flag --

face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads

Temperature --

T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

Magnetic Virtual Displacement --

VD(I), VD(J), VD(K), VD(L), VD(M), VD(N),
VD(O), VD(P)

Source Current Density, if KEYOPT(1) = 0 --

spare, spare, JSZ(I), PHASE(I), spare, spare,
JSZ(J), PHASE(J), spare, spare, JSZ(K), PHASE(K),
spare, spare, JSZ(L), PHASE(L) spare, spare,
JSZ(M), PHASE(M), spare, spare, JSZ(N), PHASE(N),
spare, spare, JSZ(O), PHASE(O), spare, spare,
JSZ(P), PHASE(P)

Voltage Loading, if KEYOPT(1) = 2 --

VLTG(I), PHASE(I), VLTG(J), PHASE(J), VLTG(K), PHASE(K),
VLTG(L), PHASE(L), VLTG(M), PHASE(M), VLTG(N), PHASE(N),
VLTG(O), PHASE(O), VLTG(P), PHASE(P)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Special Features

Adaptive descent
Birth and death

KEYOPT(1)
Element degrees of freedom:

0 --
AZ degree of freedom: static domain, induced eddy current domain

1 --
AZ, VOLT degrees of freedom: current-fed massive conductor

2 --
AZ, CURR degrees of freedom: voltage-fed stranded coil

3 --
AZ, CURR, EMF degrees of freedom: circuit-coupled stranded coil

4 --
AZ, CURR, EMF degrees of freedom: circuit-coupled massive conductor

KEYOPT(2)
Element conventional velocity:

0 --
Velocity effects ignored

1 --
Conventional velocity formulation (not available if KEYOPT(1) = 2, 3, or 4)

KEYOPT(3)
Element behavior:

0 --
Plane

1 --
Axisymmetric

KEYOPT(4)
Element coordinate system:

0 --
Element coordinate system is parallel to the global coordinate system

1 --
Element coordinate system is based on the element I-J side

KEYOPT(5)
Extra element output:

0 --
Basic element printout

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
Integration point printout

2 --
Nodal magnetic field printout

KEYOPT(7)
Store magnetic forces for coupling with elements:

0 --
Midside node (higher-order) structural elements

1 --
Non-midside node structural elements

Table 53.2: PLANE53 Real Constants

No.	Name	Description
KEYOPT(1) ≥ 2 - voltage forced or electromagnetic-circuit coupled analyses (coils or massive conductors)		
1	CARE	Coil cross-sectional area; required when KEYOPT(1) = 2, 3, 4
2	TURN	Total number of coil turns (stranded coil only), default is 1; KEYOPT(1) = 2, 3
3	LENG	Coil length in Z-direction, (required for planar models only), default is 1 meter; KEYOPT(1) = 2, 3, 4
4	DIRZ	1 for current in positive z-axis, -1 for current in negative z-axis; KEYOPT(1) = 2, 3, 4
5	FILL	Coil fill factor *; KEYOPT(1) = 2, 3
KEYOPT(2) = 1 (and KEYOPT(1) = 0 or 1) - Velocity effects of a conducting body		
6	VELOX	Velocity component in X-direction (global Cartesian)
7	VELOY	Velocity component in Y-direction (global Cartesian)
8	OMEGAZ	Angular (rotational) velocity (Hz, cycles/sec) about the Z-axis (global Cartesian), at the pivot point
9	XLOC	Pivot point X-location (global Cartesian coordinate)
10	YLOC	Pivot point Y-location (global Cartesian coordinate)

* The ratio between the actual conductor area and the area of the coil or element geometry. Often, the element approximation will differ from the profile of the conductor, especially in multi-strand coils.

PLANE53 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in Table 53.3: PLANE53 Element Output Definitions (p. 262)

Several items are illustrated in Figure 53.2: PLANE53 Magnetic Element Output (p. 262). The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 53.2: PLANE53 Magnetic Element Output

Because of different sign conventions for Cartesian and polar coordinate systems, magnetic flux density vectors point in opposite directions for planar (KEYOPT(3) = 0) and axisymmetric (KEYOPT(3) = 1) analyses.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 53.3: PLANE53 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	2
CENT:X, Y	Global location XC, YC	Y	Y
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	Y	Y
LOC	Output location (X, Y)	1	-
MUX, MUY	Magnetic secant permeability	1	1
H:X, Y	Magnetic field intensity components	1	1
H:SUM	Vector magnitude of H	1	1
B:X, Y	Magnetic flux density components (X, Y)	1	1
B:SUM	Vector magnitude of B	1	1
JSZ	Source current density, valid for static analysis only	1	1
JTZ	Total current density	1	1
JHEAT:	Joule heat generation per unit volume	1	1
FJB(X, Y)	Lorentz force components	1	1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
FMX(X, Y)	Maxwell force components	1	1
FVW(X, Y, SUM)	Virtual work force components	1	1
FMAG:X, Y	Combined (FJB or FMX) force components	-	1
ERES	Element resistance value (for stranded coils only)	-	1
EIND	Element inductance value (for stranded coils only)	-	1
DMUXX, DMUYY	Differential permeability	1	1
V:X, Y, SUM	Velocity components	1	1
MRE	Magnetic Reynolds number	1	1
TJB(Z)	Lorentz torque about global Cartesian +Z-axis	1	1
TMX(Z)	Maxwell torque about global Cartesian +Z-axis	1	1
TVW(Z)	Virtual work torque about global Cartesian +Z-axis	1	1

1. The solution value is output only if calculated (based upon input data). The element solution is at the centroid.

Note

JT represents the total measurable current density in a conductor, including eddy current effects, and velocity effects if calculated.

For axisymmetric solutions with KEYOPT(4) = 0, the X and Y directions correspond to the radial and axial directions, respectively.

For harmonic analysis, Joule losses (JHEAT), forces (FJB(X, Y), FMX(X, Y), FVW(X, Y)), and torque (TJB(Z), TMX(Z), TVW(Z)) represent time-average values. These values are stored in both the "Real" and "Imaginary" data sets. The macros **POWERH**, **FMAGSUM**, and **TORQSUM** can be used to retrieve this data.

Inductance values (EIND) obtained for KEYOPT(1) = 2, 3, or 4 are only valid under the following conditions: the problem is linear (constant permeability), there are no permanent magnets in the model, and only a single coil exists in the model.

For KEYOPT(1)=2 and 3, JT represents the effective current density (including non-conducting material represented by the FILL factor). JHEAT represents the effective Joule heat generation rate (including non-conducting material represented by the FILL factor).

2. Available only at centroid as a ***GET** item.

Table 53.4: PLANE53 Miscellaneous Element Output

Description	Names of Items Output	O	R
Nodal Solution	H, HSUM, B, BSUM, FJB, FMX, V, VSUM	1	-

1. Output at each node, if KEYOPT(5) = 2

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Table 53.5: PLANE53 Item and Sequence Numbers (p. 264) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) of this manual for more information. The following notation is used in Table 53.5: PLANE53 Item and Sequence Numbers (p. 264):

Name
output quantity as defined in the Table 53.3: PLANE53 Element Output Definitions (p. 262)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 53.5: PLANE53 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
JSZ	SMISC	1
MUX	NMISC	1
MUY	NMISC	2
FVWX	NMISC	3
FVWY	NMISC	4
FVWSUM	NMISC	5
JTZ	NMISC	7
ERES	NMISC	8
EIND	NMISC	9
DMUXX	NMISC	10
DMUYY	NMISC	11
VX	NMISC	12
VY	NMISC	13
MRE	NMISC	15
TJB(X Y)	NMISC	16
TMX(X, Y)	NMISC	17
TVW(X, Y)	NMISC	18

PLANE53 Assumptions and Restrictions

- The area of the element must be positive.
- The element must lie in a global X-Y plane as shown in Figure 53.1: PLANE53 Geometry (p. 257), and the Y-axis must be the axis of symmetry for axisymmetric analyses. An axisymmetric structure should be modeled in the +X quadrants.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- A face with a removed midside node implies that the potential varies linearly, rather than parabolically, along that face. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.
- Current density loading (**BFE**,,JS) is only valid for the AZ option (KEYOPT(1) = 0). For the VOLT, AZ option (KEYOPT(1) = 1) use **F**,,AMPS.
- When this element does not have the VOLT degree of freedom (KEYOPT(1) = 0), for a harmonic or transient analysis, its behavior depends on the applied load. For a **BFE**,,JS load, the element acts as a stranded conductor. Without **BFE**,,JS loads, it acts as a solid conductor modeling eddy current effects.
  - In this respect, PLANE53 (and PLANE13) are not like the 3-D elements SOLID97, SOLID236, and SOLID237. When SOLID97, SOLID236 and SOLID237 do not have the VOLT degree of freedom, they act as stranded conductors.
- Permanent magnets are not permitted in a harmonic analysis.
- For magnetostatic analyses, the VOLT, AZ option is not allowed.
- For harmonic and transient (time-varying) analyses, the ANSYS product does not support the analysis of coupled velocity and circuit effects.
- A 2-D planar or axisymmetric skin-effect analysis (where eddy current formation is permitted in conducting regions with impressed current loading) is performed by setting KEYOPT(1) = 1, specifying a resistivity, and coupling all VOLT degrees of freedom for elements in each of such regions.
- For voltage forced magnetic field (KEYOPT(1) = 2) and circuit coupled problems (KEYOPT(1) = 3,4), note the following additional restrictions:
  - Only MKS units are allowed.
  - The permeability and conductivity are isotropic and constant.
  - All CURR degrees of freedom in a coil region must be coupled (**CP** command).
  - All EMF degrees of freedom in a coil region must be coupled (**CP** command).
- For circuit coupled transient analyses, use THETA = 1.0, the default value, on the **TINTP** command to specify the backward Euler method. For more information, refer to the *Mechanical APDL Theory Reference*, as well as the description of the **TINTP** command in the *Command Reference*.

For velocity effects (KEYOPT(2) = 1), note the following restrictions:

- Velocity effects are valid only for AZ or AZ-VOLT DOF options.
- Isotropic resistivity.
- Solution accuracy may degrade if the element magnetic Reynolds number is much greater than 1.0. (See the discussion of magnetic field analysis in the *Low-Frequency Electromagnetic Analysis Guide*.)
- If KEYOPT(1) $\geq$ 2 or KEYOPT(2) $\geq$ 1, unsymmetric matrices are produced.
- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, the elements must have the same reaction force (see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PLANE53 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Emag

- The birth and death special feature is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PLANE55

2-D Thermal Solid

MP ME <> PR PRN DS <> <> PP EME MFS
Product Restrictions

PLANE55 Element Description

PLANE55 can be used as a plane element or as an axisymmetric ring element with a 2-D thermal conduction capability. The element has four nodes with a single degree of freedom, temperature, at each node.

The element is applicable to a 2-D, steady-state or transient thermal analysis. The element can also compensate for mass transport heat flow from a constant velocity field. If the model containing the temperature element is also to be analyzed structurally, the element should be replaced by an equivalent structural element (such as PLANE182).

A similar element with midside node capability is PLANE77. A similar axisymmetric element which accepts nonaxisymmetric loading is PLANE75.

An option exists that allows the element to model nonlinear steady-state fluid flow through a porous medium. With this option the thermal parameters are interpreted as analogous fluid flow parameters. See PLANE55 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 55.1: PLANE55 Geometry

PLANE55 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 55.1: PLANE55 Geometry (p. 267). The element is defined by four nodes and the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems (p. 55). Specific heat and density are ignored for steady-state solutions. Properties not input default as described in Linear Material Properties in the *Material Reference*.

Element loads are described in Nodal Loading (p. 49). Convection or heat flux (but not both) and radiation may be input as surface loads at the element faces as shown by the circled numbers on Figure 55.1: PLANE55 Geometry (p. 267).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Heat generation rates may be input as element body loads at the nodes. If the node I heat generation rate HG(I) is input, and all others are unspecified, they default to HG(I).

A mass transport option is available with KEYOPT(8). With this option the velocities VX and VY must be input as real constants (in the element coordinate system). Also, temperatures should be specified along the entire inlet boundary to assure a stable solution. With mass transport, you should use specific heat (C) and density (DENS) material properties instead of enthalpy (ENTH).

The nonlinear porous flow option is selected with KEYOPT(9) = 1. For this option, temperature is interpreted as pressure and the absolute permeabilities of the medium are input as material properties KXX and KYY. Properties DENS and VISC are used for the mass density and viscosity of the fluid. See the *Mechanical APDL Theory Reference* for a description of the properties C and MU, which are used in calculating the coefficients of permeability, with reference to the Z terms ignored. Temperature boundary conditions input with the **D** command are interpreted as pressure boundary conditions, and heat flow boundary conditions input with the **F** command are interpreted as mass flow rate (mass/time).

This element can also have a Z-depth specified by KEYOPT(3) and real constant THK. Be careful when using this option with other physics, especially radiation. Radiation view factors will be based on a unit Z-depth (only).

A summary of the element input is given in "PLANE55 Input Summary" (p. 268). A general description of element input is given in Element Input (p. 43). For axisymmetric applications see Harmonic Axisymmetric Elements (p. 83).

PLANE55 Input Summary

Nodes

I, J, K, L

Degrees of Freedom

TEMP

Real Constants

THK, VX, VY

THK = Thickness (used only if KEYOPT(3) = 3)

VX = Mass transport velocity in X (used only if KEYOPT(8) > 0)

VY = Mass transport velocity in Y (used only if KEYOPT(8) > 0)

Material Properties

MP command: KXX, KYY, DENS, C, ENTH, VISC, MU (VISC and MU used only if KEYOPT (9) = 1.

Do not use ENTH with KEYOPT(8) = 1 or 2.

Surface Loads

Convection or Heat Flux (but not both) and Radiation (using Lab = RDSF) --

face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads

Heat Generations --

HG(I), HG(J), HG(K), HG(L)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Special Features

Birth and death

KEYOPT(1)

How to evaluate film coefficient:

0 --

Evaluate film coefficient (if any) at average film temperature, (TS + TB)/2

1 --

Evaluate at element surface temperature, TS

2 --

Evaluate at fluid bulk temperature, TB

3 --

Evaluate at differential temperature, |TS - TB|

KEYOPT(3)

Element behavior:

0 --

Plane

1 --

Axisymmetric

3 --

Plane with Z-depth, specified via real constant THK.

KEYOPT(4)

Element coordinate system:

0 --

Element coordinate system is parallel to the global coordinate system

1 --

Element coordinate system is based on the element I-J side.

KEYOPT(8)

Mass transport effects:

0 --

No mass transport effects

1 --

Mass transport with VX and VY

2 --

Same as 1 but also print mass transport heat flow

KEYOPT(9)

Nonlinear fluid flow option:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0 --
Standard heat transfer element

1 --
Nonlinear steady-state fluid flow analogy element (temperature degree of freedom interpreted as pressure)

PLANE55 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output as shown in Table 55.1: PLANE55 Element Output Definitions (p. 270)

For an axisymmetric analysis the face area and the heat flow rate are on a full 360° basis. Convection heat flux is positive out of the element; applied heat flux is positive into the element. If KEYOPT(9) = 1, the standard thermal output should be interpreted as the analogous fluid flow output. The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50) and of postprocessing data in Degenerated Shape Elements (p. 39). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 55.1: PLANE55 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	4
HGEN	Heat generations HG(I), HG(J), HG(K), HG(L)	Y	-
TG:X, Y, SUM	Thermal gradient components and vector sum at centroid	Y	Y
TF:X, Y, SUM	Thermal flux (heat flow rate/cross-sectional area) components and vector sum at centroid	Y	Y
FACE	Face label	1	-
AREA	Face area	1	1
NODES	Face nodes	1	1
HFILM	Film coefficient at each node of face	1	-
TBULK	Bulk temperature at each node of face	1	-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
TAVG	Average face temperature	1	1
HEAT RATE	Heat flow rate across face by convection	1	1
HFAVG	Average film coefficient of the face	-	1
TBAVG	Average face bulk temperature	-	1
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	1
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	1	-
HFLUX	Heat flux at each node of face	1	-
HEAT FLOW BY MASS TRANS-PORT	Heat flow rate across face by mass transport	2	-
PRESSURE GRAD	Total pressure gradient and its X and Y components	3	-
MASS FLUX	Mass flow rate per unit cross-sectional area	3	-
FLUID VELOCITY	Total fluid velocity and its X and Y components	3	-

1. If a surface load is input
2. If KEYOPT(8) = 2
3. If KEYOPT(9) = 1
4. Available only at centroid as a ***GET** item.

Table 55.2: PLANE55 Item and Sequence Numbers (p. 271) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) of this manual for more information. The following notation is used in Table 55.2: PLANE55 Item and Sequence Numbers (p. 271):

Name
output quantity as defined in the Table 55.1: PLANE55 Element Output Definitions (p. 270)

Item
predetermined Item label for **ETABLE** command

FC*n*
sequence number for solution items for element Face *n*

Table 55.2: PLANE55 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	FC1	FC2	FC3	FC4
AREA	NMISC	1	7	13	19
HFAVG	NMISC	2	8	14	20
TAVG	NMISC	3	9	15	21
TBAVG	NMISC	4	10	16	22

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	FC1	FC2	FC3	FC4
HEAT RATE	NMISC	5	11	17	23
HFLXAVG	NMISC	6	12	18	24

PLANE55 Assumptions and Restrictions

- The element must not have a negative or a zero area.
- The element must lie in an X-Y plane as shown in Figure 55.1: PLANE55 Geometry (p. 267) and the Y-axis must be the axis of symmetry for axisymmetric analyses. An axisymmetric structure should be modeled in the +X quadrants.
- A triangular element may be formed by defining duplicate K and L node numbers as described in Degenerated Shape Elements (p. 39).
- The specific heat and enthalpy are evaluated at each integration point to allow for abrupt changes (such as melting) within a coarse grid of elements.
- If the thermal element is to be replaced by a PLANE182 structural element with surface stresses requested, the thermal element should be oriented with face IJ or face KL as a free surface. A free surface of the element (that is, not adjacent to another element and not subjected to a boundary constraint) is assumed to be adiabatic.
- Thermal transients having a fine integration time step and a severe thermal gradient at the surface will also require a fine mesh at the surface.
- If KEYOPT(8) > 0, unsymmetric matrices are produced.
- When mass flow is activated (KEYOPT(8)=1 or 2), the element Peclet number should be less than 1:

  $Pe = \rho*v*L*Cp/(2*k) < 1.0$

  Where L is an element length scale based on the flow direction and element geometry. See PLANE55 in the *Mechanical APDL Theory Reference* for more details.

PLANE55 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- This element does not have the mass transport or fluid flow options. KEYOPT(8) and KEYOPT(9) can only be set to 0 (default).
- The VX and VY real constants are not applicable.
- The VISC and MU material properties are not applicable.
- The element does not have the birth and death feature.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SHELL61

Axisymmetric-Harmonic Structural Shell

MP ME ST <> <> <> <> <> PP EME MFS
Product Restrictions

SHELL61 Element Description

SHELL61 has four degrees of freedom at each node: translations in the nodal x, y, and z directions and a rotation about the nodal z-axis. The loading may be axisymmetric or nonaxisymmetric. Various loading cases are described in Harmonic Axisymmetric Elements with Nonaxisymmetric Loads (p. 84).

Extreme orientations of the conical shell element result in a cylindrical shell element or an annular disc element. The shell element may have a linearly varying thickness. See SHELL61 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 61.1: SHELL61 Geometry

SHELL61 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 61.1: SHELL61 Geometry (p. 273). The element is defined by two nodes, two end thicknesses, the number of harmonic waves (*MODE* on the **MODE** command), a symmetry condition (*ISYM* on the **MODE** command), and the orthotropic material properties. The element coordinate system is shown in Figure 61.2: SHELL61 Stress Output (p. 276). θ is in the tangential (hoop) direction. The *MODE* or *ISYM* parameters are discussed in detail in Harmonic Axisymmetric Elements with Nonaxisymmetric Loads (p. 84).

The material may be orthotropic, with nine elastic constants required for its description. The element loading may be input as any combination of harmonically varying temperatures and pressures. Harmonically varying nodal forces, if any, should be input on a full 360° basis.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The element may have variable thickness. The thickness is assumed to vary linearly between the nodes. If the element has a constant thickness, only TK(I) is required. Real constant ADMSUA is used to define an added mass per unit area.

Element loads are described in Nodal Loading (p. 49). Harmonically varying pressures may be input as surface loads on the element faces as shown by the circled numbers on Figure 61.1: SHELL61 Geometry (p. 273). Positive pressures act into the element. The pressures are applied at the surface of the element rather than at the centroidal plane so that some thickness effects can be considered. These include the increase or decrease in size of surface area the load is acting on and (in the case of a nonzero Poisson's ratio) an interaction effect causing the element to grow longer or shorter under equal pressures on both surfaces. Material properties EY, PRXY, and PRYZ (or EY, NUXY, and NUYZ) are required for this effect.

Harmonically varying temperatures may be input as element body loads at the four corner locations shown in Figure 61.1: SHELL61 Geometry (p. 273). The first corner temperature T1 defaults to TUNIF. If all other temperatures are unspecified, they default to T1. If only T1 and T2 are input, T3 defaults to T2 and T4 defaults to T1. For any other input pattern, unspecified temperatures default to TUNIF.

KEYOPT(1) is used for temperature loading with *MODE* greater than zero and temperature-dependent material properties. Material properties may only be evaluated at a constant (nonharmonically varying) temperature. If *MODE* equals zero, the material properties are always evaluated at the average element temperature. KEYOPT(3) is used to include or suppress the extra displacement shapes.

A summary of the element input is given in "SHELL61 Input Summary" (p. 274). A general description of element input is given in Element Input (p. 43).

SHELL61 Input Summary

Nodes

I, J

Degrees of Freedom

UX, UY, UZ, ROTZ

Real Constants

TK(I) - Shell thickness at node I
TK(J) - Shell thickness at node J (TK(J) defaults to TK(I))
ADMSUA - Added mass/unit area

Material Properties

MP command: EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), ALPX, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ), DENS, GXZ, ALPD, BETD. (X is meridional, Y is through-the-thickness, and Z is circumferential.)

Surface Loads

Pressures --

face 1 (I-J) (top, in -Y direction)
face 2 (I-J) (bottom, in +Y direction)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Body Loads

Temperatures --
T1, T2, T3, T4

Mode Number
Number of harmonic waves around the circumference (**MODE**)

Loading Condition
Symmetry condition (**MODE**)

Special Features
Stress stiffening

KEYOPT(1)
If MODE is greater than zero, use temperatures for:

0 --
Use temperatures only for thermal bending (evaluate material properties at TREF)

1 --
Use temperatures only for material property evaluation (thermal strains are not computed)

KEYOPT(3)
Extra displacement shapes:

0 --
Include extra displacement shapes

1 --
Suppress extra displacement shapes

KEYOPT(4)
Member force and moment output:

0 --
No printout of member forces and moments

1 --
Print out member forces and moments in the element coordinate system

KEYOPT(6)
Location of element solution output:

0 --
Output solution at mid-length only

***N* --**
Output solution at *N* equally spaced interior points and at end points (where *N* = 1, 3, 5, 7 or 9)

SHELL61 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Additional element output as shown in Table 61.1: SHELL61 Element Output Definitions (p. 276)

Several items are illustrated in Figure 61.2: SHELL61 Stress Output (p. 276). The printout may be displayed at the centroid, at the end points and at *N* equally spaced interior points, where *N* is the KEYOPT(6) value. For example, if *N* = 3, printout will be produced at end I, 1/4 length, mid-length (centroid), 3/4 length, and at end J. Printout location number 1 is always at end I. Stress components which are inherently zero are printed for clarity.

In the displacement printout, the UZ components are out-of-phase with the UX and UY components. For example, in the *MODE* = 1, *ISYM* = 1 loading case, UX and UY are the peak values at $\theta = 0°$ and UZ is the peak value at $\theta = 90°$. We recommend that you always use the *angle* field on the **SET** command when postprocessing the results. For more information about harmonic elements, see Harmonic Axisymmetric Elements with Nonaxisymmetric Loads (p. 84)

A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

Figure 61.2: SHELL61 Stress Output

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 61.1: SHELL61 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
MAT	Material number	Y	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
LENGTH	Distance between node I and node J	Y	Y
XC, YC	Location where results are reported	Y	2
TEMP	Temperatures T1, T2, T3, T4	Y	Y
PRES	Pressures P1 (top) at nodes I,J; P2 (bottom) at nodes I,J	Y	Y
MODE	Number of waves in loading	Y	Y
ISYM	Loading key: 1 = symmetric, -1 = antisymmetric	Y	Y
T(X, Z, XZ)	In-plane element X, Z, and XZ forces at KEYOPT(6) location(s)	Y	Y
M(X, Z, XZ)	Out-of-plane element X, Z, and XZ moments at KEYOPT(6) location(s)	Y	Y
MFOR(X, Y, Z), MMOMZ	Member forces and member moment for each node in the element coordinate system	1	Y
PK ANG	Angle where stresses have peak values: 0 and 90/MODE°. Blank if *MODE* = 0.	Y	Y
S(M, THK, H, MH)	Stresses (meridional, through-thickness, hoop, meridional-hoop) at PK ANG locations, repeated for top, middle, and bottom of shell	Y	Y
EPEL(M, THK, H, MH)	Elastic strains (meridional, through-thickness, hoop, meridional-hoop) at PK ANG locations, repeated for top, middle, and bottom of shell	Y	Y
EPTH(M, THK, H, MH)	Thermal strains (meridional, through-thickness, hoop, meridional-hoop) at PK ANG locations, repeated for top, middle, and bottom of shell	Y	Y

1. These items are printed only if KEYOPT(4) = 1.

2. Available only at centroid as a ***GET** item.

Table 61.2: SHELL61 Item and Sequence Numbers (KEYOPT(6) = 0 or 1) (p. 278) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 61.2: SHELL61 Item and Sequence Numbers (KEYOPT(6) = 0 or 1) (p. 278):

Name
output quantity as defined in the Table 61.1: SHELL61 Element Output Definitions (p. 276)

Item
predetermined Item label for **ETABLE** command

I,J
sequence number for data at nodes I and J

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

IL*n*

sequence number for data at Intermediate Location *n*

Table 61.2: SHELL61 Item and Sequence Numbers (KEYOPT(6) = 0 or 1)

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	I	IL1	J
Top				
SM	LS	1	13	25
STHK	LS	2	14	26
SH	LS	3	15	27
SMH	LS	4	16	28
EPELM	LEPEL	1	13	25
EPELTHK	LEPEL	2	14	26
EPELH	LEPEL	3	15	27
EPELMH	LEPEL	4	16	28
EPTHM	LEPTH	1	13	25
EPTH-THK	LEPTH	2	14	26
EPTHH	LEPTH	3	15	27
EPTHMH	LEPTH	4	16	28
Mid				
SM	LS	5	17	29
STHK	LS	6	18	30
SH	LS	7	19	31
SMH	LS	8	20	32
EPELM	LEPEL	5	17	29
EPELTHK	LEPEL	6	18	30
EPELH	LEPEL	7	19	31
EPELMH	LEPEL	8	20	32
EPTHM	LEPTH	5	17	29
EPTH-THK	LEPTH	6	18	30
EPTHH	LEPTH	7	19	31
EPTHMH	LEPTH	8	20	32
Bot				
SM	LS	9	21	33
STHK	LS	10	22	34
SH	LS	11	23	35
SMH	LS	12	24	36

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	I	IL1	J
EPELM	LEPEL	9	21	33
EPELTHK	LEPEL	10	22	34
EPELH	LEPEL	11	23	35
EPELMH	LEPEL	12	24	36
EPTHM	LEPTH	9	21	33
EPTH-THK	LEPTH	10	22	34
EPTHH	LEPTH	11	23	35
EPTHMH	LEPTH	12	24	36
Element				
MFORX	SMISC	1	-	7
MFORY	SMISC	2	-	8
MFORZ	SMISC	3	-	9
MMOMZ	SMISC	6	-	12
TX	SMISC	13	19	25
TZ	SMISC	14	20	26
TXZ	SMISC	15	21	27
MX	SMISC	16	22	28
MZ	SMISC	17	23	29
MXZ	SMISC	18	24	30
P1	SMISC	31	-	32
P2	SMISC	35	-	36

		Corner Location			
		1	**2**	**3**	**4**
TEMP	LBFE	1	2	3	4

Table 61.3: SHELL61 Item and Sequence Numbers (KEYOPT(6) = 3)

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	I	IL1	IL2	IL3	J
Top						
SM	LS	1	13	25	37	49
STHK	LS	2	14	26	38	50
SH	LS	3	15	27	39	51
SMH	LS	4	16	28	40	52
EPELM	LEPEL	1	13	25	37	49

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	I	IL1	IL2	IL3	J
EPELTHK	LEPEL	2	14	26	38	50
EPELH	LEPEL	3	15	27	39	51
EPELMH	LEPEL	4	16	28	40	52
EPTHM	LEPTH	1	13	25	37	49
EPTH-THK	LEPTH	2	14	26	38	50
EPTHH	LEPTH	3	15	27	39	51
EPTHMH	LEPTH	4	16	28	40	52
Mid						
SM	LS	5	17	29	41	53
STHK	LS	6	18	30	42	54
SH	LS	7	19	31	43	55
SMH	LS	8	20	32	44	56
EPELM	LEPEL	5	17	29	41	53
EPELTHK	LEPEL	6	18	30	42	54
EPELH	LEPEL	7	19	31	43	55
EPELMH	LEPEL	8	20	32	44	56
EPTHM	LEPTH	5	17	29	41	53
EPTH-THK	LEPTH	6	18	30	42	54
EPTHH	LEPTH	7	19	31	43	55
EPTHMH	LEPTH	8	20	32	44	56
Bot						
SM	LS	9	21	33	45	57
STHK	LS	10	22	34	46	58
SH	LS	11	23	35	47	59
SMH	LS	12	24	36	48	60
EPELM	LEPEL	9	21	33	45	57
EPELTHK	LEPEL	10	22	34	46	58
EPELH	LEPEL	11	23	35	47	59
EPELMH	LEPEL	12	24	36	48	60
EPTHM	LEPTH	9	21	33	45	57
EPTH-THK	LEPTH	10	22	34	46	58
EPTHH	LEPTH	11	23	35	47	59
EPTHMH	LEPTH	12	24	36	48	60
Element						

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	I	IL1	IL2	IL3	J
MFORX	SMISC	1	-	-	-	7
MFORY	SMISC	2	-	-	-	8
MFORZ	SMISC	3	-	-	-	9
MMOMZ	SMISC	6	-	-	-	12
TX	SMISC	13	19	25	31	37
TZ	SMISC	14	20	26	32	38
TXZ	SMISC	15	21	27	33	39
MX	SMISC	16	22	28	34	40
MZ	SMISC	17	23	29	35	41
MXZ	SMISC	18	24	30	36	42
P1	SMISC	43	-	-	-	44
P2	SMISC	47	-	-	-	48

		Corner Location			
		1	2	3	4
TEMP	LBFE	1	2	3	4

Table 61.4: SHELL61 Item and Sequence Numbers (KEYOPT(6) = 5)

Output Quantity Name	ETABLE and ESOL Command Input							
	Item	I	IL1	IL2	IL3	IL4	IL5	J
Top								
SM	LS	1	13	25	37	49	61	73
STHK	LS	2	14	26	38	50	62	74
SH	LS	3	15	27	39	51	63	75
SMH	LS	4	16	28	40	52	64	76
EPELM	LEPEL	1	13	25	37	49	61	73
EPELTHK	LEPEL	2	14	26	38	50	62	74
EPELH	LEPEL	3	15	27	39	51	63	75
EPELMH	LEPEL	4	16	28	40	52	64	76
EPTHM	LEPTH	1	13	25	37	49	61	73
EPTH-THK	LEPTH	2	14	26	38	50	62	74
EPTHH	LEPTH	3	15	27	39	51	63	75
EPTHMH	LEPTH	4	16	28	40	52	64	76
Mid								
SM	LS	5	17	29	41	53	65	77

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input							
	Item	I	IL1	IL2	IL3	IL4	IL5	J
STHK	LS	6	18	30	42	54	66	78
SH	LS	7	19	31	43	55	67	79
SMH	LS	8	20	32	44	56	68	80
EPELM	LEPEL	5	17	29	41	53	65	77
EPELTHK	LEPEL	6	18	30	42	54	66	78
EPELH	LEPEL	7	19	31	43	55	67	79
EPELMH	LEPEL	8	20	32	44	56	68	80
EPTHM	LEPTH	5	17	29	41	53	65	77
EPTH-THK	LEPTH	6	18	30	42	54	66	78
EPTHH	LEPTH	7	19	31	43	55	67	79
EPTHMH	LEPTH	8	20	32	44	56	68	80
Bot								
SM	LS	9	21	33	45	57	69	81
STHK	LS	10	22	34	46	58	70	82
SH	LS	11	23	35	47	59	71	83
SMH	LS	12	24	36	48	60	72	84
EPELM	LEPEL	9	21	33	45	57	69	81
EPELTHK	LEPEL	10	22	34	46	58	70	82
EPELH	LEPEL	11	23	35	47	59	71	83
EPELMH	LEPEL	12	24	36	48	60	72	84
EPTHM	LEPTH	9	21	33	45	57	69	81
EPTH-THK	LEPTH	10	22	34	46	58	70	82
EPTHH	LEPTH	11	23	35	47	59	71	83
EPTHMH	LEPTH	12	24	36	48	60	72	84
Element								
MFORX	SMISC	1	-	-	-	-	-	7
MFORY	SMISC	2	-	-	-	-	-	8
MFORZ	SMISC	3	-	-	-	-	-	9
MMOMZ	SMISC	6	-	-	-	-	-	12
TX	SMISC	13	19	25	31	37	43	49
TZ	SMISC	14	20	26	32	38	44	50
TXZ	SMISC	15	21	27	33	39	45	51
MX	SMISC	16	22	28	34	40	46	52
MZ	SMISC	17	23	29	35	41	47	53
MXZ	SMISC	18	24	30	36	42	48	54

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input							
	Item	I	IL1	IL2	IL3	IL4	IL5	J
P1	SMISC	55	-	-	-	-	-	56
P2	SMISC	59	-	-	-	-	-	60

		Corner Location			
		1	2	3	4
TEMP	LBFE	1	2	3	4

Table 61.5: SHELL61 Item and Sequence Numbers (KEYOPT(6) = 7)

Output Quantity Name	ETABLE and ESOL Command Input									
	Item	I	IL1	IL2	IL3	IL4	IL5	IL6	IL7	J
Top										
SM	LS	1	13	25	37	49	61	73	85	97
STHK	LS	2	14	26	38	50	62	74	86	98
SH	LS	3	15	27	39	51	63	75	87	99
SMH	LS	4	16	28	40	52	64	76	88	100
EPELM	LEPEL	1	13	25	37	49	61	73	85	97
EPELTHK	LEPEL	2	14	26	38	50	62	74	86	98
EPELH	LEPEL	3	15	27	39	51	63	75	87	99
EPELMH	LEPEL	4	16	28	40	52	64	76	88	100
EPTHM	LEPTH	1	13	25	37	49	61	73	85	97
EPTH-THK	LEPTH	2	14	26	38	50	62	74	86	98
EPTHH	LEPTH	3	15	27	39	51	63	75	87	99
EPTHMH	LEPTH	4	16	28	40	52	64	76	88	100
Mid										
SM	LS	5	17	29	41	53	65	77	89	101
STHK	LS	6	18	30	42	54	66	78	90	102
SH	LS	7	19	31	43	55	67	79	91	103
SMH	LS	8	20	32	44	56	68	80	92	104
EPELM	LEPEL	5	17	29	41	53	65	77	89	101
EPELTHK	LEPEL	6	18	30	42	54	66	78	90	102
EPELH	LEPEL	7	19	31	43	55	67	79	91	103
EPELMH	LEPEL	8	20	32	44	56	68	80	92	104
EPTHM	LEPTH	5	17	29	41	53	65	77	89	101
EPTH-THK	LEPTH	6	18	30	42	54	66	78	90	102

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quant-ity Name	ETABLE and ESOL Command Input									
	Item	I	IL1	IL2	IL3	IL4	IL5	IL6	IL7	J
EPTHH	LEPTH	7	19	31	43	55	67	79	91	103
EPTHMH	LEPTH	8	20	32	44	56	68	80	92	104
Bot										
SM	LS	9	21	33	45	57	69	81	93	105
STHK	LS	10	22	34	46	58	70	82	94	106
SH	LS	11	23	35	47	59	71	83	95	107
SMH	LS	12	24	36	48	60	72	84	96	108
EPELM	LEPEL	9	21	33	45	57	69	81	93	105
EPELTHK	LEPEL	10	22	34	46	58	70	82	94	106
EPELH	LEPEL	11	23	35	47	59	71	83	95	107
EPELMH	LEPEL	12	24	36	48	60	72	84	96	108
EPTHM	LEPTH	9	21	33	45	57	69	81	93	105
EPTH-THK	LEPTH	10	22	34	46	58	70	82	94	106
EPTHH	LEPTH	11	23	35	47	59	71	83	95	107
EPTHMH	LEPTH	12	24	36	48	60	72	84	96	108
Element										
MFORX	SMISC	1	-	-	-	-	-	-	-	7
MFORY	SMISC	2	-	-	-	-	-	-	-	8
MFORZ	SMISC	3	-	-	-	-	-	-	-	9
MMOMZ	SMISC	6	-	-	-	-	-	-	-	12
TX	SMISC	13	19	25	31	37	43	49	55	61
TZ	SMISC	14	20	26	32	38	44	50	56	62
TXZ	SMISC	15	21	27	33	39	45	51	57	63
MX	SMISC	16	22	28	34	40	46	52	58	64
MZ	SMISC	17	23	29	35	41	47	53	59	65
MXZ	SMISC	18	24	30	36	42	48	54	60	66
P1	SMISC	67	-	-	-	-	-	-	-	68
P2	SMISC	71	-	-	-	-	-	-	-	72

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

		Corner Location			
		1	**2**	**3**	**4**
TEMP	LBFE	1	2	3	4

Table 61.6: SHELL61 Item and Sequence Numbers (KEYOPT(6) = 9)

Output Quantity Label	ETABLE and ESOL Command Input											
	Item	**I**	**IL1**	**IL2**	**IL3**	**IL4**	**IL5**	**IL6**	**IL7**	**IL8**	**IL9**	**J**
Top												
SM	LS	1	13	25	37	49	61	73	85	97	109	121
STHK	LS	2	14	26	38	50	62	74	86	98	110	122
SH	LS	3	15	27	39	51	63	75	87	99	111	123
SMH	LS	4	16	28	40	52	64	76	88	100	112	124
EPELM	LEPEL	1	13	25	37	49	61	73	85	97	109	121
EPELTHK	LEPEL	2	14	26	38	50	62	74	86	98	110	122
EPELH	LEPEL	3	15	27	39	51	63	75	87	99	111	123
EPELMH	LEPEL	4	16	28	40	52	64	76	88	100	112	124
EPTHM	LEPTH	1	13	25	37	49	61	73	85	97	109	121
EPTH-THK	LEPTH	2	14	26	38	50	62	74	86	98	110	122
EPTHH	LEPTH	3	15	27	39	51	63	75	87	99	111	123
EPTHMH	LEPTH	4	16	28	40	52	64	76	88	100	112	124
Mid												
SM	LS	5	17	29	41	53	65	77	89	101	113	125
STHK	LS	6	18	30	42	54	66	78	90	102	114	126
SH	LS	7	19	31	43	55	67	79	91	103	115	127
SMH	LS	8	20	32	44	56	68	80	92	104	116	128
EPELM	LEPEL	5	17	29	41	53	65	77	89	101	113	125
EPELTHK	LEPEL	6	18	30	42	54	66	78	90	102	114	126
EPELH	LEPEL	7	19	31	43	55	67	79	91	103	115	127
EPELMH	LEPEL	8	20	32	44	56	68	80	92	104	116	128
EPTHM	LEPTH	5	17	29	41	53	65	77	89	101	113	125
EPTH-THK	LEPTH	6	18	30	42	54	66	78	90	102	114	126
EPTHH	LEPTH	7	19	31	43	55	67	79	91	103	115	127
EPTHMH	LEPTH	8	20	32	44	56	68	80	92	104	116	128
Bot												
SM	LS	9	21	33	45	57	69	81	93	105	117	129
STHK	LS	10	22	34	46	58	70	82	94	106	118	130
SH	LS	11	23	35	47	59	71	83	95	107	119	131

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Label	ETABLE and ESOL Command Input											
	Item	I	IL1	IL2	IL3	IL4	IL5	IL6	IL7	IL8	IL9	J
SMH	LS	12	24	36	48	60	72	84	96	108	120	132
EPELM	LEPEL	9	21	33	45	57	69	81	93	105	117	129
EPELTHK	LEPEL	10	22	34	46	58	70	82	94	106	118	130
EPELH	LEPEL	11	23	35	47	59	71	83	95	107	119	131
EPELMH	LEPEL	12	24	36	48	60	72	84	96	108	120	132
EPTHM	LEPTH	9	21	33	45	57	69	81	93	105	117	129
EPTH-THK	LEPTH	10	22	34	46	58	70	82	94	106	118	130
EPTHH	LEPTH	11	23	35	47	59	71	83	95	107	119	131
EPTHMH	LEPTH	12	24	36	48	60	72	84	96	108	120	132
Element												
MFORX	SMISC	1	-	-	-	-	-	-	-	-	-	7
MFORY	SMISC	2	-	-	-	-	-	-	-	-	-	8
MFORZ	SMISC	3	-	-	-	-	-	-	-	-	-	9
MMOMZ	SMISC	6	-	-	-	-	-	-	-	-	-	12
TX	SMISC	13	19	25	31	37	43	49	55	61	67	73
TZ	SMISC	14	20	26	32	38	44	50	56	62	68	74
TXZ	SMISC	15	21	27	33	39	45	51	57	63	69	75
MX	SMISC	16	22	28	34	40	46	52	58	64	70	76
MZ	SMISC	17	23	29	35	41	47	53	59	65	71	77
MXZ	SMISC	18	24	30	36	42	48	54	60	66	72	78
P1	SMISC	79	-	-	-	-	-	-	-	-	-	80
P2	SMISC	83	-	-	-	-	-	-	-	-	-	84

		Corner Location			
		1	**2**	**3**	**4**
TEMP	LBFE	1	2	3	4

SHELL61 Assumptions and Restrictions

- The axisymmetric shell element must be defined in the global X-Y plane and must not have a zero length. Both ends must have nonnegative X coordinate values and the element must not lie along the global Y-axis.
- If the element has a constant thickness, only TK(I) need be defined. TK(I) must not be zero.
- The element thickness is assumed to vary linearly from node I to node J. Some thick shell effects have been included in the formulation of SHELL61 but it cannot be properly considered to be a thick shell element. If these effects are important, it is recommended to use PLANE25.
- The element assumes a linear elastic material.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Post analysis superposition of results is valid only with other linear elastic solutions.
- Strain energy does not consider thermal effects.
- The element should not be used with the large deflection option.
- The element may not be deactivated with the **EKILL** command.
- You can use only axisymmetric (**MODE**,0) loads without significant torsional stresses to generate the stress state used for stress stiffened modal analyses using this element.

SHELL61 Product Restrictions

There are no product restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID65

3-D Reinforced Concrete Solid

MP ME ST <> <> <> <> <> PP EME MFS
Product Restrictions

SOLID65 Element Description

Although this legacy element is available for use in your analysis, ANSYS, Inc. recommends using the current-technology element SOLID185, specifying either full integration with the $\bar{B}$ method (KEYOPT(3) = 0 and KEYOPT(2) = 3), or simplified enhanced strain formulation (KEYOPT(3) = 0 and KEYOPT(2) = 0). For more information, see Microplane in the *Material Reference*.

SOLID65 is used for the 3-D modeling of solids with or without reinforcing bars (rebar). The solid is capable of cracking in tension and crushing in compression. In concrete applications, for example, the solid capability of the element may be used to model the concrete while the rebar capability is available for modeling reinforcement behavior. Other cases for which the element is also applicable would be reinforced composites (such as fiberglass), and geological materials (such as rock). The element is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. Up to three different rebar specifications may be defined.

The concrete element is similar to a 3-D structural solid but with the addition of special cracking and crushing capabilities. The most important aspect of this element is the treatment of nonlinear material properties. The concrete is capable of cracking (in three orthogonal directions), crushing, plastic deformation, and creep. The rebar are capable of tension and compression, but not shear. They are also capable of plastic deformation and creep. See SOLID65 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 65.1: SOLID65 Geometry

SOLID65 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 65.1: SOLID65 Geometry (p. 289). The element is defined by eight nodes and the isotropic material properties. The element has one solid material and up to three rebar materials. Use the **MAT** command to input the concrete material properties. Rebar specifications, which are input as real constants, include the material number (MAT), the volume ratio (VR), and the orientation angles (THETA, PHI). The rebar orientations can be graphically verified with the **/ESHAPE** command.

The volume ratio is defined as the rebar volume divided by the total element volume. The orientation is defined by two angles (in degrees) from the element coordinate system. The element coordinate system orientation is as described in Coordinate Systems (p. 55). A rebar material number of zero or equal to the element material number removes that rebar capability.

Additional concrete material data, such as the shear transfer coefficients, tensile stresses, and compressive stresses are input in the data table, for convenience, as described in Table 65.1: SOLID65 Concrete Material Data (p. 293). Typical shear transfer coefficients range from 0.0 to 1.0, with 0.0 representing a smooth crack (complete loss of shear transfer) and 1.0 representing a rough crack (no loss of shear transfer). This specification may be made for both the closed and open crack. When the element is cracked or crushed, a small amount of stiffness is added to the element for numerical stability. The stiffness multiplier CSTIF is used across a cracked face or for a crushed element, and defaults to 1.0E-6.

Element loads are described in Nodal Loading (p. 49). Pressures may be input as surface loads on the element faces as shown by the circled numbers on Figure 65.1: SOLID65 Geometry (p. 289). Positive pressures act into the element. Temperatures and fluences may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF. Similar defaults occurs for fluence except that zero is used instead of TUNIF.

Use the **BETAD** command to specify the global value of damping. If **MP**,BETD is defined for the material number of the element (assigned with the **MAT** command), it is summed with the value from the **BETAD** command. Similarly, use the **TREF** command to supply the global value of reference temperature. If **MP**,REFT is defined for the material number of the element, it is used for the element instead of the value from the **TREF** command. But if **MP**,REFT is defined for the material number of the rebar, it is used instead of either the global or element value.

KEYOPT(1) is used to include or suppress the extra displacement shapes. KEYOPT(5) and KEYOPT(6) provide various element printout options (see Element Solution (p. 51)).

The stress relaxation associated with KEYOPT(7) = 1 is used only to help accelerate convergence of the calculations when cracking is imminent. (A multiplier for the amount of tensile stress relaxation can be input as constant C9 in the data table; see Table 65.1: SOLID65 Concrete Material Data (p. 293)) The relaxation does not represent a revised stress-strain relationship for post-cracking behavior. After the solution converges to the cracked state, the modulus normal to the crack face is set to zero. Thus, the stiffness is zero normal to the crack face. See the *Mechanical APDL Theory Reference* for details.

The program warns when each unreinforced element crushes at all integration points. If this warning is unwanted, it can be suppressed with KEYOPT(8) = 1.

If solution convergence is a problem, it is recommended to set KEYOPT(3) = 2 and apply the load in very small load increments.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can include the effects of pressure load stiffness in a geometric nonlinear analysis using **SOLCONTROL**,,,INCP. Pressure load stiffness effects are included in linear eigenvalue buckling automatically. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

A summary of the element input is given in "SOLID65 Input Summary" (p. 291). A general description of element input is given in Element Input (p. 43).

SOLID65 Input Summary

Nodes

I, J, K, L, M, N, O, P

Degrees of Freedom

UX, UY, UZ

Real Constants

MAT1, VR1, THETA1, PHI1, MAT2, VR2,
THETA2, PHI2, MAT3, VR3, THETA3, PHI3, CSTIF

(where MAT*n* is material number, VR*n* is volume ratio, and THETA*n* and PHI*n* are orientation angles for up to 3 rebar materials)

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, ALPX (or CTEX *or* THSX), PRXY or NUXY, DENS (for concrete),
EX, ALPX (or CTEX *or* THSX), DENS (for each rebar), ALPD
Specify BETD only once for the element. (Issue the **MAT** command to assign the material property set.) REFT may be specified once for the element, or may be assigned on a per-rebar basis. See the discussion in "SOLID65 Input Data" (p. 290) for more information.

Surface Loads

Pressures --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperatures --

T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

Fluences --

FL(I), FL(J), FL(K), FL(L), FL(M), FL(N), FL(O), FL(P)

Special Features

Adaptive descent
Birth and death
Creep
Large deflection
Large strain
Stress stiffening

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(1)

Extra displacement shapes:

0 --

Include extra displacement shapes

1 --

Suppress extra displacement shapes

KEYOPT(3)

Behavior of totally crushed unreinforced elements:

0 --

Base

1 --

Suppress mass and applied loads, and warning message (see KEYOPT(8))

2 --

Features of 1 and apply consistent Newton-Raphson load vector.

KEYOPT(5)

Concrete linear solution output:

0 --

Print concrete linear solution only at centroid

1 --

Repeat solution at each integration point

2 --

Nodal stress printout

KEYOPT(6)

Concrete nonlinear solution output:

0 --

Print concrete nonlinear solution only at centroid

3 --

Print solution also at each integration point

KEYOPT(7)

Stress relaxation after cracking:

0 --

No tensile stress relaxation after cracking

1 --

Include tensile stress relaxation after cracking to help convergence

KEYOPT(8)

Warning message for totally crushed unreinforced element:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0 --
Print the warning

1 --
Suppress the warning

SOLID65 Concrete Information

The data listed in Table 65.1: SOLID65 Concrete Material Data (p. 293) is entered in the data table with the **TB** commands. Data not input are assumed to be zero, except for defaults described below. The constant table is started by using the **TB**command (with *Lab* = CONCR). Up to eight constants may be defined with the **TBDATA** commands following a temperature definition on the **TBTEMP** command. Up to six temperatures (*NTEMP* = 6 maximum on the **TB** command) may be defined with the **TBTEMP** commands. The constants (C1-C9) entered on the **TBDATA** commands (6 per command), after each **TBTEMP** command, are:

Table 65.1: SOLID65 Concrete Material Data

Constant	Meaning
1	Shear transfer coefficients for an open crack.
2	Shear transfer coefficients for a closed crack.
3	Uniaxial tensile cracking stress.
4	Uniaxial crushing stress (positive).
5	Biaxial crushing stress (positive).
6	Ambient hydrostatic stress state for use with constants 7 and 8.
7	Biaxial crushing stress (positive) under the ambient hydrostatic stress state (constant 6).
8	Uniaxial crushing stress (positive) under the ambient hydrostatic stress state (constant 6).
9	Stiffness multiplier for cracked tensile condition, used if KEYOPT(7) = 1 (defaults to 0.6).

Absence of the data table removes the cracking and crushing capability. A value of -1 for constant 3 or 4 also removes the cracking or crushing capability, respectively. If constants 1-4 are input and constants 5-8 are omitted, the latter constants default as discussed in the *Mechanical APDL Theory Reference*. If any one of Constants 5-8 are input, there are no defaults and all 8 constants must be input.

SOLID65 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 65.2: SOLID65 Element Output Definitions (p. 294)

Several items are illustrated in Figure 65.2: SOLID65 Stress Output (p. 294). The element stress directions are parallel to the element coordinate system. Nonlinear material printout appears only if nonlinear properties are specified. Rebar printout appears only for the rebar defined. If cracking or crushing is possible, printout for the concrete is also at the integration points, since cracking or crushing may occur at any integration point. The **PLCRACK** command can be used in POST1 to display the status of the

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

integration points. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

Figure 65.2: SOLID65 Stress Output

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 65.2: SOLID65 Element Output Definitions

Name	Definition	O	R
EL	Element number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
NREINF	Number of rebar	Y	-
VOLU:	Volume	Y	Y
PRES	Pressures P1 at nodes J, I, L, K; P2 at I, J, N, M; P3 at J, K, O, N; P4 at K, L, P, O; P5 at L, I, M, P; P6 at M, N, O, P	Y	Y
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	Y	Y
FLUEN	Fluences FL(I), FL(J), FL(K), FL(L), FL(M), FL(N), FL(O), FL(P)	Y	Y
XC, YC, ZC	Location where results are reported	Y	6
S:X, Y, Z, XY, YZ, XZ	Stresses	1	1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
S:1, 2, 3	Principal stresses	1	1
S:INT	Stress intensity	1	1
S:EQV	Equivalent stress	1	1
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	1	1
EPEL:1, 2, 3	Principal elastic strains	1	-
EPEL:EQV	Equivalent elastic strains [7]	1	1
EPTH:X, Y, Z, XY, YZ, XZ	Average thermal strains	1	1
EPTH:EQV	Equivalent thermal strains [7]	1	1
EPPL:X, Y, Z, XY, YZ, XZ	Average plastic strains	4	4
EPPL:EQV	Equivalent plastic strains [7]	4	4
EPCR:X, Y, Z, XY, YZ, XZ	Average creep strains	4	4
EPCR:EQV	Equivalent creep strains [7]	4	4
NL:EPEQ	Average equivalent plastic strain	4	4
NL:SRAT	Ratio of trial stress to stress on yield surface	4	4
NL:SEPL	Average equivalent stress from stress-strain curve	4	4
NL:HPRES	Hydrostatic pressure	-	4
THETCR, PHICR	THETA and PHI angle orientations of the normal to the crack plane	1	1
STATUS	Element status	2	2
IRF	Rebar number	3	-
MAT	Material number	3	-
VR	Volume ratio	3	-
THETA	Angle of orientation in X-Y plane	3	-
PHI	Angle of orientation out of X-Y plane	3	-
EPEL	Uniaxial elastic strain	3	-
S	Uniaxial stress	3	-
EPEL	Average uniaxial elastic strain	5	5
EPPL	Average uniaxial plastic strain	5	5
SEPL	Average equivalent stress from stress-strain curve	5	5
EPCR	Average uniaxial creep strain	5	5

1. Concrete solution item (output for each integration point (if KEYOPT(5) = 1) and the centroid)

2. The element status table (Table 65.4: SOLID65 Element Status Table (p. 296)) uses the following terms:

   - Crushed - solid is crushed.

   - Open - solid is cracked and the crack is open.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Closed - solid is cracked but the crack is closed.
- Neither - solid is neither crushed nor cracked.

3. Rebar solution item repeats for each rebar
4. Concrete nonlinear integration point solution (if KEYOPT(6) = 3 and the element has a nonlinear material)
5. Rebar nonlinear integration point solution (if KEYOPT(6) = 3 and the rebar has a nonlinear material)
6. Available only at centroid as a ***GET** item.
7. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.

Table 65.3: SOLID65 Miscellaneous Element Output

Description	Names of Items Output	O	R
Nodal Stress Solution	TEMP, S(X, Y, Z, XY, YZ, XZ), SINT, SEQV	1	-

1. Output at each node, if KEYOPT(5) = 2

Table 65.4: SOLID65 Element Status Table

Status	Status in Direction 1	Status in Direction 2	Status in Direction 3
1	Crushed	Crushed	Crushed
2	Open	Neither	Neither
3	Closed	Neither	Neither
4	Open	Open	Neither
5	Open	Open	Open
6	Closed	Open	Open
7	Closed	Open	Neither
8	Open	Closed	Open
9	Closed	Closed	Open
10	Open	Closed	Neither
11	Open	Open	Closed
12	Closed	Open	Closed
13	Closed	Closed	Neither
14	Open	Closed	Closed
15	Closed	Closed	Closed
16	Neither	Neither	Neither

Table 65.5: SOLID65 Item and Sequence Numbers (p. 297) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 65.5: SOLID65 Item and Sequence Numbers (p. 297):

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name
output quantity as defined in the Table 65.2: SOLID65 Element Output Definitions (p. 294)

Item
predetermined Item label for **ETABLE** command

I,J,...,P
sequence number for data at nodes I,J,...,P

IP
sequence number for Integration Point solution items

Table 65.5: SOLID65 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	Rebar 1	Rebar 2	Rebar 3
EPEL	SMISC	1	3	5
SIG	SMISC	2	4	6
EPPL	NMISC	41	45	49
EPCR	NMISC	42	46	50
SEPL	NMISC	43	47	51
SRAT	NMISC	44	48	52

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	I	J	K	L	M	N	O	P
P1	SMISC	8	7	10	9	-	-	-	-
P2	SMISC	11	12	-	-	14	13	-	-
P3	SMISC	-	15	16	-	-	18	17	-
P4	SMISC	-	-	19	20	-	-	22	21
P5	SMISC	24	-	-	23	25	-	-	26
P6	SMISC	-	-	-	-	27	28	29	30
S:1	NMISC	1	6	11	16	21	26	31	36
S:2	NMISC	2	7	12	17	22	27	32	37
S:3	NMISC	3	8	13	18	23	28	33	38
S:INT	NMISC	4	9	14	19	24	29	34	39
S:EQV	NMISC	5	10	15	20	25	30	35	40
FLUEN	NMISC	109	110	111	112	113	114	115	116

	Output Quantity Name	ETABLE and ESOL Command Input								
		Item	Integration Point							
			1	2	3	4	5	6	7	8
	STATUS	NMISC	53	60	67	74	81	88	95	102

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Output Quantity Name	ETABLE and ESOL Command Input								
		Item	Integration Point							
			1	2	3	4	5	6	7	8
Dir 1	THETCR	NMISC	54	61	68	75	82	89	96	103
	PHICR	NMISC	55	62	69	76	83	90	97	104
Dir 2	THETCR	NMISC	56	63	70	77	84	91	98	105
	PHICR	NMISC	57	64	71	78	85	92	99	106
Dir 3	THETCR	NMISC	58	65	72	79	86	93	100	107
	PHICR	NMISC	59	66	73	80	87	94	101	108

SOLID65 Assumptions and Restrictions

- Zero volume elements are not allowed.
- Elements may be numbered either as shown in Figure 65.1: SOLID65 Geometry (p. 289) or may have the planes IJKL and MNOP interchanged. Also, the element may not be twisted such that the element has two separate volumes. This occurs most frequently when the elements are not numbered properly.
- All elements must have eight nodes.
- A prism-shaped element may be formed by defining duplicate K and L and duplicate O and P node numbers (see Degenerated Shape Elements (p. 39)). A tetrahedron shape is also available. The extra shapes are automatically deleted for tetrahedron elements.
- Whenever the rebar capability of the element is used, the rebar are assumed to be "smeared" throughout the element. The sum of the volume ratios for all rebar must not be greater than 1.0.
- The element is nonlinear and requires an iterative solution.
- When both cracking and crushing are used together, care must be taken to apply the load slowly to prevent possible fictitious crushing of the concrete before proper load transfer can occur through a closed crack. This usually happens when excessive cracking strains are coupled to the orthogonal uncracked directions through Poisson's effect. Also, at those integration points where crushing has occurred, the output plastic and creep strains are from the previous converged substep. Furthermore, when cracking has occurred, the elastic strain output includes the cracking strain. The lost shear resistance of cracked and/or crushed elements cannot be transferred to the rebar, which have no shear stiffness.
- The following two options are not recommended if cracking or crushing nonlinearities are present:
  - Stress-stiffening effects.
  - Large strain and large deflection. Results may not converge or may be incorrect, especially if significantly large rotation is involved.

SOLID65 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LINK68

Coupled Thermal-Electric Line

MP ME <> PR PRN <> <> <> PP EME MFS
Product Restrictions

LINK68 Element Description

LINK68 is a uniaxial element in 3-D space with the ability to conduct heat and electrical current between its nodes. Joule heat generated by the current flow is also included in the heat balance. The element has two degrees of freedom, temperature and voltage, at each node. The thermal-electrical line element may be used in a steady-state or transient thermal analysis, although no transient electrical capacitance or inductance effects are included in the element.

The element is linear but requires an iterative solution to include the Joule heating effect in the thermal solution. If no electrical effects are present, the conducting bar element (LINK33) may be used. If the model containing the thermal-electrical element is also to be analyzed structurally, the element should be replaced by an equivalent structural element. See LINK68 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 68.1: LINK68 Geometry

LINK68 Input Data

The geometry, node locations, and the coordinate system for this thermal-electrical line element are shown in Figure 68.1: LINK68 Geometry (p. 299). The element is defined by two nodes, the cross-sectional area, and the material properties. In an axisymmetric analysis the area should be input on a full 360° basis. The thermal conductivity and electrical resistivity are in the element longitudinal direction. The specific heat and density may be assigned any values for steady-state solutions.

The electrical material property, RSVX, is the resistivity of the material. The resistance of the element is calculated from RSVX*length/AREA. The resistivity, like any other material property, may be input as a function of temperature. Properties not input default as described in Linear Material Properties in the *Material Reference*.

The word VOLT should be input for the *Lab* variable on the **D** command and the voltage input for the value. The word AMPS should be input for the *Lab* variable on the **F** command and the current into the node input for the value.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Element loads are described in Nodal Loading (p. 49). Element body loads may be input as heat generation rates at the nodes. The node J heat generation rate HG(J) defaults to the node I heat generation rate HG(I). This rate is in addition to the Joule heat generated by the current flow.

The current being calculated via this element can be directly coupled into a 3-D magnetostatic analysis [**BIOT**].

A summary of the element input is given in "LINK68 Input Summary" (p. 300). A general description of element input is given in Element Input (p. 43).

LINK68 Input Summary

Nodes
I, J

Degrees of Freedom
TEMP, VOLT

Real Constants
AREA - Cross-sectional area

Material Properties
MP command: KXX, DENS, C, ENTH, RSVX

Surface Loads
None

Body Loads

Heat Generations --
HG(I), HG(J)

Special Features

Birth and death

KEYOPTS
None

LINK68 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures and voltages included in the overall nodal solution
- Additional element output as shown in Table 68.1: LINK68 Element Output Definitions (p. 301)

The heat flow and the current flow into the nodes may be printed with the **OUTPR** command. The Joule heat generated this substep is used to determine the temperature distribution calculated for the next substep. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 68.1: LINK68 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	1
HGEN	Heat generations HG(I), HG(J)	Y	-
TG	Thermal gradient at centroid	Y	Y
TF	Thermal flux at centroid (heat flow/cross-sectional area)	Y	Y
EF	Electric field (voltage gradient)	Y	Y
JS	Current density (voltage flux)	Y	Y
CUR	Current	Y	Y
JHEAT:	Joule heat generation per unit volume	Y	Y

1. Available only at centroid as a ***GET** item.

Table 68.2: LINK68 Item and Sequence Numbers (p. 301) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 68.2: LINK68 Item and Sequence Numbers (p. 301):

Name
output quantity as defined in the Table 68.1: LINK68 Element Output Definitions (p. 301)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 68.2: LINK68 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
TG	NMISC	1
TF	NMISC	2

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
EF	NMISC	3
JS	NMISC	4
CUR	NMISC	5

LINK68 Assumptions and Restrictions

- Heat and current are assumed to flow only in the element longitudinal direction.
- The element must not have a zero length, that is, nodes I and J may not be coincident.
- A free end of the element (that is, not adjacent to another element and not subjected to a boundary constraint) is assumed to adiabatic.
- No conversion is included between electrical heat units and mechanical heat units.
- The resistivity may be divided by a conversion factor, such as 3.415 Btu/Hr per watt, to get Joule heat in mechanical units. Current (input and output) should also be converted for consistent units.
- If a current is specified at the same node that a voltage is specified, the current is ignored.
- The electrical and thermal solutions are coupled via an iterative procedure.
- There is no conversion required when consistent units are used.
- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, the elements must have the same reaction force (see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*).
- This element cannot be used in a distributed solution.

LINK68 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The birth and death special feature is not allowed.

ANSYS Emag 3-D

- This element has only electric field capability, and does not have thermal capability.
- The element may only be used in a steady-state electric analysis.
- The only active degree of freedom is VOLT.
- The only allowable material property is RSVX.
- No body loads are applicable.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The birth and death special feature is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID70

3-D Thermal Solid

MP ME <> PR PRN DS <> <> PP EME MFS
Product Restrictions

SOLID70 Element Description

SOLID70 has a 3-D thermal conduction capability. The element has eight nodes with a single degree of freedom, temperature, at each node. The element is applicable to a 3-D, steady-state or transient thermal analysis. The element also can compensate for mass transport heat flow from a constant velocity field. If the model containing the conducting solid element is also to be analyzed structurally, the element should be replaced by an equivalent structural element (such as SOLID185). See SOLID90 for a similar thermal element, with mid-edge node capability.

An option exists that allows the element to model nonlinear steady-state fluid flow through a porous medium. With this option, the thermal parameters are interpreted as analogous fluid flow parameters. For example, the temperature degree of freedom becomes equivalent to a pressure degree of freedom. See SOLID70 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 70.1: SOLID70 Geometry

SOLID70 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 70.1: SOLID70 Geometry (p. 305). The element is defined by eight nodes and the orthotropic material properties. A prism-shaped element, a tetrahedral-shaped element, and a pyramid-shaped element may also be formed as shown in Figure 70.1: SOLID70 Geometry (p. 305). Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Coordinate Systems (p. 55). Specific heat and density are ignored for steady-state solutions. Properties not input default as described in Linear Material Properties in the *Material Reference*.

Element loads are described in Nodal Loading (p. 49). Convection or heat flux (but not both) and radiation may be input as surface loads at the element faces as shown by the circled numbers on Figure 70.1: SOLID70 Geometry (p. 305).

Heat generation rates may be input as element body loads at the nodes. If the node I heat generation rate HG(I) is input, and all others are unspecified, they default to HG(I).

The nonlinear porous flow option is selected with KEYOPT(7) = 1. For this option, temperature is interpreted as pressure and the absolute permeability of the medium are input as material properties KXX, KYY, and KZZ. Properties DENS and VISC are used for the mass density and viscosity of the fluid. Properties C and MU are used in calculating the coefficients of permeability as described in the *Mechanical APDL Theory Reference*. Temperature boundary conditions input with the **D** command are interpreted as pressure boundary conditions, and heat flow boundary conditions input with the **F** command are interpreted as mass flow rate (mass/time).

A mass transport option is available with KEYOPT(8). With this option the velocities VX, VY, and VZ must be input as real constants (in the element coordinate system). Also, temperatures should be specified along the entire inlet boundary to assure a stable solution. With mass transport, you should use specific heat (C) and density (DENS) material properties instead of enthalpy (ENTH).

A summary of the element input is given in "SOLID70 Input Summary" (p. 306). A general description of element input is given in Element Input (p. 43).

SOLID70 Input Summary

Nodes

I, J, K, L, M, N, O, P

Degrees of Freedom

TEMP

Real Constants

Mass transport effects (KEYOPT(8) = 1):
VX - X direction of mass transport velocity
VY - Y direction of mass transport velocity
VZ - Z direction of mass transport velocity

Material Properties

MP command: KXX, KYY, KZZ, DENS, C, ENTH, VISC, MU (VISC and MU used only if KEYOPT(7) = 1.

Do not use ENTH with KEYOPT(8) = 1).

Surface Loads

Convection or Heat Flux (but not both) and Radiation (using Lab = RDSF) --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Body Loads

Heat Generations --
HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P)

Special Features
Birth and death

KEYOPT(2)
Evaluation of film coefficient:

0 --
Evaluate film coefficient (if any) at average film temperature, (TS + TB)/2

1 --
Evaluate at element surface temperature, TS

2 --
Evaluate at fluid bulk temperature, TB

3 --
Evaluate at differential temperature |TS-TB|

KEYOPT(4)
Element coordinate system defined:

0 --
Element coordinate system is parallel to the global coordinate system

1 --
Element coordinate system is based on the element I-J side

KEYOPT(7)
Nonlinear fluid flow option:

0 --
Standard heat transfer element

1 --
Nonlinear steady-state fluid flow analogy element

Note

Temperature degree of freedom interpreted as pressure.

KEYOPT(8)
Mass transport effects:

0 --
No mass transport effects

1 --
Mass transport with VX, VY, VZ

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID70 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output as shown in Table 70.1: SOLID70 Element Output Definitions (p. 308)

Convection heat flux is positive out of the element; applied heat flux is positive into the element. If KEYOPT(7) = 1, the standard thermal output should be interpreted as the analogous fluid flow output. The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 70.1: SOLID70 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	3
HGEN	Heat generations HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P)	Y	-
TG:X, Y, Z, SUM	Thermal gradient components and vector sum at centroid	Y	Y
TF:X, Y, Z, SUM	Thermal flux (heat flow rate/cross-sectional area) components and vector sum at centroid	Y	Y
FACE	Face label	1	-
AREA	Face area	1	1
NODES	Face nodes	1	-
HFILM	Film coefficient at each node of face	1	-
TBULK	Bulk temperature at each node of face	1	-
TAVG	Average face temperature	1	1
HEAT RATE	Heat flow rate across face by convection	1	1
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	1	-
HFAVG	Average film coefficient of the face	-	1
TBAVG	Average face bulk temperature	-	1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	1
HFLUX	Heat flux at each node of face	1	-
PRESSURE GRAD	Total pressure gradient and its X, Y, and Z components	2	-
MASS FLUX	Mass flow rate per unit cross-sectional area	2	-
FLUID VELOCITY	Total fluid velocity and its X, Y, and Z components	2	-

1. Output if a surface load is input
2. Output if KEYOPT(7) = 1
3. Available only at centroid as a ***GET** item.

Table 70.2: SOLID70 Item and Sequence Numbers (p. 309) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 70.2: SOLID70 Item and Sequence Numbers (p. 309):

Name
output quantity as defined in the Table 70.1: SOLID70 Element Output Definitions (p. 308)

Item
predetermined Item label for **ETABLE** command

FC*n*
sequence number for solution items for element Face *n*

Table 70.2: SOLID70 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input						
	Item	FC1	FC2	FC3	FC4	FC5	FC6
AREA	NMISC	1	7	13	19	25	31
HFAVG	NMISC	2	8	14	20	26	32
TAVG	NMISC	3	9	15	21	27	33
TBAVG	NMISC	4	10	16	22	28	34
HEAT RATE	NMISC	5	11	17	23	29	35
HFLXAVG	NMISC	6	12	18	24	30	36

SOLID70 Assumptions and Restrictions

- The element must not have a zero volume. This occurs most frequently when the element is not numbered properly.
- Elements may be numbered either as shown in Figure 70.1: SOLID70 Geometry (p. 305) or may have the planes IJKL and MNOP interchanged.
- A prism or tetrahedron shaped element may be formed by defining duplicate node numbers as described in Degenerated Shape Elements (p. 39).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The specific heat and enthalpy are evaluated at each integration point to allow for abrupt changes (such as for melting) within a coarse grid.
- If the thermal element is to be replaced by a SOLID185 structural element with surface stresses requested, the thermal element should be oriented such that face I-J-N-M and/or face K-L-P-O is a free surface.
- A free surface of the element (that is, not adjacent to another element and not subjected to a boundary constraint) is assumed to be adiabatic.
- Thermal transients having a fine integration time step and a severe thermal gradient at the surface will also require a fine mesh at the surface.
- If KEYOPT(8) > 0, unsymmetric matrices are produced.
- When mass flow is activated (KEYOPT(8)=1), the element Peclet number should be less than 1:

  $Pe = \rho*v*L*Cp/(2*k) < 1.0$

  Where L is an element length scale based on the element geometry. See SOLID70 in the *Mechanical APDL Theory Reference* for more details.

SOLID70 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- This element does not have the mass transport or fluid flow options. KEYOPT(7) and KEYOPT(8) can only be set to 0 (default).
- The VX, VY, and VZ real constants are not applicable.
- The VISC and MU material properties are not applicable.
- The element does not have the birth and death feature.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MASS71

Thermal Mass

MP ME ST PR PRN DS <> <> PP EME MFS
Product Restrictions

MASS71 Element Description

MASS71 is a point element having one degree of freedom, temperature, at the node. The element may be used in a transient thermal analysis to represent a body having thermal capacitance capability but negligible internal thermal resistance, that is, no significant temperature gradients within the body. The element also has a temperature-dependent heat generation capability. The lumped thermal mass element is applicable to a 1-D, 2-D, or 3-D steady-state or transient thermal analysis. See MASS71 in the *Mechanical APDL Theory Reference* for more details about this element.

In a steady-state solution the element acts only as a temperature-dependent heat source or sink. Other elements having special thermal applications are the COMBIN14 and COMBIN40 elements. These elements, which are normally used in structural models, may be used for thermally analogous situations.

If the model containing the thermal mass element is also to be analyzed structurally, the thermal element should be replaced by an equivalent structural element (such as MASS21)

Figure 71.1: MASS71 Geometry

MASS71 Input Data

The lumped thermal mass element is defined by one node (as shown in Figure 71.1: MASS71 Geometry (p. 311)) and a thermal capacitance (Heat/Degree). When used with axisymmetric elements, the thermal capacitance should be input on a full 360° basis. The thermal capacitance (CON1) may be input as a real constant or calculated (KEYOPT(3)) from the real constant volume (CON1) and either the DENS and C or ENTH material properties. KEYOPT(3) determines whether CON1 is interpreted as volume or thermal capacitance.

The heat generation is applied directly as a nodal load and is not first multiplied by the volume. Thus, if KEYOPT(3) = 0 (that is, when using the specific heat matrix), the heat generation must be adjusted to account for the volume. For an axisymmetric analysis the heat generation rate should be input on a full 360° basis. A temperature-dependent heat generation rate of the following polynomial form may be input:

$$\ddot{q}(T) = A_1 + A_2T + A_3T^{A_4} + A_5T^{A_6}$$

where T is the absolute temperature from the previous substep. The constants, A_1 through A_6, should be entered as real constants. If any of the constants A_2 through A_6 are nonzero, KEYOPT(4) must be set to 1. Also, if temperatures are not absolute, the offset conversion [**TOFFST**] must be specified.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Alternately, the heat generation expression may be defined as a temperature-dependent material property (QRATE) with the **MP** commands. QRATE can be input as numerical values or as tabular inputs evaluated as a function of temperature, time, and location.

A summary of the element input is given in "MASS71 Input Summary" (p. 312). A general description of element input is given in Element Input (p. 43). See Harmonic Axisymmetric Elements (p. 83) for more details.

MASS71 Input Summary

Nodes

I

Degrees of Freedom

TEMP

Real Constants

CON1, A1, A2, A3, A4, A5
A6
See Table 71.1: MASS71 Real Constants (p. 313) for a description of the real constants

Material Properties

MP command: QRATE, DENS, C, ENTH if KEYOPT(3) = 0 or QRATE if KEYOPT(3) = 1

Surface Loads

None

Body Loads

None (heat generation may be defined as a function of temperature by using real constants A1, A2, ... or by the QRATE material property definition.)

Special Features

Birth and death
Nonlinearity if heat generation is defined as a function of temperature

KEYOPT(3)

Interpretation of real constant CON1:

0 --
Interpret CON1 as volume (with either DENS and C or ENTH supplied as material properties)

1 --
Interpret CON1 as thermal capacitance (DENS*C*volume)

KEYOPT(4)

Temperature dependent heat generation:

0 --
No temperature-dependent heat generation (required if all real constants A2-A6 are zero)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
Include temperature-dependent heat generation (required if any real constants A2-A6 are nonzero)

Table 71.1: MASS71 Real Constants

No.	Name	Description
1	CON1	Volume or thermal capacitance (see KEYOPT(3))
2	A1	Constant for temperature function
3	A2	Constant for temperature function
4	A3	Constant for temperature function
5	A4	Constant for temperature function
6	A5	Constant for temperature function
7	A6	Constant for temperature function

MASS71 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output as shown in Table 71.2: MASS71 Element Output Definitions (p. 313)

The heat generation is in units of Heat/Time and is positive into the node. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 71.2: MASS71 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODE	Node I	Y	Y
XC, YC, ZC	Location where results are reported	Y	1
TEMP	Element (node) temperature	Y	Y
HEAT	Heat generation rate into node	Y	Y

1. Available only at centroid as a ***GET** item.

Table 71.3: MASS71 Item and Sequence Numbers (p. 314) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 71.3: MASS71 Item and Sequence Numbers (p. 314):

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name
output quantity as defined in the Table 71.2: MASS71 Element Output Definitions (p. 313)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 71.3: MASS71 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
HEAT	SMISC	1
TEMP	SMISC	2

MASS71 Assumptions and Restrictions

- When using the element with a temperature-dependent heat generation in a steady-state solution, an iterative solution is required.
- The heat generation is calculated at the uniform temperature for the first substep.

MASS71 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The birth and death special feature is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PLANE75

Axisymmetric-Harmonic 4-Node Thermal Solid

MP ME ST <> <> <> <> <> PP EME MFS
Product Restrictions

PLANE75 Element Description

PLANE75 is used as an axisymmetric ring element with a 3-D thermal conduction capability. The element has four nodes with a single degree of freedom, temperature, at each node. The element is a generalization of the axisymmetric version of PLANE55 in that it allows nonaxisymmetric loading. Various loading cases are described in Harmonic Axisymmetric Elements with Nonaxisymmetric Loads (p. 84).

The element is applicable to an axisymmetric geometry for steady-state or transient thermal analyses. See PLANE75 in the *Mechanical APDL Theory Reference* for more details about this element. If the model containing the element is also to be analyzed structurally, the element should be replaced by the equivalent structural element (such as PLANE25). A similar thermal element with midside node capability is PLANE78.

Figure 75.1: PLANE75 Geometry

PLANE75 Input Data

The geometry, node locations, and the coordinate system for this axisymmetric thermal solid element are shown in Figure 75.1: PLANE75 Geometry (p. 315). The data input is essentially the same as for PLANE55 and is described in "PLANE55 Input Data" (p. 267). The element input data also includes the number of harmonic waves (*MODE*) and the symmetry condition (*ISYM*) on the **MODE** command. If *MODE* = 0 and *ISYM* = 1, the element behaves similarly to the axisymmetric case of PLANE55. The *MODE* and *ISYM* parameters describe the type of temperature distribution and are discussed in detail in Harmonic Axisymmetric Elements with Nonaxisymmetric Loads (p. 84).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Element loads are described in Nodal Loading (p. 49). Harmonically varying bulk temperatures or heat fluxes (but not both) may be input as surface loads on the element faces as shown by the circled numbers on Figure 75.1: PLANE75 Geometry (p. 315). Harmonically varying heat generation rates may be input as element body loads at the nodes. If the node I heat generation rate HG(I) is input and all others are unspecified, they default to HG(I).

A summary of the element input is given in "PLANE75 Input Summary" (p. 316). A general description of element input is given in Element Input (p. 43).

PLANE75 Input Summary

Nodes
I, J, K, L

Degrees of Freedom
TEMP

Real Constants
None

Material Properties
MP command: KXX, KYY, KZZ, DENS, C, ENTH

Surface Loads

Convections --
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Heat Fluxes --
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads

Heat Generations --
HG(I), HG(J), HG(K), HG(L)

Mode Number
Number of harmonic waves around the circumference (**MODE**)

Loading Condition
Symmetry condition (**MODE**)

Special Features
Birth and death

KEYOPTS
None

PLANE75 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output as shown in Table 75.1: PLANE75 Element Output Definitions (p. 317)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Convection heat flux is positive out of the element; applied heat flux is positive into the element. The element output directions are parallel to the element coordinate system. The face area and the heat flow rate are on a full 360° basis. For more information about harmonic elements, see Harmonic Axisymmetric Elements with Nonaxisymmetric Loads (p. 84). A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 75.1: PLANE75 Element Output Definitions

Name	Definition	O	R
EL	Element number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	3
HGEN	Heat generations HG(I), HG(J), HG(K), HG(L)	Y	-
MODE	Number of waves in loading	Y	-
TG:X, Y, SUM, Z	Thermal gradient components and vector sum (X and Y) at centroid	1	1
TF:X, Y, SUM, Z	Thermal flux (heat flow rate/cross-sectional area) components and vector sum (X and Y) at centroid	1	1
FACE	Face label	2	-
NODES	Face nodes	2	-
AREA	Face area	2	2
TAVG, TBULK	Average of the two end nodal temperatures evaluated at peak value, fluid bulk temperature evaluated at peak value	2	2
HEAT RATE	Heat flow rate across face by convection	2	2
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	2	-
HFAVG	Average film coefficient of the face	-	2
TBAVG	Average face bulk temperature	-	2
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	2
HFLUX	Heat flux at each node of face	2	-

1. Gradient and flux peak at THETA = 0 and THETA = 90 ÷ MODE degrees

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Output if a surface load is input

3. Available only at centroid as a ***GET** item.

Table 75.2: PLANE75 Item and Sequence Numbers (p. 318) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 75.2: PLANE75 Item and Sequence Numbers (p. 318):

Name
output quantity as defined in the Table 75.1: PLANE75 Element Output Definitions (p. 317)

Item
predetermined Item label for **ETABLE** command

FC*n*
sequence number for solution items for element Face *n*

Table 75.2: PLANE75 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	FC1	FC2	FC3	FC4
AREA	NMISC	1	7	13	19
HFAVG	NMISC	2	8	14	20
TAVG	NMISC	3	9	15	21
TBAVG	NMISC	4	10	16	22
HEAT RATE	NMISC	5	11	17	23
HFLXAVG	NMISC	6	12	18	24

PLANE75 Assumptions and Restrictions

- The element must not have a negative or a zero area.
- The element must lie in the global X-Y plane as shown in Figure 75.1: PLANE75 Geometry (p. 315) and the Y-axis must be the axis of symmetry for axisymmetric analyses.
- An axisymmetric structure should be modeled in the +X quadrants.
- A triangular element may be formed by defining duplicate K and L node numbers as described in Degenerated Shape Elements (p. 39).
- If the thermal element is to be replaced by the analogous structural element (PLANE25) with surface stresses requested, the thermal element should be oriented so that face I-J (and also face K-L, if applicable) is a free surface.
- A free surface of the element (that is, not adjacent to another element and not subjected to a boundary constraint) is assumed to be adiabatic.
- Thermal transients having a fine integration time step and a severe thermal gradient at the surface will also require a fine mesh at the surface.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Temperature-dependent material properties (including the film coefficient) are assumed to be axisymmetric even if the temperature varies harmonically.
- If MODE = 0, properties are evaluated at the temperatures calculated in the previous substep (or at TUNIF if for the first substep).
- If MODE > 0, properties are evaluated at temperatures calculated from the previous MODE = 0 substep; if no MODE = 0 substep exists, then evaluation is done at 0.0 degrees.

PLANE75 Product Restrictions

There are no product restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

PLANE77

2-D 8-Node Thermal Solid

MP ME <> PR PRN DS <> <> PP EME MFS
Product Restrictions

PLANE77 Element Description

PLANE77 is a higher order version of the 2-D, 4-node thermal element (PLANE55). The element has one degree of freedom, temperature, at each node. The 8-node elements have compatible temperature shapes and are well suited to model curved boundaries.

The 8-node thermal element is applicable to a 2-D, steady-state or transient thermal analysis. See PLANE77 in the *Mechanical APDL Theory Reference* for more details about this element. If the model containing this element is also to be analyzed structurally, the element should be replaced by an equivalent structural element (such as PLANE183). A similar axisymmetric thermal element which accepts nonaxisymmetric loading is PLANE78.

Figure 77.1: PLANE77 Geometry

PLANE77 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 77.1: PLANE77 Geometry (p. 321). The element is defined by eight nodes and orthotropic material properties. A triangular-shaped element may be formed by defining the same node number for nodes K, L and O.

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems (p. 55). Specific heat and density are ignored for steady-state solutions. Properties not input default as described in Linear Material Properties in the *Material Reference*.

Element loads are described in Nodal Loading (p. 49). Convection or heat flux (but not both) and radiation may be input as surface loads at the element faces as shown by the circled numbers on Figure 77.1: PLANE77 Geometry (p. 321). Heat generation rates may be input as element body loads at the nodes. If the node I heat generation rate HG(I) is input, and all others are unspecified, they default to HG(I). If all corner node heat generation rates are specified, each midside node heat generation rate defaults to the average heat generation rate of its adjacent corner nodes.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This element can also have a Z-depth specified by KEYOPT(3) and real constant THK. Be careful when using this option with other physics, especially radiation. Radiation view factors will be based on a unit Z-depth (only).

A summary of the element input is given in "PLANE77 Input Summary" (p. 322). A general description of element input is given in Element Input (p. 43). For axisymmetric applications see Harmonic Axisymmetric Elements (p. 83).

PLANE77 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
TEMP

Real Constants
THK

THK = Thickness (used only if KEYOPT(3) = 3)

Material Properties
MP command: KXX, KYY, DENS, C, ENTH

Surface Loads

Convection or Heat Flux (but not both) and Radiation (using Lab = RDSF) --
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads

Heat Generations --
HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P)

Special Features
Birth and death

KEYOPT(1)
Specific heat matrix:

0 --
Consistent specific heat matrix

1 --
Diagonalized specific heat matrix

KEYOPT(3)
Element behavior:

0 --
Plane

1 --
Axisymmetric

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3 --
Plane with Z-depth, specified via real constant THK.

PLANE77 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output as shown in Table 77.1: PLANE77 Element Output Definitions (p. 323)

The element output directions are parallel to the element coordinate system. For an axisymmetric analysis the face area and the heat flow rate are on a full 360° basis. Convection heat flux is positive out of the element; applied heat flux is positive into the element. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 77.1: PLANE77 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	2
HGEN	Heat generations HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P)	Y	-
TG:X, Y, SUM	Thermal gradient components and vector sum at centroid	Y	Y
TF:X, Y, SUM	Thermal flux (heat flow rate/cross-sectional area) components and vector sum at centroid	Y	Y
FACE	Face label	1	-
NODES	Face nodes	1	-
AREA	Face area	1	1
HFILM	Film coefficient	1	-
TAVG	Average face temperature	1	1
TBULK	Fluid bulk temperature	1	-
HEAT RATE	Heat flow rate across face by convection	1	1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	1	-
HFAVG	Average film coefficient of the face	-	1
TBAVG	Average face bulk temperature	-	1
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	1
HFLUX	Heat flux at each node of face	1	-

1. Output only if a surface load is input

2. Available only at centroid as a ***GET** item.

Table 77.2: PLANE77 Item and Component Labels (p. 324) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 77.2: PLANE77 Item and Component Labels (p. 324):

Name
output quantity as defined in the Table 77.1: PLANE77 Element Output Definitions (p. 323)

Item
predetermined Item label for **ETABLE** command

FC*n*
sequence number for solution items for element Face *n*

Table 77.2: PLANE77 Item and Component Labels

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	FC1	FC2	FC3	FC4
AREA	NMISC	1	7	13	19
HFAVG	NMISC	2	8	14	20
TAVG	NMISC	3	9	15	21
TBAVG	NMISC	4	10	16	22
HEAT RATE	NMISC	5	11	17	23
HFLXAVG	NMISC	6	12	18	24

PLANE77 Assumptions and Restrictions

- The area of the element must be positive.
- The 2-D element must lie in an X-Y plane as shown in Figure 77.1: PLANE77 Geometry (p. 321) and the Y-axis must be the axis of symmetry for axisymmetric analyses.
- An axisymmetric structure should be modeled in the +X quadrants.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- A face with a removed midside node implies that the temperature varies linearly, rather than parabolically, along that face. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.
- The specific heat and enthalpy are evaluated at each integration point to allow for abrupt changes (such as melting) within a coarse grid.
- If the thermal element is to be replaced by a PLANE183 structural element with surface stresses requested, the thermal element may be oriented such that face IJ and/or face KL is a free surface. A free surface of the element (that is, not adjacent to another element and not subjected to a boundary constraint) is assumed to be adiabatic.
- Thermal transients having a fine integration time step and a severe thermal gradient at the surface will require a fine mesh at the surface.

PLANE77 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The birth and death special feature is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

PLANE78

Axisymmetric-Harmonic 8-Node Thermal Solid

MP ME <> <> <> <> <> <> PP EME MFS
Product Restrictions

PLANE78 Element Description

PLANE78 is used as an axisymmetric ring element with a 3-D thermal conduction capability. The element has one degree of freedom, temperature, at each node. PLANE78 is a generalization of PLANE77 in that it allows nonaxisymmetric loading. Various loading cases are described in Harmonic Axisymmetric Elements with Nonaxisymmetric Loads (p. 84).

The 8-node elements have compatible temperature shapes and are well suited to model curved boundaries.

The element is applicable to an axisymmetric geometry for steady-state or transient thermal analyses. See PLANE78 in the *Mechanical APDL Theory Reference* for more details about this element. If the model containing the element is also to be analyzed structurally, the element should be replaced by the equivalent structural element (such as PLANE83).

Figure 78.1: PLANE78 Geometry

PLANE78 Input Data

The geometry, node locations, and the coordinate system for this axisymmetric thermal solid element are shown in Figure 78.1: PLANE78 Geometry (p. 327). The data input is essentially the same as for PLANE77 and is described in "PLANE77 Input Data" (p. 321). The element input data also includes the number of harmonic waves (*MODE* on the **MODE** command) and the symmetry condition (*ISYM* on the **MODE** command). If *MODE* = 0 and *ISYM* = 1, the element behaves similarly to the axisymmetric case of PLANE77. If *MODE* equals 1, the temperature is assumed to be 0° along an entire diameter. The *MODE* and *ISYM* parameters describe the type of temperature distribution and are discussed in detail in Harmonic Axisymmetric Elements with Nonaxisymmetric Loads (p. 84).

Element loads are described in Nodal Loading (p. 49). Harmonically varying bulk temperatures or heat fluxes (but not both) may be input as surface loads on the element faces as shown by the circled numbers on Figure 78.1: PLANE78 Geometry (p. 327). Harmonically varying heat generation rates may

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

be input as element body loads at the nodes. If the node I heat generation rate HG(I) is input and all others are unspecified, they default to HG(I). If all corner node heat generation rates are specified, each midside node heat generation rate defaults to the average heat generation rate of its adjacent corner nodes.

A summary of the element input is given in "PLANE78 Input Summary" (p. 328). A general description of element input is given in Element Input (p. 43).

PLANE78 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
TEMP

Real Constants
None

Material Properties
MP command: KXX, KYY, KZZ, DENS, C, ENTH

Surface Loads

Convections --
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Heat Fluxes --
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads

Heat Generations --
HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P)

Mode Number
Number of harmonic waves around the circumference (**MODE**)

Special Features
Birth and death

Loading Condition
Symmetry condition (**MODE**)

KEYOPT(1)
Specific heat matrix:

0 --
Consistent specific heat matrix

1 --
Diagonalized specific heat matrix

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PLANE78 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output as shown in Table 78.1: PLANE78 Element Output Definitions (p. 329)

Convection heat flux is positive out of the element; applied heat flux is positive into the element. The element output directions are parallel to the element coordinate system. The face area and the heat flow rate are on a full 360° basis. For more information about harmonic elements, see Harmonic Axisymmetric Elements with Nonaxisymmetric Loads (p. 84). A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 78.1: PLANE78 Element Output Definitions

Name	Definition	O	R
EL	Element number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
MODE	Number of waves in loading	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	3
HGEN	Heat generations HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P)	Y	-
TG:X, Y, SUM, Z	Thermal gradient components and vector sum (X and Y) at centroid	1	1
TF:X, Y, SUM, Z	Thermal flux (heat flow rate/cross-sectional area) components and vector sum (X and Y) at centroid	1	1
FACE	Face label	2	2
NODES	Face nodes	2	2
AREA	Face area	2	2
HFILM	Film coefficient	2	2
TAVG, TBULK	Average of the two end nodal temperatures evaluated at peak value, fluid bulk temperature at peak value	2	2
HEAT RATE	Heat flow rate across face by convection	2	2
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	2	2

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
HFAVG	Average film coefficient of the face	-	2
TBAVG	Average face bulk temperature	-	2
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	2
HFLUX	Heat flux at each node of face	2	2

1. Gradient and flux peak at THETA = 0 and THETA = 90 ÷ Mode degrees

2. Output only if a surface load is input

3. Available only at centroid as a ***GET** item.

Table 78.2: PLANE78 Item and Sequence Numbers (p. 330) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 78.2: PLANE78 Item and Sequence Numbers (p. 330):

Name
output quantity as defined in the Table 78.1: PLANE78 Element Output Definitions (p. 329)

Item
predetermined Item label for **ETABLE** command

FC*n*
sequence number for solution items for element Face *n*

Table 78.2: PLANE78 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	FC1	FC2	FC3	FC4
AREA	NMISC	1	7	13	19
HFAVG	NMISC	2	8	14	20
TAVG	NMISC	3	9	15	21
TBAVG	NMISC	4	10	16	22
HEAT RATE	NMISC	5	11	17	23
HFLXAVG	NMISC	6	12	18	24

PLANE78 Assumptions and Restrictions

- The element must not have a negative or a zero area.

- The element must lie in the global X-Y plane as shown in Figure 78.1: PLANE78 Geometry (p. 327) and the Y-axis must be the axis of symmetry for axisymmetric analyses.

- An axisymmetric structure should be modeled in the +X quadrants.

- A face with a removed midside node implies that the temperature varies linearly, rather than parabolically, along that face. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If the thermal element is to be replaced by the analogous structural element (PLANE83) with surface stresses requested, the thermal element should be oriented so that face IJ (and also face KL, if applicable) is a free surface. A free surface of the element (that is, not adjacent to another element and not subjected to a boundary constraint) is assumed to be adiabatic.
- Thermal transients having a fine integration time step and a severe thermal gradient at the surface will also require a fine mesh at the surface.
- Temperature-dependent material properties (including the film coefficient) are assumed to be axisymmetric even if the temperature varies harmonically.
- If *MODE* = 0, properties are evaluated at the temperatures calculated in the previous substep (or at TUNIF if for the first substep).
- If *MODE* > 0, properties are evaluated at temperatures calculated from the previous *MODE* = 0 substep; if no *MODE* = 0 substep exists, then evaluation is done at 0.0 degrees.

PLANE78 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

PLANE83

Axisymmetric-Harmonic 8-Node Structural Solid

MP ME ST <> <> <> <> <> PP EME MFS
Product Restrictions

PLANE83 Element Description

Although this legacy element is available for use in your analysis, ANSYS, Inc. recommends using a current-technology element such as SOLID273 (KEYOPT(6) = 0).

PLANE83 is used for 2-D modeling of axisymmetric structures with nonaxisymmetric loading. Examples of such loading are bending, shear, or torsion. The element has three degrees of freedom per node: translations in the nodal x, y, and z directions. For unrotated nodal coordinates, these directions correspond to the radial, axial, and tangential directions, respectively.

This element is a higher order version of the 2-D, four-node element (PLANE25). It provides more accurate results for mixed (quadrilateral-triangular) automatic meshes and can tolerate irregular shapes without as much loss of accuracy. The loading need not be axisymmetric. Various loading cases are described in Harmonic Axisymmetric Elements with Nonaxisymmetric Loads (p. 84).

The 8-node elements have compatible displacement shapes and are well suited to model curved boundaries. See PLANE83 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 83.1: PLANE83 Geometry

PLANE83 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 83.1: PLANE83 Geometry (p. 333). Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems (p. 55). Z-direction material properties (EZ, ALPZ, etc.) may be input. MODE and ISYM are used to describe the harmonic loading condition. (See Harmonic Axisymmetric Elements with Nonaxisymmetric Loads (p. 84) for more details.)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Element loads are described in Nodal Loading (p. 49). Harmonically varying pressures may be input as surface loads on the element faces as shown by the circled numbers on Figure 83.1: PLANE83 Geometry (p. 333). Positive pressures act into the element. Harmonically varying temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). If all corner node temperatures are specified, each midside node temperature defaults to the average temperature of its adjacent corner nodes. For any other input temperature pattern, unspecified temperatures default to TUNIF.

KEYOPT(3) is used for temperature loading with MODE > 0 and temperature-dependent material properties. Material properties may only be evaluated at a constant (nonharmonically varying) temperature. If MODE = 0, the material properties are always evaluated at the average element temperature. If MODE > 0, TREF must be input as zero.

KEYOPT(4), (5), and (6) provide various element printout options. (See Element Solution (p. 51).)

A summary of the element input is given in "PLANE83 Input Summary" (p. 334). A general description of element input is given in Element Input (p. 43).

PLANE83 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
UX, UY, UZ

Real Constants
None

Material Properties

MP command: EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ), DENS, GXY, GYZ, GXZ, ALPD, BETD

Surface Loads

Pressures --
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

Mode Number
Number of harmonic waves around the circumference (**MODE**)

Loading Condition
Symmetry condition (**MODE**)

Special Features

Birth and death
Stress stiffening

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(1)
Element coordinate system:

0 --
Element coordinate system is parallel to the global coordinate system

1 --
Element coordinate system is based on the element I-J side

KEYOPT(3)
If MODE is greater than zero, use temperatures for:

0 --
Use temperatures only for thermal bending (evaluate material properties at TREF)

1 --
Use temperatures only for material property evaluation (thermal strains are not computed)

KEYOPT(4)
Extra stress output:

0 --
Basic element solution (not extra output)

1 --
Repeat basic solution for all integration points

2 --
Nodal stress solution

KEYOPT(5)
Combined stress output:

0 --
No combined stress solution

1 --
Combined stress solution at centroid and nodes

KEYOPT(6)
Extra surface output (surface solution is valid only for isotropic materials):

0 --
Basic element solution (no extra output)

1 --
Surface solution for face I-J also

2 --
Surface solution for both faces I-J and K-L also

PLANE83 Output Data

The solution output associated with the element is in two forms:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 83.1: PLANE83 Element Output Definitions (p. 336)

Several items are illustrated in Figure 83.2: PLANE83 Stress Output (p. 336).

In the displacement printout, the UZ component is out-of-phase with the UX and UY components. For example, in the MODE = 1, ISYM = 1 loading case, UX and UY are the peak values at $\theta = 0°$ and UZ is the peak value at $\theta = 90°$. The same occurs for the reaction forces (FX, FY, etc.). We recommend that you always use the *angle* field on the **SET** command when postprocessing the results. For more information about harmonic elements, see Harmonic Axisymmetric Elements with Nonaxisymmetric Loads (p. 84).

The element stress directions are parallel to the element coordinate system. The sign convention on the surface shears is such that for a rectangular element that is lined up parallel to the axes with node J in the positive Y direction from node I, the shear stresses on surfaces I-J and K-L are analogous to the centroidal SYZ in both definition and sign. Stress components which are inherently zero for a load case are printed for clarity. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

Figure 83.2: PLANE83 Stress Output

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 83.1: PLANE83 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Corner nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
ISYM	Loading key: 1 = symmetric, -1 = antisymmetric	Y	-
MODE	Number of waves in loading	Y	Y
VOLU:	Volume	Y	Y
PRES	Pressures P1 at nodes J,I; P2 at K,J; P3 at L,K; P4 at I,L	Y	Y
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	Y	Y
S:X, Y, Z	Direct stresses (radial, axial, hoop) at *PK ANG* locations	Y	Y
S:XY, YZ, XZ	Shear stresses (radial-axial, axial-hoop, radial-hoop) at *PK ANG* locations	Y	Y
S:1, 2, 3	Principal stresses at both *PK ANG* locations as well as where extreme occurs (EXTR); if MODE = 0, only one location is given.	1	1
S:INT	Stress intensity at both *PK ANG* locations as well as where extreme occurs (EXTR); if MODE = 0, only one location is given.	1	1
S:EQV	Equivalent stress at both *PK ANG* locations as well as where extreme occurs (EXTR); if MODE = 0, only one location is given.	1	1
EPEL:X, Y, Z, XY	Elastic strain	Y	Y
EPEL:EQV	Equivalent elastic strain [4]	-	Y
EPTH:X, Y, Z, XY	Average thermal strains	Y	Y
EPTH:EQV	Equivalent thermal strain [4]	-	Y
PK ANG	Angle where stresses have peak values: 0 and 90/MODE°. Blank if MODE = 0.	Y	Y
XC, YC	Location where results are reported	Y	3
FACE	Face label	2	2
TEMP	Surface average temperature	2	2
EPEL(PAR, PER, Z, SH)	Surface strains (parallel, perpendicular, hoop, shear) at *PK ANG* locations and where extreme occurs (EXTR)	2	2
S(PAR, PER, Z, SH)	Surface stresses (parallel, perpendicular, hoop, shear) at *PK ANG* locations and where extreme occurs (EXTR)	2	2

1. These items are output only if KEYOPT(5) = 1.
2. These items are printed only if KEYOPT(6) is greater than zero.
3. Available only at centroid as a ***GET** item.
4. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Table 83.2: PLANE83 Item and Sequence Numbers (p. 338) lists output available through the **ETABLE** command using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 83.2: PLANE83 Item and Sequence Numbers (p. 338):

Name
output quantity as defined in the Table 83.1: PLANE83 Element Output Definitions (p. 336)

Item
predetermined Item label for **ETABLE** command

I,J,K,L
sequence number for data at nodes I,J,K,L

Table 83.2: PLANE83 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	I	J	K	L
P1	SMISC	2	1	-	-
P2	SMISC	-	4	3	-
P3	SMISC	-	-	6	5
P4	SMISC	7	-	-	8
THETA = 0					
S1	NMISC	1	16	31	46
S2	NMISC	2	17	32	47
S3	NMISC	3	18	33	48
SINT	NMISC	4	19	34	49
SEQV	NMISC	5	20	35	50
THETA = 90/MODE					
S1	NMISC	6	21	36	51
S2	NMISC	7	22	37	52
S3	NMISC	8	23	38	53
SINT	NMISC	9	24	39	54
SEQV	NMISC	10	25	40	55
EXTR Values					
S1	NMISC	11	26	41	56
S2	NMISC	12	27	42	57
S3	NMISC	13	28	43	58
SINT	NMISC	14	29	44	59
SEQV	NMISC	15	30	45	60

The NMISC items (1 thru 60) in the above table represent the combined stress solution, KEYOPT(5) = 1. If MODE = 0, their values are zero at THETA = 90/MODE and at EXTR.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See Surface Solution for the item and sequence numbers for surface output for the **ETABLE** command.

PLANE83 Assumptions and Restrictions

- The area of the element must be positive.
- The element must be defined in the global X-Y plane as shown in Figure 83.1: PLANE83 Geometry (p. 333) and the Y-axis must be the axis of symmetry for axisymmetric analyses. An axisymmetric structure should be modeled in the +X quadrants.
- A face with a removed midside node implies that the displacement varies linearly, rather than parabolically, along that face. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.
- The element assumes a linear elastic material.
- Post-analysis superposition of results is valid only with other linear elastic solutions.
- The element should not be used with the large deflection option.
- The element may not be deactivated with the **EKILL** command.
- The element temperature is taken to be the average of the nodal temperatures.
- Surface stress printout is valid only if the conditions described in Element Solution (p. 51) are met.

Modeling hints:

- If shear effects are important in a shell-like structure, at least two elements through the thickness should be used.
- You can use only axisymmetric (**MODE**,0) loads without significant torsional stresses to generate the stress state used for stress stiffened modal analyses using this element.

PLANE83 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID87

3-D 10-Node Tetrahedral Thermal Solid

MP ME <> PR PRN DS <> <> PP EME MFS
Product Restrictions

SOLID87 Element Description

SOLID87 is well suited to model irregular meshes (such as produced from various CAD/CAM systems). The element has one degree of freedom, temperature, at each node.

The element is applicable to a 3-D, steady-state or transient thermal analysis. See SOLID87 in the *Mechanical APDL Theory Reference* for more details about this element. If the model containing this element is also to be analyzed structurally, the element should be replaced by the equivalent structural element (such as SOLID187). A 20-node thermal solid element, SOLID90, is also available.

Figure 87.1: SOLID87 Geometry

SOLID87 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 87.1: SOLID87 Geometry (p. 341).

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems (p. 55). Specific heat and density are ignored for steady-state solutions. Properties not input default as described in Linear Material Properties in the *Material Reference*.

Element loads are described in Nodal Loading (p. 49). Convection or heat flux (but not both) and radiation may be input as surface loads at the element faces as shown by the circled numbers on Figure 87.1: SOLID87 Geometry (p. 341). Heat generation rates may be input as element body loads at the nodes. If the node I heat generation rate HG(I) is input, and all others are unspecified, they default to HG(I). If all corner node heat generation rates are specified, each midside node heat generation rate defaults to the average heat generation rate of its adjacent corner nodes.

For phase change problems, use KEYOPT(1) = 1 (diagonalized specific heat matrix). For convection regions with strong thermal gradients, use KEYOPT(5) = 1 (consistent convection matrix).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A summary of the element input is given in "SOLID87 Input Summary" (p. 342). A general description of element input is given in Element Input (p. 43).

SOLID87 Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R

Degrees of Freedom
TEMP

Real Constants
None

Material Properties
MP command: KXX, KYY, KZZ, DENS, C, ENTH

Surface Loads

Convection or Heat Flux (but not both) and Radiation (using Lab = RDSF) --
face 1 (J-I-K), face 2 (I-J-L), face 3 (J-K-L), face 4 (K-I-L)

Body Loads

Heat Generations --
HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P), HG(Q), HG(R)

Special Features
Birth and death

KEYOPT(1)
Specific heat matrix:

0 --
Consistent specific heat matrix

1 --
Diagonalized specific heat matrix

KEYOPT(5)
Surface convection matrix:

0 --
Diagonalized convection matrix

1 --
Consistent convection matrix

SOLID87 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output as shown in Table 87.1: SOLID87 Element Output Definitions (p. 343).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Convection heat flux is positive out of the element; applied heat flux is positive into the element. The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 87.1: SOLID87 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P, Q, R	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	2
HGEN	Heat generations HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P), HG(Q), HG(R)	Y	-
TG:X, Y, Z, SUM	Thermal gradient components and vector sum at centroid	Y	Y
TF:X, Y, Z, SUM	Thermal flux (heat flow rate/cross-sectional area) components and vector sum at centroid	Y	Y
FACE	Convection face label	1	-
NODES	Convection face corner nodes	1	-
AREA	Convection face area	1	1
HFILM	Film coefficient	1	-
TAVG	Average face temperature	1	1
TBULK	Fluid bulk temperature	1	-
HEAT RATE	Heat flow rate across face by convection	1	1
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	1	-
HFAVG	Average film coefficient of the face	-	1
TBAVG	Average face bulk temperature	-	1
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	1
HFLUX	Heat flux at each node of face	1	-

1. Output if a surface load is input

2. Available only at centroid as a ***GET** item.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Table 87.2: SOLID87 Item and Sequence Numbers (p. 344) lists output available through the **ETABLE** command using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) of this manual for more information. The following notation is used in Table 87.2: SOLID87 Item and Sequence Numbers (p. 344):

Name
output quantity as defined in the Table 87.1: SOLID87 Element Output Definitions (p. 343)

Item
predetermined Item label for **ETABLE** command

FC*n*
sequence number for solution items for element Face *n*

Table 87.2: SOLID87 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	FC1	FC2	FC3	FC4
AREA	NMISC	1	7	13	19
HFAVG	NMISC	2	8	14	20
TAVG	NMISC	3	9	15	21
TBAVG	NMISC	4	10	16	22
HEAT RATE	NMISC	5	11	17	23
HFLXAVG	NMISC	6	12	18	24

SOLID87 Assumptions and Restrictions

- The element must not have a zero volume.
- Elements may be numbered either as shown in Figure 87.1: SOLID87 Geometry (p. 341) or may have node L below the IJK plane.
- An edge with a removed midside node implies that the temperature varies linearly, rather than parabolically, along that edge.
- See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.
- The specific heat and enthalpy are evaluated at each integration point to allow for abrupt changes (such as melting) within a coarse grid of elements.
- A free surface of the element (i.e., not adjacent to another element and not subjected to a boundary constraint) is assumed to be adiabatic.

SOLID87 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The birth and death special feature is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID90

3-D 20-Node Thermal Solid

MP ME <> PR PRN DS <> <> PP EME MFS
Product Restrictions

SOLID90 Element Description

SOLID90 is a higher order version of the 3-D eight node thermal element (SOLID70). The element has 20 nodes with a single degree of freedom, temperature, at each node. The 20-node elements have compatible temperature shapes and are well suited to model curved boundaries.

The 20-node thermal element is applicable to a 3-D, steady-state or transient thermal analysis. See SOLID90 in the *Mechanical APDL Theory Reference* for more details about this element. If the model containing this element is also to be analyzed structurally, the element should be replaced by the equivalent structural element (such as SOLID186).

Figure 90.1: SOLID90 Geometry

SOLID90 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 90.1: SOLID90 Geometry (p. 347). The element is defined by 20 node points and the material properties. A prism-shaped element may be formed by defining duplicate K, L, and S; A and B; and O, P, and W node numbers. A tetrahedral-shaped element and a pyramid-shaped element may also be formed as shown in Figure 90.1: SOLID90 Geometry (p. 347).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems (p. 55). Specific heat and density are ignored for steady-state solutions. Properties not input default as described in Linear Material Properties in the *Material Reference*.

Element loads are described in Nodal Loading (p. 49). Convection or heat flux (but not both) and radiation may be input as surface loads at the element faces as shown by the circled numbers on Figure 90.1: SOLID90 Geometry (p. 347). Heat generation rates may be input as element body loads at the nodes. If the node I heat generation rate HG(I) is input, and all others are unspecified, they default to HG(I). If all corner node heat generation rates are specified, each midside node heat generation rate defaults to the average heat generation rate of its adjacent corner nodes.

For phase change problems, use KEYOPT(1) = 1 (diagonalized specific heat matrix).

A summary of the element input is given in "SOLID90 Input Summary" (p. 348). A general description of element input is given in Element Input (p. 43).

SOLID90 Input Summary

Nodes

I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B

Degrees of Freedom

TEMP

Real Constants

None

Material Properties

MP command: KXX, KYY, KZZ, DENS, C, ENTH

Surface Loads

Convection or Heat Flux (but not both) and Radiation (using Lab = RDSF) --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Heat Generations --

HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P), HG(Q), HG(R),
HG(S), HG(T), HG(U), HG(V), HG(W), HG(X), HG(Y), HG(Z), HG(A), HG(B)

Special Features

Birth and death

KEYOPT(1)

Specific heat matrix:

0 --

Consistent specific heat matrix

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
Diagonalized specific heat matrix

SOLID90 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output as shown in Table 90.1: SOLID90 Element Output Definitions (p. 349)

Convection heat flux is positive out of the element; applied heat flux is positive into the element. The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 90.1: SOLID90 Element Output Definitions

Label	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	2
HGEN	Heat generations HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P), HG(Q), ..., HG(Z), HG(A), HG(B)	Y	-
TG:X, Y, Z, SUM	Thermal gradient components and vector sum at centroid	Y	Y
TF:X, Y, Z, SUM	Thermal flux (heat flow rate/cross-sectional area) components and vector sum at centroid	Y	Y
FACE	Face label	1	-
NODES	Corner nodes on this face	1	-
AREA	Face area	1	1
HFILM	Film coefficient	1	-
TAVG	Average face temperature	1	1
TBULK	Fluid bulk temperature	1	-
HEAT RATE	Heat flow rate across face by convection	1	1
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	1	-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Label	Definition	O	R
HFLUX	Heat flux at each node of face	1	-
HFAVG	Average film coefficient of the face	-	1
TBAVG	Average face bulk temperature	-	1
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	1

1. Output only if a surface load is input

2. Available only at centroid as a ***GET** item.

Table 90.2: SOLID90 Item and Sequence Numbers (p. 350) lists output available through the **ETABLE** command using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) of this manual for more information. The following notation is used in Table 90.2: SOLID90 Item and Sequence Numbers (p. 350):

Name
output quantity as defined in the Table 90.1: SOLID90 Element Output Definitions (p. 349)

Item
predetermined Item label for **ETABLE** command

FC*n*
sequence number for solution items for element Face *n*

Table 90.2: SOLID90 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input						
	Item	FC1	FC2	FC3	FC4	FC5	FC6
AREA	NMISC	1	7	13	19	25	31
HFAVG	NMISC	2	8	14	20	26	32
TAVG	NMISC	3	9	15	21	27	33
TBAVG	NMISC	4	10	16	22	28	34
HEAT RATE	NMISC	5	11	17	23	29	35
HFLXAVG	NMISC	6	12	18	24	30	36

SOLID90 Assumptions and Restrictions

- The element must not have a zero volume. This occurs most frequently when the element is not numbered properly.
- Elements may be numbered either as shown in Figure 90.1: SOLID90 Geometry (p. 347) or may have the planes IJKL and MNOP interchanged.
- The condensed face of a prism-shaped element should not be defined as a convection face.
- The specific heat and enthalpy are evaluated at each integration point to allow for abrupt changes (such as melting) within a coarse grid of elements.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If the thermal element is to be replaced by a SOLID186 structural element with surface stresses requested, the thermal element should be oriented such that face IJNM and/or face KLPO is a free surface.
- A free surface of the element (i.e., not adjacent to another element and not subjected to a boundary constraint) is assumed to be adiabatic.
- Thermal transients having a fine integration time step and a severe thermal gradient at the surface will also require a fine mesh at the surface.
- An edge with a removed midside node implies that the temperature varies linearly, rather than parabolically, along that edge.
- See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.
- Degeneration to the form of pyramid should be used with caution.
- The element sizes, when degenerated, should be small in order to minimize the field gradients.
- Pyramid elements are best used as filler elements or in meshing transition zones.

SOLID90 Product Restrictions

ANSYS Professional

- No Birth and Death.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

CIRCU94

Piezoelectric Circuit

MP ME <> <> <> <> <> <> PP EME MFS
Product Restrictions

CIRCU94 Element Description

CIRCU94 is a circuit element for use in piezoelectric-circuit analyses. The element has two or three nodes to define the circuit component and one or two degrees of freedom to model the circuit response. The element may interface with the following piezoelectric elements:

PLANE13, KEYOPT(1) = 7 coupled-field quadrilateral solid
SOLID5, KEYOPT(1) = 0 or 3 coupled-field brick
SOLID98, KEYOPT(1) = 0 or 3 coupled-field tetrahedron
PLANE223, KEYOPT(1) = 1001, coupled-field 8-node quadrilateral
SOLID226, KEYOPT(1) = 1001, coupled-field 20-node brick
SOLID227, KEYOPT(1) = 1001, coupled-field 10-node tetrahedron

CIRCU94 is applicable to full harmonic and transient analyses. For these types of analyses, you can also use CIRCU94 as a general circuit element. See CIRCU94 in the *Mechanical APDL Theory Reference* for more details about this element.

CIRCU94 Input Data

The geometry, node definition, and degree of freedom options are shown in Figure 94.1: CIRCU94 Circuit Options (p. 354) . Active nodes I and J define the resistor, inductor, capacitor and independent current source. They are connected to the electric circuit. Active nodes I and J and a passive node K define the independent voltage source. The passive node is not connected to the electric circuit. It is associated with the CURR degree of freedom (which represents electric charge for this element).

KEYOPT(1) settings and the corresponding real constants define the circuit components. Real constant input is dependent on the element circuit option used. A summary of the element input options is given in "CIRCU94 Input Summary" (p. 354). Real constants 15 (Graphical offset, GOFFST) and 16 (Element identification number, ID) are created for all components.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 94.1: CIRCU94 Circuit Options

The independent current and voltage sources (KEYOPT(1) = 3 or 4) may be excited by constant load (transient) or constant amplitude load (harmonic), sinusoidal, pulse, exponential, or piecewise linear load functions as defined by KEYOPT(2); see Figure 94.2: Load Functions and Corresponding Real Constants for Independent Current and Voltage Sources (p. 357).

The time-step size for a transient analysis is controlled by the **DELTIM** or **NSUBST** commands. The CIRCU94 element does not respond to automatic time stepping (**AUTOTS** command), but **AUTOTS** can be used as a mechanism for ramping the time step to its final value.

CIRCU94 is only compatible with elements having a VOLT DOF and an electric charge reaction solution. Electric charge reactions must all be positive or negative. KEYOPT(6) sets the electric charge reaction sign. See Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide* for more information.

CIRCU94 Input Summary

Nodes

I, J, K

Degrees of Freedom

VOLT, CURR (charge) (see Figure 94.1: CIRCU94 Circuit Options (p. 354))

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Real Constants

Dependent on KEYOPT(1) and KEYOPT(2) settings. See Table 94.1: CIRCU94 Real Constants (p. 356) for details.

Material Properties

None

Surface Loads

None

Body Loads

See KEYOPT(2)

Special Features

This element works with the large deflection and stress stiffening capabilities of PLANE13, SOLID5, SOLID98, PLANE223, SOLID226, and SOLID227.

KEYOPT(1)

Circuit component type:

0 --
Resistor

1 --
Inductor

2 --
Capacitor

3 --
Independent Current Source

4 --
Independent Voltage Source

KEYOPT(2)

Body loads (only used for KEYOPT(1) = 3 and 4):

0 --
Constant load (transient) or constant amplitude load (harmonic)

1 --
Sinusoidal load

2 --
Pulse load

3 --
Exponential load

4 --
Piecewise Linear load

KEYOPT(6)

Electric charge reaction sign:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0 --
Negative

1 --
Positive

Table 94.1: CIRCU94 Real Constants

Circuit Component and Graphics Label	KEYOPT(1)	Real Constants
Resistor (R)	0	R1 = Resistance (RES)
Inductor (L)	1	R1 = Inductance (IND) R2 = Initial inductor current (ILO)
Capacitor (C)	2	R1 = Capacitance (CAP) R2 = Initial Capacitor Voltage (VCO)
Independent Current Source (I)	3	For KEYOPT(2) = 0: R1 = Amplitude (AMPL) R2 = Phase angle (PHAS) For KEYOPT(2) > 0, see Figure 94.2: Load Functions and Corresponding Real Constants for Independent Current and Voltage Sources (p. 357).
Independent Voltage Source (V)	4	For KEYOPT(2) = 0: R1 = Amplitude (AMPL) R2 = Phase angle (PHAS) For KEYOPT(2) > 0, see Figure 94.2: Load Functions and Corresponding Real Constants for Independent Current and Voltage Sources (p. 357).

Note

For all above Circuit options, the GOFFST and ID real constants (numbers 15 and 16) are created by the Circuit Builder automatically:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 94.2: Load Functions and Corresponding Real Constants for Independent Current and Voltage Sources

(a) Sinusoidal load, KEYOPT(2) = 1

(b) Pulse load, KEYOPT(2) = 2

(c) Exponential load, KEYOPT(2) = 3

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(d) Piecewise linear load, KEYOPT(2) = 4

CIRCU94 Output Data

The element output for this element is dependent on the circuit option selected. Table 94.2: CIRCU94 Element Output Definitions (p. 358) summarizes the element output data.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 94.2: CIRCU94 Element Output Definitions

Name	Definition	O	R
For KEYOPT(1) = 0: Resistor			
EL	Element Number	Y	Y
NODES	Nodes-I,J	Y	Y
RES	Resistance	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current	Y	Y
POWER	Power loss	Y	Y
For KEYOPT(1) = 1: Inductor			
EL	Element Number	Y	Y
NODES	Nodes-I,J	Y	Y
IND	Inductance	Y	Y
IL0	Initial current	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current	Y	Y
POWER	Power absorption	Y	Y
For KEYOPT(1) = 2: Capacitor			

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes-I,J	Y	Y
CAP	Capacitance	Y	Y
VC0	Initial voltage	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current	Y	Y
POWER	Power absorption	Y	Y
For KEYOPT(1) = 3: Independent Current Source			
EL	Element Number	Y	Y
NODES	Nodes-I,J	Y	Y
CURRENT SOURCE	Real or imaginary component of applied current	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current	Y	Y
POWER	Power (loss if positive, output if negative)	Y	Y
For KEYOPT(1) = 4: Independent Voltage Source			
EL	Element Number	Y	Y
NODES	Nodes-I,J,K	Y	Y
VOLTAGE SOURCE	Real or imaginary component of applied voltage	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current at node K	Y	Y
POWER	Power (loss if positive, output if negative)	Y	Y

Table 94.3: CIRCU94 Item and Sequence Numbers (p. 359) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 94.3: CIRCU94 Item and Sequence Numbers (p. 359):

Name
output quantity as defined in Table 94.2: CIRCU94 Element Output Definitions (p. 358)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 94.3: CIRCU94 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
VOLTAGE	SMISC	1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
CURRENT	SMISC	2
POWER	NMISC	1
SOURCE (real)	NMISC	2
SOURCE (imaginary)	NMISC	3

CIRCU94 Assumptions and Restrictions

- CIRCU94 is applicable only to full harmonic and transient analyses. You cannot use CIRCU94 in a static analysis or in a transient analysis with time integration effects turned off (**TIMINT**,OFF).
- Only MKS units are allowed (**EMUNIT** command).
- Only the sparse solver is available for problems using the independent voltage source circuit option.
- This element may not be compatible with other elements with the VOLT degree of freedom. For example, it is not compatible with CIRCU124 or CIRCU125. To be compatible, the elements must have the same reaction solution for the VOLT DOF. Elements that have an electric charge reaction solution must all have the same electric charge reaction sign. For more information, see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*.

CIRCU94 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID96

3-D Magnetic Scalar Solid

MP <> <> <> <> <> EM <> PP EME <>
Product Restrictions

SOLID96 Element Description

SOLID96 has the capability of modeling 3-D magnetic fields. Scalar potential formulations (reduced (RSP), difference (DSP), or general (GSP)) are available [**MAGOPT**] for modeling magnetic fields in a static analysis. See SOLID96 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 96.1: SOLID96 Geometry

SOLID96 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 96.1: SOLID96 Geometry (p. 361). The element is defined by eight nodes and the material properties. A tetrahedral-shaped element may be formed by defining the same node numbers for nodes M, N, O, and P; and nodes K and L. A wedge-shaped element and a pyramid-shaped element may also be formed as shown in Figure 96.1: SOLID96 Geometry (p. 361). The type of units (MKS or user defined) is specified through the **EMUNIT** command. **EMUNIT** also determines the value of MUZRO. The **EMUNIT** defaults are MKS units and MUZRO = $4\pi \times 10^{-7}$ henries/meter. In addition to MUZRO, orthotropic relative permeability is available and is specified through the MURX, MURY, and MURZ material options.

MGXX, MGYY, and MGZZ represent vector components of the coercive force for permanent magnet materials. The magnitude of the coercive force is the square root of the sum of the squares of the components. The direction of polarization is determined by the components MGXX, MGYY, and MGZZ. Permanent magnet polarization directions correspond to the element coordinate directions. The element

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

coordinate system orientation is as described in Coordinate Systems (p. 55). Nonlinear magnetic B-H properties are entered with the **TB** command as described in Material Models in the *Material Reference*. Nonlinear orthotropic magnetic properties may be specified with a combination of a B-H curve and linear relative permeability. The B-H curve will be used in each element coordinate direction where a zero value of relative permeability is specified. Only one B-H curve may be specified per material.

Nodal loads are defined with the **D** and the **F** commands. With the **D** command, the *Lab* variable corresponds to the degree of freedom (MAG) and *VALUE* corresponds to the value (magnetic scalar potential). With the **F** command, the *Lab* variable corresponds to the force (FLUX) and *VALUE* corresponds to the value (magnetic flux).

Element loads are described in Nodal Loading (p. 49). Maxwell force flags may be input on the element faces indicated by the circled numbers in Figure 96.1: SOLID96 Geometry (p. 361) using the **SF** and **SFE** commands. Surfaces at which magnetic forces are to be calculated may be identified by using the MXWF label on the surface load commands (no value is required.) A maxwell stress tensor calculation is performed at these surfaces to obtain the magnetic forces. The surface flag should be applied to "air" elements adjacent to the body for which forces are required. Deleting the MXWF specification removes the flag. Maxwell forces may be made available for a subsequent structural analysis with companion elements (**LDREAD** command).

The temperature (for material property evaluation only) and magnetic virtual displacement body loads may be input based on their value at the element's nodes or as a single element value [**BF** and **BFE**]. In general, unspecified nodal values of temperature default to the uniform value specified with the **BFUNIF** or **TUNIF** commands.

Air elements in which Local Jacobian forces are to be calculated may be identified by using nodal values of 1 and 0 for the MVDI label [**BF**]. See the *Low-Frequency Electromagnetic Analysis Guide* for details.

Current for the magnetic scalar potential options are defined with the SOURC36 element, the command macro RACE, or through electromagnetic coupling. The various types of magnetic scalar potential solution options are defined with the **MAGOPT** command.

A summary of the element input is given in "SOLID96 Input Summary" (p. 362). A general description of element input is given in Element Input (p. 43).

SOLID96 Input Summary

Nodes

I, J, K, L, M, N, O, P

Degrees of Freedom

MAG

Real Constants

None

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.

MP command: MUZERO, MURX, MURY, MURZ, MGXX, MGYY, MGZZ

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface Loads

Maxwell Force Flags --
face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N), face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T (M), T(N), T(O), T(P)

MVDI --
VD(I), VD(J), VD(K), VD(L), VD(M), VD(N), VD(O), VD(P)

EF --
EFX, EFY, EFZ. See "SOLID96 Assumptions and Restrictions" (p. 365).

Special Features

Adaptive descent
Birth and death

KEYOPT(5)
Extra element output:

0 --
Basic element printout

1 --
Integration point printout

2 --
Nodal magnetic field printout

SOLID96 Output Data

The solution output associated with the element is in two forms:

- Nodal potentials included in the overall nodal solution
- Additional element output as shown in Table 96.1: SOLID96 Element Output Definitions (p. 364)

The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 96.1: SOLID96 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	2
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	Y	Y
LOC	Output location (X, Y, Z)	1	-
MUX, MUY, MUZ	Magnetic permeability	1	1
H:X, Y, Z	Magnetic field intensity components	1	1
H:SUM	Vector magnitude of H	1	1
B:X, Y, Z	Magnetic flux density components	1	1
B:SUM	Vector magnitude of B	1	1
FMX	Maxwell magnetic force components (X, Y, Z)	1	-
FVW	Virtual work force components (X, Y, Z)	1	1
Combined (FJB or FMX) force components	Combined force components	-	1

1. The solution value is printed only if calculated (based upon input data). The element solution is at the centroid.

2. Available only at centroid as a ***GET** item.

Table 96.2: SOLID96 Miscellaneous Element Output

Description	Names of Items Output	O	R
Integration Point Solution	LOC, MUX, MUY, MUZ, H, HSUM, B, BSUM	1	-
Nodal Magnetic Field Solution	H, HSUM, B, BSUM	2	-

1. Output at each integration point, if KEYOPT(5) = 1

2. Output at each corner node, if KEYOPT(5) = 2

Table 96.3: SOLID96 Item and Sequence Numbers (p. 365) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 96.3: SOLID96 Item and Sequence Numbers (p. 365):

Name

output quantity as defined in Table 96.1: SOLID96 Element Output Definitions (p. 364)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Item

predetermined Item label for **ETABLE** command

E

sequence number for single-valued or constant element data

Table 96.3: SOLID96 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
MUX	NMISC	1
MUY	NMISC	2
MUZ	NMISC	3
FVWX	NMISC	4
FVWY	NMISC	5
FVWZ	NMISC	6
FVWSUM	NMISC	7

SOLID96 Assumptions and Restrictions

- The element requires an iterative solution if nonlinear material properties are defined.
- When using SOLID96 with SOURC36 elements, the source elements must be placed so that the resulting Hs field fulfills boundary conditions for the total field.
- The element must not have a zero volume or a zero length side. This occurs most frequently when the element is not numbered properly. Elements may be numbered either as shown in Figure 96.1: SOLID96 Geometry (p. 361) or may have the planes IJKL and MNOP interchanged.
- The difference magnetic scalar potential option is restricted to singly-connected permeable regions, so that as $\mu \rightarrow \infty$ in these regions, the resulting field $H \rightarrow 0$. The reduced scalar, and general scalar potential options do not have this restriction.
- Degeneration to the form of pyramid should be used with caution. The element sizes, when degenerated, should be small in order to minimize the field gradients. Pyramid elements are best used as filler elements or in meshing transition zones.
- The solenoidal current density is required for a solution, or for any postprocessing operations.
- The electric field body load is not used during solution and is applicable only to POST1 charged particle tracing.
- In an MSP analysis, avoid using a closed domain and use an open domain, closed with natural flux parallel boundary conditions on the MAG degree of freedom, or infinite elements. If you use a closed domain, you may see incorrect results when the formulation is applied using SOLID5, SOLID96, or SOLID98 elements and the boundary conditions are not satisfied by the Hs field load calculated by the Biot-Savart procedure based on SOURC36 current source primitive input.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID96 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Emag

- The birth and death special feature is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID97

3-D Magnetic Solid

MP <> <> <> <> <> EM <> PP EME <>
Product Restrictions

SOLID97 Element Description

SOLID97 models 3-D magnetic fields. The element is defined by eight nodes, and has up to five degrees of freedom per node out of six defined DOFs; that is, the magnetic vector potential (AX, AY, AZ), the time-integrated electric potential (VOLT - classical formulation) or the electric potential (VOLT - solenoidal formulation), the electric current (CURR), and the electromotive force (EMF). SOLID97 is based on the magnetic vector potential formulation with the Coulomb gauge, and is applicable to the following low-frequency magnetic field analyses: magnetostatics, eddy currents (AC time harmonic and transient analyses), voltage forced magnetic fields (static, AC time harmonic and transient analyses), and electromagnetic-circuit coupled fields (static, AC time harmonic and transient analyses). The element has nonlinear magnetic capability for modeling B-H curves or permanent magnet demagnetization curves. See SOLID97 in the *Mechanical APDL Theory Reference* for details about this element. Elements with similar capability are PLANE53, (but without voltage forced and magnetic-circuit coupled capability), SOLID236, and SOLID237.

Degree of freedom and force labels for this element are shown in the following table.

Table 97.1: DOF and Force Labels

Degree of Freedom		Force (reaction solution)	
Label	**Label Description**	**Label**	**Label Description**
AX, AY, AZ [1]	Magnetic Vector Potential	CSGX, CSGY, CSGZ [2]	Magnetic Current Segment
VOLT	Voltage	AMPS	Current Flow
CURR	Current Flow	VLTG	Voltage Drop
EMF	Electromotive Force	CURT	Current Flow

1. The label used with the CNVTOL command is A.

2. The label used with the CNVTOL command is CSG.

Formulations

SOLID97 has two formulation options: classical and solenoidal. The classical formulation requires you to specify current density for current source loading. You must ensure that solenoidal conditions (div J = 0) are satisfied, otherwise an erroneous solution might develop. The solenoidal formulation automatically satisfies the solenoidal condition by directly solving for current (density) using a coupled current conduction and electromagnetic field solution. The solenoidal formulation is applicable to sources that are eddy current free (such as stranded coils).

Note

Both formulations may be used simultaneously, depending on the physics requirements of the model. See 3-D Nodal-Based Analyses (Static, Harmonic, and Transient) in the *Low-Fre-*

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

quency Electromagnetic Analysis Guide for information on applying these formulations to different physics regions of a model.

The SOLID97 solenoidal option is compatible with CIRCU124, CIRCU125, and TRANS126 elements allowing circuit coupling. The nonlinear symmetric solenoidal formulation is applicable to static and transient analyses. The linear unsymmetric solenoidal formulation is applicable to harmonic analysis. For more information, see 3-D Circuit Coupled Solid Source Conductor in the *Coupled-Field Analysis Guide*

Figure 97.1: SOLID97 Geometry

SOLID97 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 97.1: SOLID97 Geometry (p. 368). The element is defined by eight nodes and the material properties. A tetrahedral-shaped element may be formed by defining the same node numbers for nodes M, N, O, and P; and nodes K and L. A wedge-shaped element and a pyramid-shaped element may also be formed as shown in Figure 97.1: SOLID97 Geometry (p. 368). The type of units (MKS or user defined) is specified through the **EMUNIT** command. **EMUNIT** also determines the value of MUZRO. The **EMUNIT** defaults are MKS units and MUZRO = $4\pi \times 10^{-7}$ henries/meter. In addition to MUZRO, orthotropic relative permeability is available and is specified through the MURX, MURY, and MURZ material options. Orthotropic resistivity is specified through RSVX, RSVY, and RSVZ material property labels.

MGXX, MGYY, and MGZZ represent vector components of the coercive force for permanent magnet materials. The magnitude of the coercive force is the square root of the sum of the squares of the components. The direction of polarization is determined by the components MGXX, MGYY, and MGZZ. Permanent magnet polarization directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems (p. 55). Nonlinear magnetic B-H properties are entered with the **TB** command as described in Material Models in the *Material Reference*. Nonlinear orthotropic magnetic properties may be specified with a combination of a B-H curve and

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

linear relative permeability. The B-H curve will be used in each element coordinate direction where a zero value of relative permeability is specified. Only one B-H curve may be specified per material.

When SOLID97 is used for voltage forced or magnetic-circuit coupled analyses, the following real constants apply for coils or massive conductors:

CARE
Coil cross-sectional area.

TURN
Total number of coil turns, required for stranded coil only. Defaults to 1.

VOLU
Modeled coil volume, required for stranded coil only.

DIRX, DIRY, DIRZ
x, y, and z components of a unit vector (in the element coordinate system) representing the direction of current. Required for a stranded coil only.

CSYM
Coil symmetry factor: CSYM*VOLU = total volume of the coil. Required for stranded coil only. Defaults to 1.

FILL
Coil fill factor, required for stranded coil only. Defaults to 1.

When velocity effects of a conducting body (KEYOPT(2) = 1) are considered, the following real constants apply:

VELOX, VELOY, VELOZ
Velocity components in the Global Cartesian Coordinate system X, Y, and Z direction respectively.

OMEGAX, OMEGAY, OMEGAZ
Angular (rotational) velocity (Hz, cycles/sec) about the Global Cartesian system X, Y, and Z-axes respectively, located at the pivot point location (XLOC, YLOC, ZLOC).

XLOC, YLOC, ZLOC
Global Cartesian coordinate point locations of the rotating body in the X, Y, and Z directions respectively.

Nodal loads are defined with the **D** and the **F** commands. With the **D** command, the *Lab* variable corresponds to the degree of freedom (A_ and VOLT) and *VALUE* corresponds to the value (vector magnetic potential or the time-integrated electric potential (classical formulation) or the electric potential (solenoidal formulation). The electric potential may or may not be time integrated depending on the KEYOPT(1) selection. With the **F** command, the *Lab* variable corresponds to the force (CSG or Amps) and *VALUE* corresponds to the value (magnetic current segment or current).

Element loads are described in Nodal Loading (p. 49). Maxwell force flags may be input on the element faces indicated by the circled numbers in Figure 97.1: SOLID97 Geometry (p. 368) using the **SF** and **SFE** commands. Surfaces at which magnetic forces are to be calculated may be identified by using the MXWF label on the surface load commands (no value is required.) A Maxwell stress tensor calculation is performed at these surfaces to obtain the magnetic forces. The surface flag should be applied to "air" elements adjacent to the body for which forces are required. Deleting the MXWF specification removes the flag. Lorentz and Maxwell forces may be made available for a subsequent structural analysis with companion elements [**LDREAD**].

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The temperature (for material property evaluation only) and magnetic virtual displacement body loads may be input based on their value at the element's nodes or as a single element value [**BF** and **BFE**]. Source current density (classical formulation) and voltage body loads may be applied to an area or volume [**BFA** or **BFV**] or input as an element value [**BFE**]. In general, unspecified nodal values of temperatures default to the uniform value specified with the **BFUNIF** or **TUNIF** commands. The vector components of the current density are with respect to the element coordinate system (see "SOLID97 Assumptions and Restrictions" (p. 376) for solenoidal restriction). Joule heating may be made available for a subsequent thermal analysis with companion elements [**LDREAD**].

Air elements in which Local Jacobian forces are to be calculated may be identified by using nodal values of 1 and 0 for the MVDI label [**BF**]. See the *Low-Frequency Electromagnetic Analysis Guide* for details.

A summary of the element input is given in "SOLID97 Input Summary" (p. 370). A general description of element input is given in Element Input (p. 43).

SOLID97 Input Summary

Nodes

I, J, K, L, M, N, O, P

Degrees of Freedom

See KEYOPT(1)

Real Constants

None if KEYOPT(1) = 0 and KEYOPT(2) = 0.

For KEYOPT(1) = 2, 3, 5, or 6, and KEYOPT(2) = 0:

CARE, TURN, VOLU, DIRX, DIRY, DIRZ,
CSYM, FILL

For KEYOPT(1) = 4 and KEYOPT(2) = 0:

CARE - Coil cross-sectional area

For KEYOPT(1) = 0 or 1 and KEYOPT(2) = 1:

(blank), (blank), (blank), (blank), (blank), (blank),
(blank), (blank), VELOX, VELOY, VELOZ, OMEGAX,
OMEGAY OMEGAZ, XLOC, YLOC, ZLOC

See Table 97.2: SOLID97 Real Constants (p. 372) for a description of the real constants.

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.

MP command: MURX, MURY, MURZ, RSVX, RSVY, RSVZ, MGXX, MGYY, MGZZ

EMUNIT command: MUZERO

Surface Loads

Maxwell Force Flags --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Magnetic-Circuit Interface Flags, if KEYOPT(1) = 4:

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

MVDI --

VD(I), VD(J), VD(K), VD(L), VD(M), VD(N),
VD(O), VD(P)

Source Current Density, if KEYOPT(1) = 0 (See "SOLID97 Assumptions and Restrictions" (p. 376) for solenoidal restriction) --

JSX(I), JSY(I), JSZ(I), PHASE(I), JSX(J), JSY(J), JSZ(J), PHASE(J), JSX(K), JSY(K), JSZ(K), PHASE(K),
JSX(L), JSY(L), JSZ(L), PHASE(L), JSX(M), JSY(M), JSZ(M), PHASE(M), JSX(N), JSY(N), JSZ(N),
PHASE(N),
JSX(O), JSY(O), JSZ(O), PHASE(O), JSX(P), JSY(P), JSZ(P), PHASE(P)

Voltage Loading, if KEYOPT(1) = 2 --

VLTG(I), PHASE(I), VLTG(J), PHASE(J), VLTG(K), PHASE(K), VLTG(L), PHASE(L),
VLTG(M), PHASE(M), VLTG(N), PHASE(N), VLTG(O), PHASE(O), VLTG(P), PHASE(P)

EF --
EFX, EFY, EFZ. See "SOLID97 Assumptions and Restrictions" (p. 376).

Special Features

Adaptive descent
Birth and death

KEYOPT(1)
Element degrees of freedom and formulation selection:

Classical Formulation

0 --
AX, AY, AZ degrees of freedom: static domain, source domain

1 --
AX, AY, AZ, VOLT degrees of freedom: eddy current domain, velocity effect domain

2 --
AX, AY, AZ, CURR degrees of freedom: voltage-fed stranded coil

3 --
AX, AY, AZ, CURR, EMF degrees of freedom: circuit-coupled stranded coil

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4 --
AX, AY, AZ, VOLT, CURR degrees of freedom: circuit-coupled massive conductor

Solenoidal Formulation

5 --
AX, AY, AZ, VOLT degrees of freedom: nonlinear symmetric solenoidal formulation applicable to static and transient analyses

6 --
AX, AY, AZ, VOLT degrees of freedom: linear unsymmetric solenoidal formulation applicable to harmonic analyses

---

Note

For KEYOPT(1) = 1 and 4, the VOLT degree of freedom is time integrated (classical formulation). For KEYOPT(1) = 5 and 6, the VOLT degree of freedom is not time integrated (solenoidal formulation).

KEYOPT(2)
Element conventional velocity:

0 --
Velocity effects ignored

1 --
Conventional velocity formulation (not available if KEYOPT(1) = 2, 3, or 4)

KEYOPT(5)
Extra element output:

0 --
Basic element printout

1 --
Integration point printout

2 --
Nodal magnetic field printout

Table 97.2: SOLID97 Real Constants

No.	Name	Description
KEYOPT(1) = 2, 3, 5, 6 and KEYOPT(2) = 0		
1	CARE	Coil cross-sectional area
2	TURN	Total number of coil turns
3	VOLU	Modeled coil volume
4	DIRX	Directional cosine for X component of the current
5	DIRY	Directional cosine for Y component of the current
6	DIRZ	Directional cosine for Z component of the current
7	CSYM	Coil symmetry factor

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

No.	Name	Description
8	FILL	Coil fill factor *
KEYOPT(1) = 0 or 1 and KEYOPT(2) = 1		
1, ..., 8	(blank)	Unused for these settings
9, 10, 11	VELOX, VELOY, VELOZ	Velocity specification in the x, y, and z directions
12, 13, 14	OMEGAX, OMEGAY, OMEGAZ	Angular velocity about the X, Y, and Z axes
15, 16, 17	XLOC, YLOC, ZLOC	Pivot point location (x, y, z)

* The ratio between the actual conductor area and the area of the coil or element geometry. Often, the element approximation will differ from that of the conductor, especially in multi-strand coils.

SOLID97 Output Data

The solution output associated with the element is in two forms:

- Nodal potentials included in the overall nodal solution
- Additional element output as shown in Table 97.3: SOLID97 Element Output Definitions (p. 373)

The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 97.3: SOLID97 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	2
TEMP	Input temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	Y	Y
LOC	Output location (X, Y, Z)	1	-
MUX, MUY, MUZ	Magnetic secant permeability	1	1
H:X, Y, Z	Magnetic field intensity components	1	1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
H:SUM	Vector magnitude of H	1	1
B:X, Y, Z	Magnetic flux density components	1	1
B:SUM	Vector magnitude of B	1	1
JS:X, Y, Z	Source current density components in the global Cartesian coordinate system, valid for static analysis only	1	1
JT(X, Y, Z)	Total current density components in the global Cartesian coordinate system.	1	1
JHEAT:	Joule heat generation per unit volume	1	1
FJB(X, Y, Z)	Lorentz magnetic force components	1	-
FMX(X, Y, Z)	Maxwell magnetic force components	1	-
FVW(X, Y, Z)	Virtual work force components	1	1
Combined (FJB or FMX) force components	Combined (FJB or FMX) force components	-	1
ERES	Element resistance value (for stranded coils only)	-	1
EIND	Element inductance value (for stranded coils only)	-	1
DMUXX, DMUYY, DMUZZ	Differential permeability	1	1
V:X, Y, Z	Velocity components	1	1
V:SUM	Vector magnitude of V	1	-
MRE	Magnetic Reynolds number	1	1

1. The solution value is output only if calculated (based upon input data). The element solution is at the centroid.

Note

JT represents the total measurable current density in a conductor, including eddy current effects, and velocity effects if calculated.

For harmonic analysis, Joule losses (JHEAT) and forces (FJB(X, Y, Z), FMX(X, Y, Z), FVW(X, Y, Z)) represent time-average values. These values are stored in both the "Real" and “Imaginary” data sets. The macros **POWERH** and **FMAGSUM** can be used to retrieve this data.

Inductance values (EIND) obtained for KEYOPT(1) = 2, 3, or 4 are only valid under the following conditions: the problem is linear (constant permeability), there are no permanent magnets in the model, and only a single coil exists in the model.

For KEYOPT(1)=2 and 3, JT represents the effective current density (including non-conducting material represented by the FILL factor). JHEAT reprsents the effective Joule heat generation rate (including non-conducting material represented by the FILL factor).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Available only at centroid as a ***GET** item.

Table 97.4: SOLID97 Miscellaneous Element Output

Description	Names of Items Output	O	R
Integration Point Solution	LOC, MUX, MUY, MUZ, H, HSUM, B, BSUM	1	-
Nodal Magnetic Field Solution	H, HSUM, B, BSUM	2	-

1. Output at each integration point, if (KEYOPT(5) = 1)

2. Output at each corner node, if (KEYOPT(5) = 2)

Table 97.5: SOLID97 Item and Sequence Numbers (p. 375) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 97.5: SOLID97 Item and Sequence Numbers (p. 375):

Name
output quantity as defined in Table 97.3: SOLID97 Element Output Definitions (p. 373)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 97.5: SOLID97 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
JSX	SMISC	1
JSY	SMISC	2
JSZ	SMISC	3
JSSUM	SMISC	4
MUX	NMISC	1
MUY	NMISC	2
MUZ	NMISC	3
FVWX	NMISC	4
FVWY	NMISC	5
FVWZ	NMISC	6
FVWSUM	NMISC	7
JTX	NMISC	12
JTY	NMISC	13
JTZ	NMISC	14
JTSUM	NMISC	15

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
ERES	NMISC	16
EIND	NMISC	17
DMUXX	NMISC	18
DMUYY	NMISC	19
DMUZZ	NMISC	20
VX	NMISC	21
VY	NMISC	22
VZ	NMISC	23
MRE	NMISC	28

SOLID97 Assumptions and Restrictions

- The element requires an iterative solution if nonlinear material properties are defined.
- The element must not have a zero volume or a zero length side. This occurs most frequently when the element is not numbered properly. Elements may be numbered either as shown in Figure 97.1: SOLID97 Geometry (p. 368) or may have the planes IJKL and MNOP interchanged.
- The continuity equation must be satisfied for a proper electromagnetic analysis as explained in the *Mechanical APDL Theory Reference*. For this reason the source current density, JS, must be solenoidal (that is, $\nabla \cdot JS = 0$). You should verify that this condition is satisfied when prescribing the source current density load. If this condition is not satisfied SOLID97 can produce erroneous solutions without warning. Refer to Source Current Density (JS) in the *Low-Frequency Electromagnetic Analysis Guide* for information on how to obtain solenoidal currents when the source current density is not constant. To have ANSYS compute the current density for voltage or circuit coupled problems, apply the solenoidal formulation (KEYOPT(1) = 5 or 6).
- For models containing materials with different permeabilities, the 3-D nodal-based vector potential formulation (either static or time-dependent) is not recommended. The solution has been found to be inaccurate when the normal component of the vector potential is significant at the interface between elements of different permeability. To obtain the normal component of the vector potential in postprocessing, issue **PLVECT**,A or **PRVECT**,A in a rotated coordinate system [**RSYS**] that orients one of the vector potential components normal to the material interface.
- Current density loading (**BFE**,,JS) is only valid for the AX, AY, AZ option (KEYOPT(1) = 0). For the AX, AY, AZ, VOLT option (KEYOPT(1) = 1, 5, or 6) use **F**,,AMPS. Solenoidal loading is recommended. For more information, see 3-D Magnetostatics and Fundamentals of Edge-Based Analysis and 3-D Nodal-Based Analyses (Static, Harmonic, and Transient) in the *Low-Frequency Electromagnetic Analysis Guide*.
- The solenoidal current density is required for a solution, or for any postprocessing operations.
- When this element does not have the VOLT degree of freedom, for a harmonic or transient analysis, it acts as a stranded conductor.
- Permanent magnets are not permitted in a harmonic analysis.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You cannot use this element in a nonlinear harmonic analysis.
- The VOLT degree of freedom (KEYOPT(1) = 1) is required in all non-source regions with a specified non-zero resistivity. This allows eddy currents to be computed.
- For source conducting regions (RSVX $\neq$ 0), current loading should be applied as nodal loads (AMPS) using the solenoidal formulation. Current density loading (JS) is allowed (classical formulation), but solenoidal loading is recommended. Node coupling of the VOLT DOF may be required at symmetry planes and locations where the current is applied.
- The ANSYS product does not support the analysis of coupled velocity and circuit effects.
- For voltage forced magnetic field (KEYOPT(1) = 2) and circuit coupled problems (KEYOPT(1) = 3,4), note the following additional restrictions:
  - Only MKS units are allowed.
  - The permeability and conductivity are isotropic and constant.
  - The element coordinate system is used for specifying the current direction vector (DIRX, DIRY, DIRZ) for a stranded coil. Also, the cross sectional area of the stranded coil should not change.
  - For (KEYOPT(1) = 2 or 3), all CURR degrees of freedom in a coil region must be coupled (**CP** command), and all EMF degrees of freedom in a coil region must be coupled.
  - For (KEYOPT(1) = 4), all CURR degrees of freedom on the input face and output face of a massive conductor must be coupled.
- For circuit coupled transient analyses, use THETA = 1.0, the default value, on the **TINTP** command to specify the backward Euler method. For more information, refer to the *Mechanical APDL Theory Reference*, as well as the description of the **TINTP** command in the *Command Reference*.
- For velocity effects (KEYOPT(2) = 1), note the following restrictions:
  - Velocity effects are valid only for the AX, AY, AZ, VOLT DOF option.
  - Velocity effects cannot be included in a static analysis. To simulate a static analysis, execute a harmonic analysis at a very low frequency and retrieve the "real" results for the solution.
  - Velocity effects are available only in a linear analysis.
  - Isotropic resistivity.
  - Solution accuracy may degrade if the element magnetic Reynolds number is much greater than 1.0. (See the discussion of magnetic fields in the *Low-Frequency Electromagnetic Analysis Guide*.)
- If (KEYOPT(1) = 2, 3,4, or 6) or (KEYOPT(2) $\geq$ 1), unsymmetric matrices are produced.
- Degeneration to the form of pyramid should be used with caution. The element sizes, when degenerated, should be small in order to minimize the field gradients. Pyramid elements are best used as filler elements or in meshing transition zones.
- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, the elements must have the same reaction force (see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The solenoidal formulations do not model eddy current effects.
- The electric field body load is not used during solution and is applicable only to POST1 charged particle tracing.

SOLID97 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Emag

- The birth and death special feature is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID98

Tetrahedral Coupled-Field Solid

MP ME <> <> <> <> EM <> PP EME MFS
Product Restrictions

SOLID98 Element Description

Although this legacy element is available for use in your analysis, ANSYS, Inc. recommends using a current-technology element such as SOLID227.

SOLID98 is a 10-node tetrahedral version of the 8-node SOLID5 element. The element has a quadratic displacement behavior and is well suited to model irregular meshes (such as produced from various CAD/CAM systems). When used in structural and piezoelectric analyses, SOLID98 has large deflection and stress stiffening capabilities.

The element is defined by ten nodes with up to six degrees of freedom at each node (see KEYOPT(1)). See SOLID98 in the *Mechanical APDL Theory Reference* for more details about this element. The 3-D magnetic, thermal, electric, piezoelectric, and structural field capability is similar to that described for SOLID5.

Figure 98.1: SOLID98 Geometry

SOLID98 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 98.1: SOLID98 Geometry (p. 379). The element input data is essentially the same as for SOLID5 except that there are 10 nodes instead of 8.

Various combinations of nodal loading are available for this element (depending upon the KEYOPT(1) value). Nodal loads are defined with the **D** and the **F** commands. With the **D** command, the *Lab* variable corresponds to the degree of freedom (UX, UY, UZ, TEMP, VOLT, MAG) and *VALUE* corresponds to the value (displacements, temperature, voltage, scalar magnetic potential). With the **F** command, the *Lab* variable corresponds to the force (FX, FY, FZ, HEAT, AMPS, FLUX) and *VALUE* corresponds to the value (force, heat flow, current or charge, magnetic flux). Nonlinear magnetic B-H, piezoelectric, and anisotropic elastic properties are entered with the **TB** command as described in Material Models in the *Material Reference*. Nonlinear orthotropic magnetic properties may be specified with a combination of a B-H

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

curve and linear relative permeability. The B-H curve will be used in each element coordinate direction where a zero value of relative permeability is specified. Only one B-H curve may be specified per material.

Element loads are described in Nodal Loading (p. 49). Pressure, convection or heat flux (but not both), radiation, and Maxwell force flags may be input on the element faces indicated by the circled numbers in Figure 98.1: SOLID98 Geometry (p. 379) using the **SF** and **SFE** commands. Positive pressures act into the element. Surfaces at which magnetic forces are to be calculated may be identified by using the MXWF label on the surface load commands (no value is required.) A Maxwell stress tensor calculation is performed at these surfaces to obtain the magnetic forces. These forces are applied in solution as structural loads. The surface flag should be applied to "air" elements adjacent to the body for which forces are required. Deleting the MXWF specification removes the flag.

The body loads; temperature, heat generation rate and magnetic virtual displacement may be input based on their value at the element's nodes or as a single element value [**BF** and **BFE**]. When the temperature degree of freedom is active (KEYOPT(1) = 0, 1 or 8), applied body force temperatures [**BF**, **BFE**] are ignored. In general, unspecified nodal values of temperatures and heat generation rate default to the uniform value specified with the **BFUNIF** or **TUNIF** commands. Calculated Joule heating (JHEAT) is applied in subsequent iterations as heat generation rate loading.

If the temperature degree of freedom is present, the calculated temperatures override any input nodal temperatures.

Air elements in which Local Jacobian forces are to be calculated may be identified by using nodal values of 1 and 0 for the MVDI label [**BF**]. See the *Low-Frequency Electromagnetic Analysis Guide* for details. These forces are not applied in solution as structural loads.

A summary of the element input is given in "SOLID98 Input Summary" (p. 380). A general description of element input is given in Element Input (p. 43).

SOLID98 Input Summary

Nodes

I, J, K, L, M, N, O, P, Q, R

Degrees of Freedom

UX, UY, UZ, TEMP, VOLT, MAG if KEYOPT(1) = 0
TEMP, VOLT, MAG if KEYOPT(1) = 1
UX, UY, UZ if KEYOPT(1) = 2
UX, UY, UZ, VOLT if KEYOPT(1) = 3
TEMP if KEYOPT(1) = 8
VOLT if KEYOPT(1) = 9
MAG if KEYOPT(1) = 10

Real Constants

None

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, EY, EZ, (PRXY, PRYZ, PRXZ or NUXY, NUYZ, NUXZ), ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ), DENS, GXY, GYZ, GXZ, ALPD, BETD, KXX, KYY, KZZ, C, NTH, MUZERO, MURX, MURY, MURZ, RSVX, RSVY, RSVZ, E, MGXX, MGYY, MGZZ, PERX, PERY, PERZ

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface Loads

Pressure, Convection or Heat Flux (but not both), Radiation (using Lab = RDSF), and Maxwell Force Flags --
face 1 (J-I-K), face 2 (I-J-L), face 3 (J-K-L), face 4 (K-I-L)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P), T(Q), T(R)

Heat Generations --
HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P), HG(Q), HG(R)

MVDI --
VD(I), VD(J), VD(K), VD(L), VD(M), VD(N), VD(O), VD(P), VD(Q), VD(R)

EF --
EFX, EFY, EFZ. See "SOLID98 Assumptions and Restrictions" (p. 386).

Special Features

Adaptive descent
Birth and death
Large deflection
Stress stiffening

KEYOPT(1)

Degree of freedom selection:

0 --
UX, UY, UZ, TEMP, VOLT, MAG

1 --
TEMP, VOLT, MAG

2 --
UX, UY, UZ

3 --
UX, UY, UZ, VOLT

8 --
TEMP

9 --
VOLT

10 --
MAG

KEYOPT(3)

Specific heat matrix:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0 --
Consistent specific heat matrix

1 --
Diagonalized specific heat matrix

KEYOPT(5)
Extra element output:

0 --
Basic element printout

2 --
Nodal stress or magnetic field printout

SOLID98 Output Data

The solution output associated with the element is in two forms:

- Nodal degree of freedom results included in the overall nodal solution
- Additional element output as shown in Table 98.1: SOLID98 Element Output Definitions (p. 383)

Several items are illustrated in Figure 98.2: SOLID98 Element Output (p. 382). The component output directions are parallel to the element coordinate system. The reaction forces, heat flow, current, and magnetic flux at the nodes can be printed with the **OUTPR** command. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

Figure 98.2: SOLID98 Element Output

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 98.1: SOLID98 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Corner nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	3
PRES	Pressures P1 at nodes J, I, K; P2 at I, J, L; P3 at J, K, L; P4 at K, I, L	Y	Y
TEMP(INPUT)	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P), T(Q), T(R)	Y	Y
HGEN(INPUT)	Heat generations HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P), HG(Q), HG(R)	Y	-
S:X, Y, Z, XY, YZ, XZ	Stresses	1	1
S:1, 2, 3	Principal stresses	1	1
S:INT	Stress intensity	1	1
S:EQV	Equivalent stress	1	1
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	1	1
EPEL:1, 2, 3	Principal elastic strains	1	-
EPEL:EQV	Equivalent elastic strains [4]	1	1
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	1	1
EPTH:EQV	Equivalent thermal strain [4]	1	1
LOC	Output location (X, Y, Z)	1	1
MUX, MUY, MUZ	Magnetic permeability	1	1
H:X, Y, Z	Magnetic field intensity components	1	1
H:SUM	Vector magnitude of H	1	1
B:X, Y, Z	Magnetic flux density components	1	1
B:SUM	Vector magnitude of B	1	1
FJB	Lorentz magnetic force components (X, Y, Z)	1	-
FMX	Maxwell magnetic force components (X, Y, Z)	1	-
FVW	Virtual work force components (X, Y, Z)	1	1
Combined (FJB or FMX) force components	Combined (FJB or FMX) force components	-	1
EF:X, Y, Z	Electric field components	1	1
EF:SUM	Vector magnitude of EF	1	1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
JS:X, Y, Z	Source current density components	1	1
JSSUM	Vector magnitude of JS	1	1
JHEAT:	Joule heat generation per unit volume	1	1
D:X, Y, Z	Electric flux density components	1	1
D:SUM	Vector magnitude of D	1	1
U(E, D, M)	Elastic (UE), dielectric (UD), and electromechanical coupled (UM) energies	1	1
TG:X, Y, Z	Thermal gradient components	1	1
TG:SUM	Vector magnitude of TG	1	1
TF:X, Y, Z	Thermal flux components	1	1
TF:SUM	Vector magnitude of TF (Heat flow rate/unit cross-section area)	1	1
FACE	Face label	2	2
AREA	Face area	2	2
NODES	Face nodes	2	-
HFILM	Film coefficient at each node of face	2	-
TBULK	Bulk temperature at each node of face	2	-
TAVG	Average face temperature	2	2
HEAT RATE	Heat flow rate across face by convection	2	2
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	2	-
HFLUX	Heat flux at each node of face	2	-
HFAVG	Average film coefficient of the face	2	2
TBAVG	Average face bulk temperature	-	2
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	2

1. The solution value is output only if calculated (based upon input data). The element solution is at the centroid.

2. Output only if a surface load is input.

3. Available only at centroid as a ***GET** item.

4. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY).

Table 98.2: SOLID98 Miscellaneous Element Output

Description	Names of Items Output	O	R
Nodal Stress Solution	LOC, SINT, SEQV, EPEL(X, Y, Z, XY, YZ, XZ), EPEL(1, 2, 3), S(X, Y, Z, XY, YZ, XZ), S(1, 2, 3)	1	-
Nodal Magnetic Field Solution	H, HSUM, B, BSUM	2	-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Output at each vertex node, if KEYOPT(5) = 2 and structural DOF

2. Output at each vertex node, if KEYOPT(5) = 2 and magnetic DOF

Table 98.3: SOLID98 Item and Sequence Numbers (p. 385) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 98.3: SOLID98 Item and Sequence Numbers (p. 385):

Name
output quantity as defined in Table 98.1: SOLID98 Element Output Definitions (p. 383)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J,...,L
sequence number for data at nodes I,J,...,L

FC*n* -
sequence number for solution items for element Face *n*

Table 98.3: SOLID98 Item and Sequence Numbers

Output Quant-ity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
P1	SMISC	-	2	1	3	-
P2	SMISC	-	4	5	-	6
P3	SMISC	-	-	7	8	9
P4	SMISC	-	11	-	10	12
MUX	NMISC	1	-	-	-	-
MUY	NMISC	2	-	-	-	-
MUZ	NMISC	3	-	-	-	-
FVWX	NMISC	4	-	-	-	-
FVWY	NMISC	5	-	-	-	-
FVWZ	NMISC	6	-	-	-	-
FVWSUM	NMISC	7	-	-	-	-
UE	NMISC	16	-	-	-	-
UD	NMISC	17	-	-	-	-
UM	NMISC	18	-	-	-	-

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	FC1	FC2	FC3	FC4
AREA	NMISC	19	25	31	37

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	FC1	FC2	FC3	FC4
HFAVG	NMISC	20	26	32	38
TAVG	NMISC	21	27	33	39
TBAVG	NMISC	22	28	34	40
HEAT RATE	NMISC	23	29	35	41
HFLXAVG	NMISC	24	30	36	42

SOLID98 Assumptions and Restrictions

- The element requires an iterative solution for field coupling (displacement, temperature, electric, magnetic, but not piezoelectric)
- When using SOLID98 with SOURC36 elements, the source elements must be placed so that the resulting Hs field fulfills boundary conditions for the total field.
- The element must not have a zero volume. Elements may be numbered either as shown in Figure 98.1: SOLID98 Geometry (p. 379) or may have node L below the IJK plane. in the *Modeling and Meshing Guide*
- An edge with a removed midside node implies that the displacement varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) for more information about the use of midside nodes.
- The difference scalar magnetic potential option is restricted to singly-connected permeable regions, so that as $\mu \rightarrow \infty$ in these regions, the resulting field $H \rightarrow 0$. The reduced scalar and general scalar potential options do not have this restriction.
- Temperatures and heat generation rates, if internally calculated, include any user defined heat generation rates.
- Large-deflection capabilities available for KEYOPT(1) = 2 and 3 are not available for KEYOPT(1) = 0. Stress stiffening is available for KEYOPT(1) = 0, 2, and 3.
- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, the elements must have the same reaction solution for the VOLT DOF. Elements that have an electric charge reaction solution must all have the same electric charge reaction sign. For more information, see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*.
- The electric field body load is not used during solution and is applicable only to POST1 charged particle tracing.
- In an MSP analysis, avoid using a closed domain and use an open domain, closed with natural flux parallel boundary conditions on the MAG degree of freedom, or infinite elements. If you use a closed domain, you may see incorrect results when the formulation is applied using SOLID5, SOLID96, or SOLID98 elements and the boundary conditions are not satisfied by the Hs field load calculated by the Biot-Savart procedure based on SOURC36 current source primitive input.
- If you used the MAG degree of freedom, you cannot restart a job in ANSYS Mechanical using `Jobname.DB` and `Jobname.ESAV` files that were created by ANSYS Multiphysics.
- This element cannot be used in a distributed solution.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID98 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Mechanical

Unless the Emag option is enabled, the following restrictions apply:

- This element does not have magnetic capability.
- The MAG degree of freedom is not active.
- KEYOPT(1) cannot be set to 10. If KEYOPT(1) = 0 (default) or 1, the MAG degree of freedom is inactive.
- KEYOPT(1) cannot be set to 9.
- The magnetic material properties (MUZERO, MUR_, MG__, and the BH data table) are not allowed.
- The Maxwell force flags and magnetic virtual displacements body loads are not applicable.

ANSYS Emag

- This element has only magnetic and electric field capability, and does not have structural, thermal, or piezoelectric capability.
- The only active degrees of freedom are MAG and VOLT.
- If KEYOPT(1) = 1, the TEMP degree of freedom is inactive. KEYOPT(1) settings of 0, 2, 3 and 8 are not allowed.
- The only allowable material properties are the magnetic and electric properties (MUZRO through PERZ, plus the BH data table).
- The only applicable surface loads are Maxwell force flags. The only applicable body loads are temperatures (for material property evaluation only) and magnetic virtual displacements.
- The element does not have stress stiffening or birth and death features.
- KEYOPT(3) is not applicable.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

INFIN110

2-D Infinite Solid

MP ME <> <> <> <> EM <> PP EME <>
Product Restrictions

INFIN110 Element Description

INFIN110 models an open boundary of a 2-D unbounded field problem. A single layer of elements is used to represent an exterior sub-domain of semi-infinite extent. The layer models the effect of far-field decay in magnetic, electrostatic, thermal, or electric current conduction analyses. For information about enclosed elements and analysis types, see Table 110.1: INFIN110 Analyses (p. 391). INFIN110 may be used for planar and axisymmetric analyses. See INFIN110 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 110.1: INFIN110 Geometry

INFIN110 Input Data

The geometry, node locations, and the coordinate system for the element are shown in Figure 110.1: INFIN110 Geometry (p. 389). KEYOPT(1) specifies the degree of freedom to be used. KEYOPT(2) specifies whether a 4-node or 8-node element is used.

Only one layer of INFIN110 elements should be used between the finite element model and the exterior (infinite) surface. The nodes may be input starting at any corner node, but the face opposite of the finite element model (the exterior face) must be flagged as an infinite surface. This is usually done by selecting the nodes at the outer surface and issuing the **SF**,all,INF command. The other faces have no meaning. For best results, edges connecting the inner and outer surfaces of the infinite element should be radial from the center of the model.

A summary of the element input is given in "INFIN110 Input Summary" (p. 390). A general description of element input is given in Element Input (p. 43). For axisymmetric applications see Harmonic Axisymmetric Elements (p. 83).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

INFIN110 Input Summary

Nodes

I, J, K, L (if KEYOPT(2) = 0)
I, J, K, L, M, N, O, P (if KEYOPT(2) = 1)

Degrees of Freedom
Set by KEYOPT(1). See Table 110.1: INFIN110 Analyses (p. 391).

Real Constants
None

Material Properties
See Table 110.2: INFIN110 Material Properties (p. 391).

Surface Loads

Infinite Surface Flags --
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads
None

Special Features
None

KEYOPT(1)
Element degrees of freedom. See Table 110.1: INFIN110 Analyses (p. 391).

KEYOPT(2)
Element definition:

0 --
4-node quadrilateral

1 --
8-node quadrilateral

KEYOPT(3)
Element behavior:

0 --
Plane

1 --
Axisymmetric

KEYOPT(6)
Electric charge reaction sign:

0 --
Positive

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
Negative

Analysis categories are shown in the following table. KEYOPT(1) specifies the element degree of freedom.

Table 110.1: INFIN110 Analyses

Analysis Category	KEYOPT(1)	DOF Label	Reaction Solution	Enclosed Elements	Analysis Type
Magnetic	0	AZ	Magnetic Current Segment (F label = CSG)	PLANE13 PLANE53	Static Harmonic Transient
Electrostatic	1	VOLT	Electric Charge (F label = CHRG)	PLANE121	Static Harmonic
Thermal	2	TEMP	Heat Flow (F label = HEAT)	PLANE35 PLANE55 PLANE77	Steady-state Transient
Electric Current Conduction	3	VOLT	Electric Current (F label = AMPS)	PLANE230	Steady-state Harmonic Transient

INFIN110 material properties are shown in the following table. Nonzero material properties must be defined. Material properties are defined with the **MP**, **MPDATA** and **EMUNIT** commands.

Table 110.2: INFIN110 Material Properties

Analysis Category	KEYOPT(1)	Material Properties
Magnetic	0	MUZERO
Electrostatic	1	PERX, PERY, LSST, RSVX, RSVY
Thermal	2	KXX, KYY, DENS, C
Electric Current Conduction	3	RSVX, RSVY, LSST, PERX, PERY

INFIN110 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Additional element output as shown in Table 110.3: Element Output Definitions (p. 392)

Several items are illustrated in Figure 110.2: INFIN110 Element Output (p. 392). The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

Figure 110.2: INFIN110 Element Output

The following notation is used in Table 110.3: Element Output Definitions (p. 392):

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 110.3: Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L (KEYOPT(2) = 0) Nodes - I, J, K, L, M, N, O, P (KEYOPT(2) = 1)	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	4
MUZERO	Magnetic permeability of free space	1	1
PERX, PERY	Electric relative permittivity (element coordinates)	2	2
KXX, KYY	Thermal conductivity (element coordinates)	3	3

1. If KEYOPT(1) = 0

2. If KEYOPT(1) = 1

3. If KEYOPT(1) = 2

4. Available only at centroid as a ***GET** item.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Table 110.4: INFIN110 Item and Sequence Numbers (p. 393) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 110.4: INFIN110 Item and Sequence Numbers (p. 393):

Name
output quantity as defined in the Table 110.3: Element Output Definitions (p. 392)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 110.4: INFIN110 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
MUZERO	NMISC	1
PERX	NMISC	1
PERY	NMISC	2
KXX	NMISC	1
KYY	NMISC	2

INFIN110 Assumptions and Restrictions

- The area of the quadrilateral infinite element must be nonzero.
- The element cannot degenerate to a triangle.
- The exterior surface (for example, KL or KOL in Figure 110.1: INFIN110 Geometry (p. 389)) of the element must be flagged using the *INF* option on the **SF** family of commands.
- Only one layer of infinite elements can be used on the exterior boundary of the finite element model.
- The lines JK and IL of the infinite element IJKL (in Figure 110.2: INFIN110 Element Output (p. 392)) should either be parallel or divergent from each other. That is, the enclosed surface should be convex and the infinite domain must be represented by one layer of infinite elements without overlap or gap. Ideally, the length OJ should equal JK, and OI should equal IL. The point "O" is the "pole" of mapping for the infinite element. The pole is chosen arbitrarily, and may or may not coincide with the origin of the coordinate system. For best results, the poles should be placed at the centers of disturbances (loads). There can be multiple poles for a problem. See the *Mechanical APDL Theory Reference* for more about poles.
- Although this element can have 8 nodes (KEYOPT(2) = 1), for theoretical reasons (see the *Mechanical APDL Theory Reference*), only 5 nodes are included in the solution.
- The element assumes that the degree of freedom (DOF) value at infinity is *always* zero (0.0). That is, the DOF value at infinity is *not* affected by **TUNIF**, **D**, or other load commands.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The infinite elements are not included in solution result displays but may be viewed in element displays [**EPLOT**].
- There are considerations in the application of INFIN110 that will lead to optimal performance in the analysis of your model. These consideration are covered in detail in the *Low-Frequency Electromagnetic Analysis Guide*.
- When used in a model with the higher-order elements PLANE35, PLANE53, PLANE77, PLANE121, and PLANE230 use the higher-order setting for INFIN110 (KEYOPT(2) = 1).
- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, elements must have the same reaction solution for the VOLT DOF. Elements that have an electric charge reaction solution must all have the same electric charge reaction sign. KEYOPT(6) sets the electric charge reaction sign. For more information, see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*.

INFIN110 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Mechanical

These restrictions apply unless the Emag option is enabled.

- This element does not have magnetic, electrostatic, or electric current conduction capability.
- The AZ and VOLT degrees of freedom are not active. KEYOPT(1) defaults to 2 (TEMP) instead of 0 and cannot be changed.
- The material properties MUZERO, PERX, PERY, RSVX, RSVY, and LSST are not allowed.

ANSYS Emag

- This element has only magnetic, electrostatic, and electric current conduction capability, and does not have thermal capability.
- The only active degrees of freedom are AZ and VOLT. KEYOPT(1) can only be set to 0 or 1.
- The only allowable material properties are MUZERO, PERX, PERY, RSVX, RSVY, and LSST.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

INFIN111

3-D Infinite Solid

MP ME ST <> <> <> EM <> PP EME MFS
Product Restrictions

INFIN111 Element Description

INFIN111 models an open boundary of a 3-D unbounded field problem. A single layer of elements is used to represent an exterior sub-domain of semi-infinite extent. The layer models the effect of far-field decay in magnetic, electrostatic, thermal, or electric current conduction analyses. For information about enclosed elements and analysis types, see Table 111.1: INFIN111 Analyses (p. 397). See INFIN111 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 111.1: INFIN111 Geometry

INFIN111 Input Data

The geometry, node locations, and the coordinate system for the element are shown in Figure 111.1: INFIN111 Geometry (p. 395). KEYOPT(1) specifies the degree(s) of freedom to be used. KEYOPT(2) specifies whether a 8-node or 20-node element is used.

Only one layer of INFIN111 elements should be used between the finite element model and the exterior (infinite) surface. The nodes may be input starting at any corner node, but the face opposite of the finite element model (the exterior face) must be flagged as an infinite surface. This is usually done by selecting the nodes at the outer surface and issuing the **SF**,all,INF command. The other faces have no meaning. For best results, edges connecting the inner and outer surfaces of the infinite element should be radial from the center of the model.

A summary of the element input is given in "INFIN111 Input Summary" (p. 396). A general description of element input is given in Element Input (p. 43).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

INFIN111 Input Summary

Nodes

I, J, K, L, M, N, O, P (if KEYOPT(2) = 0)
I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B (if KEYOPT(2) = 1)

Degrees of Freedom
Set by KEYOPT(1). See Table 111.1: INFIN111 Analyses (p. 397).

Real Constants
None

Material Properties
See Table 111.2: INFIN111 Material Properties (p. 397).

Surface Loads

Infinite Surface Flags --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads
None

Special Features
None

KEYOPT(1)
Element degree of freedom. See Table 111.1: INFIN111 Analyses (p. 397).

KEYOPT(2)
Element definition:

0 --
8-node brick

1 --
20-node brick

KEYOPT(6)
Electric charge reaction sign:

0 --
Negative

1 --
Positive

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Analysis categories are shown in the following table. KEYOPT(1) specifies the element degree of freedom.

Table 111.1: INFIN111 Analyses

Analysis Category	KEYOPT(1)	DOF Label	Reaction Solution	Enclosed Elements	Analysis Type
Magnetic	0	MAG	Magnetic Flux (F label = FLUX)	SOLID5 SOLID96 SOLID98	Static
Magnetic	1	AX, AY, AZ	Magnetic Current Segments (F label = CSG)	SOLID97	Static Harmonic Transient
Electrostatic	2	VOLT	Electric Charge (F label = CHRG)	SOLID122 SOLID123	Static Harmonic
Thermal	3	TEMP	Heat Flow (F label = HEAT)	SOLID70 SOLID87 SOLID90	Steady-state Transient
Electric Current Conduction	4	VOLT	Electric Current (F label = AMPS)	SOLID231 SOLID232	Steady-state Harmonic Transient

INFIN111 material properties are shown in the following table. Nonzero material properties must be defined. Material properties are defined with the **MP**, **MPDATA** and **EMUNIT** commands.

Table 111.2: INFIN111 Material Properties

Analysis Category	KEYOPT(1)	Material Properties
Magnetic	0	MUZERO
Magnetic	1	MUZERO
Electrostatic	2	PERX, PERY, PERZ, LSST, RSVX, RSVY, RSVZ
Thermal	3	KXX, KYY, KZZ, DENS, C
Electric Current Conduction	4	RSVX, RSVY, RSVZ, LSST, PERX, PERY, PERZ

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

INFIN111 Output Data

The solution output associated with the element is in two forms:

- Nodal potentials included in the overall nodal solution
- Additional element output as shown in Table 111.3: INFIN111 Element Output Definitions (p. 398)

The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 111.3: INFIN111 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P (if KEYOPT(2) = 0); Nodes - I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B (if KEYOPT(2) = 1)	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	4
MUZERO	Magnetic permeability of free space	1	1
PERX, PERY, PERZ	Electric relative permittivity	2	2
KXX, KYY, KZZ	Thermal conductivity	3	3

1. If KEYOPT(1) = 0 or 1
2. If KEYOPT(1) = 2
3. If KEYOPT(1) = 3
4. Available only at centroid as a ***GET** item.

Table 111.4: INFIN111 Item and Sequence Numbers (p. 399) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 111.4: INFIN111 Item and Sequence Numbers (p. 399):

Name

output quantity as defined in the Table 111.3: INFIN111 Element Output Definitions (p. 398)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Item

predetermined Item label for **ETABLE** command

E

sequence number for single-valued or constant element data

Table 111.4: INFIN111 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
MUZERO	NMISC	1
PERX	NMISC	1
PERY	NMISC	2
PERZ	NMISC	3
KXX	NMISC	1
KYY	NMISC	2
KZZ	NMISC	3

INFIN111 Assumptions and Restrictions

- Assumptions and restrictions listed for INFIN110 elements also apply to INFIN111 elements (see INFIN110 Assumptions and Restrictions).
- There are considerations in the application of INFIN111 that will lead to optimal performance in the analysis of your model. These consideration are covered in detail in the *Low-Frequency Electromagnetic Analysis Guide*.
- When used in a model with the higher-order elements SOLID87, SOLID90, SOLID98, SOLID122, SOLID123, SOLID231, and SOLID232, use the higher-order setting for INFIN111 (KEYOPT(2) = 1).
- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, elements must have the same reaction solution for the VOLT DOF. Elements that have an electric charge reaction solution must all have the same electric charge reaction sign. KEYOPT(6) sets the electric charge reaction sign. For more information, see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*.

INFIN111 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Mechanical

These restrictions apply unless the Emag option is enabled.

- This element does not have magnetic, electrostatic, or electric current conduction capability.
- The MAG, AX, AY, AZ, and VOLT degrees of freedom are not active. KEYOPT(1) defaults to 3 (TEMP) instead of 0 and cannot be changed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The material properties MUZERO, PERX, PERY, PERZ, RSVX, RSVY, RSVZ, and LSST are not allowed.

ANSYS Emag

- This element has only magnetic, electrostatic, and electric current conduction capability, and does not have thermal capability.
- TEMP is not allowed as a degree of freedom. KEYOPT(1) can only be set to 0, 1 or 2.
- The only allowable material properties are MUZERO, PERX, PERY, PERZ, RSVX, RSVY, RSVZ, and LSST.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FLUID116

Coupled Thermal-Fluid Pipe

MP ME <> PR PRN <> <> <> PP EME MFS
Product Restrictions

FLUID116 Element Description

FLUID116 is a 3-D element with the ability to conduct heat and transmit fluid between its two primary nodes. Heat flow is due to the conduction within the fluid and the mass transport of the fluid. Convection may be accounted for either with additional nodes and convection areas or with surface elements SURF151 and SURF152. In both cases, the film coefficient may be related to the fluid flow rate. The element may have two different types of degrees of freedom, temperature and/or pressure.

The thermal-flow element may be used in a steady-state or transient thermal analysis. If the model containing the thermal-flow element is also to be analyzed structurally, the element should be replaced by an equivalent (or null) structural element. See FLUID116 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 116.1: FLUID116 Geometry

FLUID116 Input Data

The geometry, node locations, and the coordinate system for this thermal-flow pipe element are shown in Figure 116.1: FLUID116 Geometry (p. 401). The element is defined by two primary nodes, two additional nodes if convection is desired, several real constants (see Table 116.1: FLUID116 Element Real Constants (p. 406)), and the material properties. The length L of the element is determined from the two primary node locations.

The material properties can be input as numerical values or as tabular inputs evaluated as a function of pressure, temperature, velocity, time, and location. If temperature or pressure, you need to activate the appropriate pressure or temperature degrees of freedom. Tabular material properties are calculated before the first iteration (i.e., using initial values [**IC**]).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The fluid mass density ρ (Mass/Length^3) is input as property DENS or computed following the ideal gas law if the real constant R_{gas} is present. If KEYOPT(2) = 2, 3, or 4, the convection film coefficient h_f (Heat/Length^2*Time*Deg) is input by the options defined by KEYOPT(4). If KEYOPT(2) = 1, convection surfaces using FLUID116 velocities and other information are stored and can be used by SURF151 or SURF152 and optionally the user programmable feature USRSURF116 in order to determine film coefficients and bulk temperatures as a function of velocities and other parameters. The input tables are explained in detail in Table 116.2: FLUID116 Empirical Data Table (Optional) (p. 407). The thermal conductivity k_{xx} (Heat/Length*time*Deg) acts in the element longitudinal direction and is input as property KXX. The specific heat c_p (Heat/Mass*Deg or Heat*Length/Force*Time^2*Deg) is input as property C. The fluid viscosity μ is input as property VISC. In an axisymmetric analysis, such as for annular flow, the flow area, the convection areas, and all other input should be on a full 360° basis.

KEYOPT(2) = 3 and 4 are variations of KEYOPT(2) = 2 used to avoid an artificial reduction of the change in temperature in the last element next to an inlet or outlet with no specified temperature. If such an inlet or outlet is at node I, use KEYOPT(2) = 3 and if it is at node J, use KEYOPT(2) = 4. All elements of a run of pipe should use the same KEYOPT, not just the end one. For networks where the usage of KEYOPT(2) is not obvious and the detailed temperature distribution is important, use KEYOPT(2) = 2 with a relatively fine mesh (small elements). The effect of KEYOPT(2) = 3 and 4 could be alternatively achieved by adjusting the convection areas (Real Constants 7 and 8) but it is not as convenient.

The coefficient of friction (input as property MU) is the starting value of the Moody friction factor (f). The friction factor for the first iteration is always assumed to be MU. The smooth-pipe empirical correlations are a function of Reynolds number (Re) and depend on whether the flow is laminar or turbulent (Re>2500). If a friction table is supplied (**TB**,FCON), the friction factor is recomputed each substep from the table (using linear interpolation where necessary). The table is also explained in detail in Table 116.2: FLUID116 Empirical Data Table (Optional) (p. 407).

The word PRES (or TEMP) should be input for the *Lab* variable on the **D** command and the pressure (or temperature) value input for the value. If a nodal heat (or fluid) flow rate is defined with the **F** command, input the word HEAT (or FLOW) for the *Lab* variable and input the flow rate for the value. If temperature is the only degree of freedom, (KEYOPT(1) = 1), you can input a known flow rate in units of mass/time via an **SFE**,,,HFLUX command (rather than **F**,,FLOW). Fluid weight effects are activated by specifying a nonzero acceleration and/or rotation vector [**ACEL** and/or **OMEGA**].

When using the rotational speed and slip factor real constants (real constants 7-10 in Table 116.1: FLUID116 Element Real Constants (p. 406)), you can specify either numerical values or table inputs. If specifying table inputs, enclose the table name in % signs (for example, %`tabname`%). Also, if using table inputs for rotational speed, either both real constants 7 and 8 should have the same table name reference, or real constant 8 should be unspecified. Similarly, if using table inputs for slip factor, either both real constants 9 and 10 should have the same table name reference, or real constant 10 should be unspecified. Both rotational speed and the slip factor can vary with time and location.

If tabular real constants are used, then any node in a FLUID116 network must refer to a single table name. For correct results, at any node, the table names from different elements must all be the same and a table name cannot be used along with any numerical real constant from a different element.

See Steady-State Thermal Analysis in the *Thermal Analysis Guide* for more information on using table inputs.

Element loads are described in Nodal Loading (p. 49). Element body loads may be input as heat generation rates at the nodes. The node J heat generation rate HG(J) defaults to the node I heat generation rate HG(I).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(8) is used for inputting flow losses (see Table 116.1: FLUID116 Element Real Constants (p. 406)). Momentum losses in pipes due to bends, elbows, joints, valves, etc., may be represented by a fictitious (equivalent) length of pipe L_a. This equivalent length may be input directly or calculated from an input constant K, the hydraulic diameter D, and the friction factor f.

FLUID116 elements can be used to model fluid flow between hydrostatic fluid elements, (HSFLD241 or HSFLD242). A single FLUID116 element connecting the pressure nodes of two different hydrostatic fluid elements is sufficient to model the flow. However, more FLUID116 elements can be added if the actual geometry of the connection needs to be modeled. For FLUID116 elements that are directly or indirectly connected to hydrostatic fluid element, you must set KEYOPT(1) = 3 to convert the fluid element mass flow rate to volume change (for compatibility with the hydrostatic fluid elements).

A summary of the element input is given in "FLUID116 Input Summary" (p. 403). A general description of element input is given in Element Input (p. 43).

FLUID116 Input Summary

Nodes

I, J or I, J, K, L (see KEYOPT(2))

Degrees of Freedom

PRES, TEMP if KEYOPT(1) = 0
TEMP if KEYOPT(1) = 1
PRES if KEYOPT(1) = 2 or 3

Real Constants

See Table 116.1: FLUID116 Element Real Constants (p. 406)

Material Properties

MP command: KXX, C, DENS, MU, VISC, HF

Surface Loads

You can specify imposed mass flow via an **SFE**,,,HFLUX command. Valid only when KEYOPT(1) = 1.

Body Loads

Heat Generations --

HG(I), HG(J)

Special Features

Nonlinearity

KEYOPT(1)

Pressure and temperature degrees of freedom:

0 --

PRES and TEMP degrees of freedom

1 --

TEMP degrees of freedom only

2 --

PRES degrees of freedom only. This option is not valid when FLUID116 is connected to HSFLD241 or HSFLD242.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3 --
PRES degrees of freedom only. This option is only valid when FLUID116 is directly or indirectly connected to HSFLD241 or HSFLD242.

KEYOPT(2) (used only if KEYOPT(1) = 0 or 1)

0 --
2 nodes and no convection surface or convection information

1 --
2 nodes and convection information passed to SURF151/SURF152

2 --
4 nodes and convection surface logic included with this element, convection area shared between nodes I and J

3 --
4 nodes and convection surface logic included with this element, convection area only at node I

4 --
4 nodes and convection surface logic included with this element, convection area only at node J

KEYOPT(4) (used only if KEYOPT(2) = 2, 3, or 4)
Film coefficient (h_f) definition

0 --
Use **MP**,HF

1 --
Use real constants 9 thru 12 (see Table 116.1: FLUID116 Element Real Constants (p. 406))

2 --
Use **TB**,HFLM for h_f as a function of temperature and average velocity

3 --
Use **TB**,HFLM for h_f as a function of temperature and Reynold's number

4 --
Use **TB**,HFLM for Nu as a function of temperature and Reynold's number ($h_f = K_{xx}$*Nu/diam)

5 --
Use call to User116Hf

KEYOPT(5) (used only if KEYOPT(4) = 0, 2, 3, 4, or 5)
Evaluation of film coefficient:

0 --
Average fluid temperature (TI + TJ)/2

1 --
Average wall temperature (TK +TL)/2

2 --
Average film temperature (TI + TJ + TK + TL)/4

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3 --
Differential temperature (TI + TJ)/2 - (TK + TL)/2

KEYOPT(6) (used only if KEYOPT(1) = 0 or 2)
Fluid conductance coefficient definition:

0 --
Use conductance formula

1 --
Use real constant C

2 --
Use **TB**,FCON as a function of temperature and average velocity

3 --
Use **TB**,FCON as a function of temperature and Reynold's number

4 --
Use call to User116Cond

KEYOPT(7) (used only if KEYOPT(6) = 0)
Friction factor calculation:

0 --
Use smooth pipe empirical correlations

1 --
Use **MP**,MU

2 --
Use **TB**,FCON with friction factor being a function of temperature and average velocity

3 --
Use **TB**,FCON with friction factor being a function of temperature and Reynold's number

KEYOPT(8) (used only if KEYOPT(6) = 0)
Flow losses specified by input:

0 --
Use real constant L_a as the additional length

1 --
Use real constant K as loss coefficient

KEYOPT(9)
Discretization scheme:

0 --
Upwind difference linear shape function (default). This scheme has lower order accuracy than the other schemes.

1 --
Central difference linear shape function. This scheme has higher order accuracy but it can lead to oscillations near bends.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2 --

Upwind difference exponential shape function. This scheme has high accuracy and does not produce oscillations near bends.

Table 116.1: FLUID116 Element Real Constants

(Given in the order required for input in the real constant table)			
No.	**Name**	**Definition**	**Units**
1	**D**	Hydraulic diameter.	Length
2	**A**	Flow cross-sectional area.	Length^2
3	**N_c**	Number of flow channels (defaults to 1). If greater than 1, real constants and element output are on a per channel basis.	
4-6		not currently used	
7	**$(A_n)_I$**	If KEYOPT(2) = 1, angular velocity associated with node I. If KEYOPT(2) = 2, 3, or 4, convection area between nodes I and K. Defaults to πDL/2 if KEYOPT(2) = 2, defaults to πDL if KEYOPT(2) = 3 where: L = element length	Length^2
8	**$(A_n)_J$**	If KEYOPT(2) = 1, angular velocity associated with node J. Defaults to value at node I. If KEYOPT(2) = 2, 3, or 4, convection area between nodes J and L. Defaults to πDL/2 if KEYOPT(2) = 2, defaults to πDL if KEYOPT(2) = 4	Length^2
9	**SLIPFAI**	If KEYOPT(2) = 1, slip factor at node I.	
10	**SLIPFAJ**	If KEYOPT(2) = 1, slip factor at node J. Defaults to value at node I.	
9-12	**N1, N2, N3, N4**	(Used if KEYOPT(4) = 1 and KEYOPT(2) = 2, 3, or 4) $Nu = N1 + N2\ Re^{N3}\ Pr^{N4}$ where: Re = Reynolds number (WD/ μA) Pr = Prandtl number ($C_p\mu$/KXX) C_p = specific heat For example, the Dittus-Boelter correlation for full-developed turbulent flow in smooth pipes may be input with N1 = 0.0, N2 = 0.023, N3 = 0.8, and N4 = 0.4 (heating).	
13	**P_p**	Pump pressure.	Force / Length^2
14		Used to compute conductance coefficient C where: $W = C\sqrt{\Delta p}$ Δp = pressure drop	

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(Given in the order required for input in the real constant table)			
No.	**Name**	**Definition**	**Units**
	C_r	If KEYOPT(6) = 1, conductance coefficient is used to calculate flow. Hence, $C = C_r \frac{\sqrt{\text{Weight} * \text{Length}}}{\text{Time}}$	
	L_a	If KEYOPT(6) = 0, and KEYOPT(8) = 0, additional Length of pipe to account for flow losses (for example, valves, orifices, etc.) Hence, $C = A\sqrt{2\rho D/(F(L + L_a))}$ where: ρ = DENS F = friction coefficient	
	K	If KEYOPT(6) = 0 and KEYOPT(8) = 1, this real constant is the loss coefficient K. Hence, $C = A\sqrt{2\rho D/(FL + KD)}$	
15-18		not currently used	
19	R_{gas}	Gas constant in ideal gas law ($\rho = p/(R_{gas}T_{abs})$), where T_{abs} is the absolute temperature and p = average pressure. If zero, use ρ as specified by the DENS material property.	Length^2 / Deg * Time^2
20	V_{DF}	Viscous damping multiplier. Default 0.0	
21	C_{ver}	Units conversion factor for viscous damping. Default = 1.0 $Q_v = V_{DF}C_{ver}F\pi VISC(VELOC)^2L$ = viscous heating for element, with F = 8.0 for laminar and 0.21420 for turbulent flow.	

Note

Real constants 7 through 12 and 20 and 21 are used only if KEYOPT(1) = 0 or 1 and real constants 13 through 19 are used only if KEYOPT(1) = 0 or 2.

The data in Table 116.2: FLUID116 Empirical Data Table (Optional) (p. 407) is entered in the data table with the **TB** commands. The curves are initialized by using the **TB** command. The temperature for the first curve is input with the **TBTEMP** command, followed by **TBPT** commands for up to 100 points. Up to 20 temperature-dependent curves (NTEMP = 20 maximum on the **TB** command) may be defined in this manner. The constants (X, Y) entered on the **TBPT** command (two per command).

Table 116.2: FLUID116 Empirical Data Table (Optional)

Constant	**Meaning**
Film Coefficient The film coefficient table is initialized with the **TB**,HFLM command. The **TBPT** data are:	
X	Velocity (Length/Time)
Y	Film Coefficient (Heat/(time*area*temp) The velocity may be replaced with the Reynold's number, and the film coefficient may be replaced with the Nusselt number, depending on KEYOPT(4).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Con-stant	Meaning
Fluid Conductance/Friction Factor The fluid conductance/friction factor is initialized with the **TB**,FCON command. The **TBPT** data are:	
X	Velocity (Length/Time)
Y	Corresponding friction factor value (Dimensionless)
The velocity may be replaced with the Reynold's number, and the friction factor may be replaced with the fluid conductance, depending on KEYOPT(6) and KEYOPT(7).	

FLUID116 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in Table 116.3: FLUID116 Element Output Definitions (p. 408)

The fluid flow rate is expressed in units of Mass/Time and is positive from node I to node J. In an axisymmetric analysis these flow rates and all other output are on a full 360° basis. The fluid flow rate and the heat flow rate at the nodes may be printed with the **OUTPR** command. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The following notation is used in Table 116.3: FLUID116 Element Output Definitions (p. 408):

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 116.3: FLUID116 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
XC, YC, ZC	Location where results are reported	Y	4
VELOC	Average velocity	Y	Y
RE	Reynolds number	Y	Y
FLOW RATE	Flow rate from node I to node J	Y	Y
HT COND RATE	Heat flow rate from node I to node J due to conduction	1	1
HT TRANSP RATE	Heat flow rate at node I due to mass transport	1	1
CONV AREAS (I, J)	Convection areas at nodes I and J	3	3

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
HFILM	Film coefficient	3	3
NUS	Nusselt number	3	3
PR	Prandtl number	3	3
HT CONV RATES (I, J)	Heat flow rates from nodes I to K and from nodes J to L due to convection	3	3
HGVD	Heat generation due to direct input and viscous damping	1	1
TEMP	Temperature	-	1
PUMP PR	Pump pressure	2	2
FRICTION	Friction factor	2	2
PRES	Pressure	-	2

1. If KEYOPT(1) = 0 or 1
2. If KEYOPT(1) = 0 or 2
3. If KEYOPT(2) = 2, 3, or 4
4. Available only at centroid as a ***GET** item.

Table 116.4: FLUID116 Item and Sequence Numbers (p. 409) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 116.4: FLUID116 Item and Sequence Numbers (p. 409):

Name
output quantity as defined in the Table 116.3: FLUID116 Element Output Definitions (p. 408)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J,K,L
sequence number for data at nodes I,J,K,L

Table 116.4: FLUID116 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
VELOC	NMISC	1	-	-	-	-
RE	NMISC	2	-	-	-	-
FLOW RATE	NMISC	3	-	-	-	-
HEAT COND RATE	NMISC	4	-	-	-	-
HEAT TRANSP RATE	NMISC	5	-	-	-	-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
CONV AREA	NMISC	-	6	7	-	-
HFILM	NMISC	8	-	-	-	-
NUS	NMISC	9	-	-	-	-
PR	NMISC	10	-	-	-	-
HEAT CONV RATE	NMISC	-	11	12	-	-
HGVD	NMISC	-	13	14	-	-
TEMP	NMISC	-	15	16	17	18
PUMP PR	NMISC	19	-	-	-	-
FRICTION	NMISC	20	-	-	-	-
PRES	NMISC	-	21	22	-	-

FLUID116 Assumptions and Restrictions

- The element must not have a zero length, so nodes I and J must not be coincident.
- Nodes K and L may be located anywhere in space, even coincident with I and J, respectively.
- D must always be nonzero.
- A defaults to $\pi D^2/4.0$ and is assumed to remain constant for the element.
- Compressibility and flow inertia effects of the fluid are not included in the element formulation.
- If temperatures are degrees of freedom, the resulting unsymmetric matrix requires twice as much memory storage for the solution as other ANSYS elements.
- HF must be nonzero for the four node element.
- MU and DENS must be nonzero if a flow solution is desired and KEYOPT(6) is not zero.
- If the flow is specified at a node also having a specified pressure, the flow constraint is ignored.
- In general, flow is usually specified at the inlet, pressure at the outlet.
- For problems involving pressure specification on inlets and outlets, the solution may converge too soon (i.e., the PRES degree of freedom has converged but FLOW has not due to a loose convergence criterion). Be sure to check your results carefully. To force more iterations, you can tighten the convergence criteria (i.e., **CNVTOL**,flow,,1e-30), or you can specify a nonzero initial condition on pressure, which could be an average of the specified inlet and outlet pressures (i.e., **IC**,all,pres,pavg). You can use both options together; however, ANSYS, Inc. recommends using a nonzero initial condition. Tightening the convergence requires you to estimate a suitable tolerance.
- More substeps are required for convergence as the flow approaches zero.
- See the **CNVTOL** command for convergence control.
- If pressure is a degree of freedom, the element is nonlinear and requires an iterative solution.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FLUID116 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

These restrictions apply when using this element with the ANSYS Professional product.

- The PRES degree of freedom (KEYOPT (1) = 0, 2) is not available with the ANSYS Professional product.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

PLANE121

2-D 8-Node Electrostatic Solid

MP <> <> <> <> <> EM <> PP EME <>
Product Restrictions

PLANE121 Element Description

PLANE121 is a 2-D, 8-node, charge-based electric element. The element has one degree of freedom, voltage, at each node. The 8-node elements have compatible voltage shapes and are well suited to model curved boundaries.

This element is based on the electric scalar potential formulation, and it is applicable to 2-D electrostatic and time-harmonic quasistatic electric field analyses. Various printout options are also available. See PLANE121 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 121.1: PLANE121 Geometry

PLANE121 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 121.1: PLANE121 Geometry (p. 413). The element is defined by eight nodes and orthotropic material properties. A triangular-shaped element may be formed by defining the same node number for nodes K, L and O.

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems (p. 55). Properties not input default as described in Linear Material Properties in the *Material Reference*.

Nodal loads are defined with the **D** (*Lab*= VOLT) and **F** (*Lab*= CHRG) commands. The nodal forces, if any, should be input per unit of depth for a plane analysis and on a full 360° basis for an axisymmetric analysis.

Element loads are described in Nodal Loading (p. 49). Surface charge densities may be input as surface loads at the element faces as shown by the circled numbers in Figure 121.1: PLANE121 Geometry (p. 413). Charge densities may be input as element body loads at the nodes. If the node I charge density CHRGD(I) is input, and all others are unspecified, they default to CHRGD(I). If all corner node charge densities are specified, each midside node charge density defaults to the average charge density of its adjacent corner nodes.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The temperature (for material property evaluation only) body loads may be input based on their value at the element's nodes or as a single element value [**BF**, **BFE**]. In general, unspecified nodal values of temperatures default to the uniform value specified with the **BFUNIF** or **TUNIF** commands.

A summary of the element input is given in "PLANE121 Input Summary" (p. 414). A general description of element input is given in Element Input (p. 43). For axisymmetric applications see Harmonic Axisymmetric Elements (p. 83).

PLANE121 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
VOLT

Real Constants
THK - Thickness (used only if KEYOPT(3) = 3)

Material Properties
MP command: PERX, PERY, LSST, RSVX, RSVY

EMUNIT command: EPZRO

Surface Loads

Surface charge densities --
CHRGS face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads

Temperature --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

Volume charge densities --
CHRGD(I), CHRGD(J), CHRGD(K), CHRGD(L), CHRGD(M), CHRGD(N), CHRGD(O), CHRGD(P)

Special Features
Birth and death

KEYOPT(3)
Element behavior:

0 --
Plane

1 --
Axisymmetric

3 --
Plane with thickness input, specified via real constant THK.

KEYOPT(4)
Element coordinate system defined:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0 --
Element coordinate system is parallel to the global coordinate system

1 --
Element coordinate system is based on the element I-J side

KEYOPT(5)
Extra element output:

0 --
Basic element printout

1 --
Repeat basic solution for all integration points

2 --
Nodal fields printout

KEYOPT(6)
Electric charge reaction sign:

0 --
Positive

1 --
Negative

PLANE121 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in Table 121.1: PLANE121 Element Output Definitions (p. 416)

Several items are illustrated in Figure 121.2: PLANE121 Output (p. 415). The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

Figure 121.2: PLANE121 Output

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 121.1: PLANE121 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	2
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	Y	Y
LOC	Output location (X, Y)	1	1
PERX, PERY	Electric relative permittivity	1	1
EF:X, Y	Electric field components	1	1
EF:SUM	Vector magnitude of EF	1	1
D:X, Y	Electric flux density components	1	1
D:SUM	Vector magnitude of D	1	1
JS:X, Y, SUM	Current density components (in the global Cartesian coordinate system) and vector magnitude [3]	1	1
JT:X, Y, SUM	Conduction current density components and magnitude [3]	1	1
JHEAT:	Joule heat generation rate per unit volume [4] [5] [6]	1	1
SENE:	Stored electric energy [6]	1	1
FMAG:X, Y	Electrostatic force [7]	-	1
CHRGD	Applied charge density	-	Y

1. The solution value is output only if calculated (based upon input data). The element solution is at the centroid.

2. Available only at centroid as a ***GET** item.

3. For a time-harmonic analysis, JS represents the sum of element conduction and displacement current densities. JT represents the element conduction current density. The element displacement current density (JD) can be derived from JS and JT as JD = JS-JT.

4. For a time-harmonic analysis, calculated Joule heat generation rate per unit volume (JHEAT) includes conduction heating and dielectric heating due to the loss tangent.

5. Calculated Joule heat generation rate per unit volume (JHEAT) may be made available for a subsequent thermal analysis with companion elements [**LDREAD**].

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. For a time-harmonic analysis, Joule losses (JHEAT) and stored energy (SENE) represent time-average values. These values are stored in both the real and imaginary data sets.

7. Use the **EMFT** macro to calculate the force distribution over the body. See the discussion on Electrostatic Forces in the *Low-Frequency Electromagnetic Analysis Guide*.

Table 121.2: PLANE121 Miscellaneous Element Output

Description	Names of Items Output	O	R
Integration Point Solution	LOC, PERX, PERY, PERZ, EF, EFSUM, D, DSUM	1	-
Nodal Solution	EF, EFSUM, D, DSUM	2	-

1. Output at each integration point, if KEYOPT(5) = 1

2. Output at each node, if KEYOPT(5) = 2

Note

For axisymmetric solutions with KEYOPT(4) = 0, the X and Y directions correspond to the radial and axial directions, respectively.

Table 121.3: PLANE121 Item and Sequence Numbers (p. 417) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 121.3: PLANE121 Item and Sequence Numbers (p. 417):

Name
output quantity as defined in the Table 121.1: PLANE121 Element Output Definitions (p. 416)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 121.3: PLANE121 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
CHRGD	SMISC	1
PERX	NMISC	1
PERY	NMISC	2
JTX	NMISC	4
JTY	NMISC	5
JTSUM	NMISC	6

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PLANE121 Assumptions and Restrictions

- The area of the element must be positive.
- The element must lie in a global X-Y plane as shown in Figure 121.2: PLANE121 Output (p. 415), and the Y-axis must be the axis of symmetry for axisymmetric analyses.
- An axisymmetric structure should be modeled in the +X quadrants.
- A face with a removed midside node implies that the potential varies linearly, rather than parabolically, along that face. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.
- This element is only compatible with elements having a VOLT DOF and an electric charge reaction solution. Electric charge reactions must all be positive or negative. KEYOPT(6) sets the electric charge reaction sign. See Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide* for more information.

PLANE121 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Emag

- The birth and death special feature is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID122

3-D 20-Node Electrostatic Solid

MP <> <> <> <> <> EM <> PP EME <>
Product Restrictions

SOLID122 Element Description

SOLID122 is a 3-D, 20-node, charge-based electric element. The element has one degree of freedom, voltage, at each node. It can tolerate irregular shapes without much loss of accuracy. SOLID122 elements have compatible voltage shapes and are well suited to model curved boundaries.

This element is applicable to 3-D electrostatic and time-harmonic quasistatic electric field analyses. Various printout options are also available. See SOLID122 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 122.1: SOLID122 Geometry

SOLID122 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 122.1: SOLID122 Geometry (p. 419). The element is defined by 20 node points and the material properties. A prism-shaped element may be formed by defining duplicate K, L, and S; A and B; and O, P, and W node numbers. A pyramid-shaped element and a tetrahedral-shaped element may also be formed as shown in Figure 122.1: SOLID122 Geometry (p. 419).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems (p. 55). Properties not input default as described in Linear Material Properties in the *Material Reference*. Nodal loads are defined with the **D** (*Lab*= VOLT) and **F** (*Lab*= CHRG) commands.

Element loads are described in Nodal Loading (p. 49). Surface charge densities may be input as surface loads at the element faces as shown by the circled numbers on Figure 122.1: SOLID122 Geometry (p. 419). Charge density may be input as element body loads at the nodes. If the node I charge densities CHRGD(I) is input, and all others are unspecified, they default to CHRGD(I). If all corner node charge densities are specified, each midside node charge density defaults to the average charge density of its adjacent corner nodes.

The temperature (for material property evaluation only) body loads may be input based on their value at the element's nodes or as a single element value [**BF**, **BFE**]. In general, unspecified nodal values of temperatures default to the uniform value specified with the **BFUNIF** or **TUNIF** commands.

A summary of the element input is given in "SOLID122 Input Summary" (p. 420). A general description of element input is given in Element Input (p. 43).

SOLID122 Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B

Degrees of Freedom
VOLT

Real Constants
None

Material Properties
MP command: PERX, PERY, PERZ, LSST, RSVX, RSVY, RSVZ

EMUNIT command: EPZRO

Surface Loads

Surface charge densities --

CHRGS face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N), face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperature --
T(I), T(J), ..., T(Z), T(A), T(B)

Volume charge densities --

CHRGD(I), CHRGD(J), ..., CHRGD(Z), CHRGD(A), CHRGD(B)

Special Features
Birth and death

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(4)
Element coordinate system defined:

0 --
Element coordinate system is parallel to the global coordinate system

1 --
Element coordinate system is based on the element I-J side

KEYOPT(5)
Extra element output:

0 --
Basic element printout

1 --
Repeat basic solution for all integration points

2 --
Nodal fields printout

KEYOPT(6)
Electric charge reaction sign:

0 --
Positive

1 --
Negative

SOLID122 Output Data

The solution output associated with the element is in two forms:

- Nodal potentials included in the overall nodal solution
- Additional element output as shown in Table 122.1: SOLID122 Element Output Definitions (p. 421)

The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 122.1: SOLID122 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	2
TEMP	Temperatures T(I), T(J), ..., T(Z), T(A), T(B)	Y	Y
LOC	Output location (X, Y, Z)	1	1
PERX, PERY, PERZ	Electric relative permittivity	1	1
EF:X, Y, Z	Electric field components	1	1
EF:SUM	Vector magnitude of EF	1	1
D:X, Y, Z	Electric flux density components	1	1
D:SUM	Vector magnitude of D	1	1
JS:X, Y, Z, SUM	Current density components (in the global Cartesian coordinate system) and vector magnitude [3]	1	1
JT:X, Y, Z, SUM	Conduction current density components and magnitude [3]	1	1
JHEAT:	Joule heat generation rate per unit volume [4] [6] [5]	1	1
SENE:	Stored electric energy [5]	1	1
FMAG:X, Y, Z	Electrostatic force [7]	-	1
CHRGD	Applied charge density	-	Y

1. The solution value is output only if calculated (based upon input data). The element solution is at the centroid.

2. Available only at centroid as a ***GET** item.

3. For a time-harmonic analysis, JS represents the sum of element conduction and displacement current densities. JT represents the element conduction current density. The element displacement current density (JD) can be derived from JS and JT as JD=JS-JT. JS can be used as a source current density for a subsequent magnetostatic analysis with companion elements [**LDREAD**].

4. For a time-harmonic analysis, calculated Joule heat generation rate per unit volume (JHEAT) includes conduction heating and dielectric heating due to the loss tangent.

5. Calculated Joule heat generation rate per unit volume (JHEAT) may be made available for a subsequent thermal analysis with companion elements [**LDREAD**].

6. For a time-harmonic analysis, Joule losses (JHEAT) and stored energy (SENE) represent time-average values. These values are stored in both the real and imaginary data sets.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Use the **EMFT** macro to calculate the force distribution over the body. See the discussion on Electrostatic Forces in the *Low-Frequency Electromagnetic Analysis Guide*.

Table 122.2: SOLID122 Miscellaneous Element Output

Description	Names of Items Output	O	R
Integration Point Solution	LOC, PERX, PERY, PERZ, EF, EFSUM, D, DSUM	1	-
Nodal Solution	EF, EFSUM, D, DSUM	2	-

1. Output at each integration point, if KEYOPT(5) = 1

2. Output at each corner node, if KEYOPT(5) = 2

Table 122.3: SOLID122 Item and Sequence Numbers (p. 423) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 122.3: SOLID122 Item and Sequence Numbers (p. 423):

Name
output quantity as defined in the Table 122.1: SOLID122 Element Output Definitions (p. 421)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 122.3: SOLID122 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
CHRGD	SMISC	1
EFX	SMISC	2
EFY	SMISC	3
EFZ	SMISC	4
PERX	NMISC	1
PERY	NMISC	2
PERZ	NMISC	3
JTX	NMISC	5
JTY	NMISC	6
JTZ	NMISC	7
JTSUM	NMISC	8

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID122 Assumptions and Restrictions

- The element must not have a zero volume or a zero length side. This occurs most frequently when the element is not numbered properly. Elements may be numbered either as shown in Figure 122.1: SOLID122 Geometry (p. 419) or in an opposite fashion.
- An edge with a removed midside node implies that the potential varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information on the use of midside nodes.
- Degeneration to the form of pyramid should be used with caution. The element sizes, when degenerated, should be small in order to minimize the field gradients. Pyramid elements are best used as filler elements or in meshing transition zones.
- This element is only compatible with elements having a VOLT DOF and an electric charge reaction solution. Electric charge reactions must all be positive or negative. KEYOPT(6) sets the electric charge reaction sign. See Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide* for more information.
- The solenoidal current density is required for a solution, or for any postprocessing operations.

SOLID122 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Emag

- The Birth and death special feature is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID123

3-D 10-Node Tetrahedral Electrostatic Solid

MP <> <> <> <> <> EM <> PP EME <>
Product Restrictions

SOLID123 Element Description

SOLID123 is a 3-D, 10-node, charge-based electric element. It is well suited to model irregular meshes (such as produced from various CAD/CAM systems). The element has one degree of freedom, voltage, at each node.

This element is applicable to 3-D electrostatic and time-harmonic quasistatic electric field analyses. Various printout options are also available. See SOLID123 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 123.1: SOLID123 Geometry

SOLID123 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 123.1: SOLID123 Geometry (p. 425).

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems (p. 55). Properties not input default as described in Linear Material Properties in the *Material Reference*. Nodal loads are defined with the **D** (*Lab* = VOLT) and **F** (*Lab* = CHRG) commands.

Element loads are described in Nodal Loading (p. 49). Surface charge densities may be input as surface loads at the element faces as shown by the circled numbers on Figure 123.1: SOLID123 Geometry (p. 425). Charge densities may be input as element body loads at the nodes. If the node I charge density CHRGD(I) is input, and all others are unspecified, they default to CHRGD(I). If all corner node charge densities are specified, each midside node charge density defaults to the average charge density of its adjacent corner nodes.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The temperature (for material property evaluation only) body loads may be input based on their value at the element's nodes or as a single element value [**BF**, **BFE**]. In general, unspecified nodal values of temperatures default to the uniform value specified with the **BFUNIF** or **TUNIF** commands.

A summary of the element input is given in "SOLID123 Input Summary" (p. 426). A general description of element input is given in Element Input (p. 43).

SOLID123 Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R

Degrees of Freedom
VOLT

Real Constants
None

Material Properties
MP command: PERX, PERY, PERZ, LSST, RSVX, RSVY, RSVZ

EMUNIT command: EPZRO

Surface Loads

Surface charge densities --
CHRGS face 1 (J-I-K), face 2 (I-J-L), face 3 (J-K-L), face 4 (K-I-L)

Body Loads

Temperature --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P), T(Q), T(R)

Volume charge densities --

CHRGD(I), CHRGD(J), CHRGD(K), CHRGD(L), CHRGD(M),
CHRGD(N), CHRGD(O), CHRGD(P), CHRGD(Q), CHRGD(R)

Special Features
Birth and death

KEYOPT(4)
Element coordinate system defined:

0 --
Element coordinate system is parallel to the global coordinate system

1 --
Element coordinate system is based on the element I-J side

KEYOPT(5)
Extra element output:

0 --
Basic element printout

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
Repeat basic solution for all integration points

2 --
Nodal fields printout

KEYOPT(6)
Electric charge reaction sign:

0 --
Positive

1 --
Negative

SOLID123 Output Data

The solution output associated with the element is in two forms:

- Nodal potentials included in the overall nodal solution
- Additional element output as shown in Table 123.1: SOLID123 Element Output Definitions (p. 427)

The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50) in the *Element Reference*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 123.1: SOLID123 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	2
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P), T(Q), T(R)	Y	Y
LOC	Output location (X, Y, Z)	1	1
PERX, PERY, PERZ	Electric relative permittivity	1	1
EF:X, Y, Z	Electric field components (X, Y, Z)	1	1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
EF:SUM	Vector magnitude of EF	1	1
D:X, Y, Z	Electric flux density components	1	1
D:SUM	Vector magnitude of D	1	1
JS:X, Y, Z, SUM	Current density components (in the global Cartesian coordinate system) and vector magnitude [3]	1	1
JT:X, Y, Z, SUM	Conduction current density components and magnitude [3]	1	1
JHEAT:	Joule heat generation rate per unit volume [4] [6] [5]	1	1
SENE:	Stored electric energy [5]	1	1
FMAG:X, Y, Z	Electrostatic force [7]	-	1
CHRGD	Applied charge density	-	Y

1. The solution value is output only if calculated (based upon input data). The element solution is at the centroid.

2. Available only at centroid as a ***GET** item.

3. For a time-harmonic analysis, JS represents the sum of element conduction and displacement current densities. JT represents the element conduction current density. The element displacement current density (JD) can be derived from JS and JT as JD=JS-JT. JS can be used as a source current density for a subsequent magnetostatic analysis with companion elements [**LDREAD**].

4. For a time-harmonic analysis, calculated Joule heat generation rate per unit volume (JHEAT) includes conduction heating and dielectric heating due to the loss tangent.

5. Calculated Joule heat generation rate per unit volume (JHEAT) may be made available for a subsequent thermal analysis with companion elements [**LDREAD**].

6. For a time-harmonic analysis, Joule losses (JHEAT) and stored energy (SENE) represent time-average values. These values are stored in both the real and imaginary data sets.

7. Use the **EMFT** macro to calculate the force distribution over the body. See the discussion on Electrostatic Forces in the *Low-Frequency Electromagnetic Analysis Guide*.

Table 123.2: SOLID123 Miscellaneous Element Output

Description	Names of Items Output	O	R
Integration Point Solution	LOC, PERX, PERY, PERZ, EF, EFSUM, D, DSUM	1	-
Nodal Solution	EF, EFSUM, D, DSUM	2	-

1. Output at each integration point, if KEYOPT(5) = 1

2. Output at each corner node, if KEYOPT(5) = 2

Table 123.3: SOLID123 Item and Sequence Numbers (p. 429) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 123.3: SOLID123 Item and Sequence Numbers (p. 429):

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name
output quantity as defined in the Table 123.1: SOLID123 Element Output Definitions (p. 427)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 123.3: SOLID123 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
CHRGD	SMISC	1
EFX	SMISC	2
EFY	SMISC	3
EFZ	SMISC	4
PERX	NMISC	1
PERY	NMISC	2
PERZ	NMISC	3
JTX	NMISC	5
JTY	NMISC	6
JTZ	NMISC	7
JTSUM	NMISC	8

SOLID123 Assumptions and Restrictions

- The element must not have a zero volume or a zero length side. This occurs most frequently when the element is not numbered properly. Elements may be numbered either as shown in Figure 123.1: SOLID123 Geometry (p. 425) or in an opposite fashion.
- An edge with a removed midside node implies that the potential varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information on the use of midside nodes.
- This element is only compatible with elements having a VOLT DOF and an electric charge reaction solution. Electric charge reactions must all be positive or negative. KEYOPT(6) sets the electric charge reaction sign. See Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide* for more information.
- The solenoidal current density is required for a solution, or for any postprocessing operations.

SOLID123 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Emag 3-D

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The birth and death special feature is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CIRCU124

Electric Circuit

MP ME <> <> <> <> EM <> PP EME MFS
Product Restrictions

CIRCU124 Element Description

CIRCU124 is a general circuit element applicable to circuit simulation. The element may also interface with electromagnetic finite elements to simulate coupled electromagnetic-circuit field interaction. The element has up to 6 nodes to define the circuit component and up to three degrees of freedom per node to model the circuit response. For electromagnetic-circuit field coupling, the element may interface with PLANE53 and SOLID97 using coupled circuit source options KEYOPT(1) = 5, 6, or 7 (see Figure 124.3: CIRCU124 Coupled Circuit Source Options (p. 433)). CIRCU124 can be directly coupled to the electromagnetic or stranded coil analysis options of PLANE233, SOLID236, and SOLID237 through the VOLT degree of freedom. CIRCU124 is applicable to static, harmonic, and transient analyses.

CIRCU124 Input Data

The geometry, node definition, and degree of freedom options are shown in Figure 124.1: CIRCU124 Circuit Element Options (p. 432) (circuit components), Figure 124.2: CIRCU124 Circuit Source Options (p. 432) (circuit source options), and Figure 124.3: CIRCU124 Coupled Circuit Source Options (p. 433) (coupled circuit source options). The element is defined by *active* and *passive* circuit nodes. *Active* nodes are those connected to an overall electric circuit, and *passive* nodes are those used internally by the element and not connected to the circuit. For the coupled circuit source options, the passive nodes are actual nodes of a source conductor modeled in the electromagnetic field domain.

Element circuit components, sources, and coupled sources are defined by KEYOPT(1) settings and its corresponding real constants. Real constant input is dependent on the element circuit option used. A summary of the element input options is given in "CIRCU124 Input Summary" (p. 433). Real constants numbers 15 and 16 are created by the GUI Circuit Builder (see the *Modeling and Meshing Guide*), and are not required input for analysis purposes. The element is characterized by up to three degrees of freedom:

- VOLT (voltage)
- CURR (current)
- EMF (potential drop)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 124.1: CIRCU124 Circuit Element Options

Figure 124.2: CIRCU124 Circuit Source Options

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 124.3: CIRCU124 Coupled Circuit Source Options

Independent voltage and current sources (KEYOPT(1) = 3 or 4) may be excited by AC/DC, sinusoidal, pulse, exponential, or piecewise linear load functions as defined by KEYOPT(2); see Figure 124.4: Load Functions and Corresponding Real Constants for Independent Current and Voltage Sources (p. 436).

The time-step size for a transient analysis is controlled by the **DELTIM** or **NSUBST** commands. The CIRCU124 element does not respond to automatic time stepping (**AUTOTS** command), but **AUTOTS** can be used as a mechanism for ramping the time step to its final value. For coupled electromagnetic-circuit problems, automatic time stepping may be used if controls are placed on degrees of freedom other than VOLT, CURR, or EMF, or loads associated with those degrees of freedom.

For problems using the CIRCU124 element, the sparse direct solver is chosen by default.

CIRCU124 Input Summary

Nodes

I, J, K, L, M, N

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Degrees of Freedom

VOLT, CURR, EMF (see Figure 124.1: CIRCU124 Circuit Element Options (p. 432))

Real Constants

Dependent on KEYOPT(1) and KEYOPT(2) settings. See Table 124.1: CIRCU124 Real Constants (p. 435) for details.

Material Properties

None

Surface Loads

None

Body Loads

See KEYOPT(2)

Special Features

Magnetic field coupling

KEYOPT(1)

Circuit component type:

0 --
Resistor

1 --
Inductor

2 --
Capacitor

3 --
Independent Current Source

4 --
Independent Voltage Source

5 --
Stranded Coil Current Source

6 --
2-D Massive Conductor Voltage Source

7 --
3-D Massive Conductor Voltage Source

8 --
Mutual Inductor

9 --
Voltage-Controlled Current Source

10 --
Voltage-Controlled Voltage Source

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11 --
Current-Controlled Voltage Source

12 --
Current-Controlled Current Source

KEYOPT(2)
Body loads available if KEYOPT(1) = 3 or 4:

0 --
DC or AC Harmonic load

1 --
Sinusoidal load

2 --
Pulse load

3 --
Exponential load

4 --
Piecewise Linear load

Table 124.1: CIRCU124 Real Constants

Circuit Option and Graphics Label	KEYOPT(1)	Real Constants
Resistor (R)	0	R1 = Resistance (RES)
Inductor (L)	1	R1 = Inductance (IND) R2 = Initial inductor current (ILO)
Capacitor (C)	2	R1 = Capacitance (CAP) R2 = Initial Capacitor Voltage (VCO)
Mutual Inductor (K)	8	R1 = Primary Inductance (IND1) R2 = Secondary Inductance (IND2) R3 = Coupling Coefficient (K)
Independent Current Source (I)	3	For KEYOPT(2) = 0: R1 = Amplitude (AMPL) R2 = Phase angle (PHAS) For KEYOPT(2) > 0: see Figure 124.4: Load Functions and Corresponding Real Constants for Independent Current and Voltage Sources (p. 436).
Voltage-Controlled Current Source (G)	9	R1 = Transconductance (GT)
Current-Controlled Current Source (F)	12	R1 = Current Gain (AI)
Independent Voltage Source (V)	4	For KEYOPT(2) = 0: R1 = Amplitude (AMPL) R2 = Phase angle (PHAS)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Circuit Option and Graphics Label	KEYOPT(1)	Real Constants
		For KEYOPT(2) > 0: see Figure 124.4: Load Functions and Corresponding Real Constants for Independent Current and Voltage Sources (p. 436).
Voltage-Controlled Voltage Source (E)	10	R1 = Voltage Gain (AV)
Current-Controlled Voltage Source (H)	11	R1 = Transresistance (RT)
Stranded Coil Current Source (N)	5 [1]	R1 = Symmetry Factor (SCAL)
2-D Massive Conductor Voltage Source (M)	6 [1]	R1 = Symmetry Factor (SCAL)
3-D Massive Conductor Voltage Source (P)	7 [1]	R1 = Symmetry Factor (SCAL)

1. Applies only to a FEA domain modeled with PLANE53 or SOLID97 elements.

Note

For all above Circuit options, the GOFFST and ID real constants (numbers 15 and 16) are created by the Circuit Builder automatically:

Figure 124.4: Load Functions and Corresponding Real Constants for Independent Current and Voltage Sources

(a) Sinusoidal load, KEYOPT(2) = 1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(b) Pulse load, KEYOPT(2) = 2

(c) Exponential load, KEYOPT(2) = 3

(d) Piecewise linear load, KEYOPT(2) = 4

CIRCU124 Output Data

The element output for this element is dependent on the circuit option selected. Table 124.2: CIRCU124 Element Output Definitions (p. 438) summarizes the element output data.

The Element Output Definitions table uses the following notation:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 124.2: CIRCU124 Element Output Definitions

Name	Definition	O	R
For KEYOPT(1) = 0: Resistor			
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
RES	Resistance	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current	Y	Y
POWER	Power loss	Y	Y
For KEYOPT(1) = 1: Inductor			
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
IND	Inductance	Y	Y
IL0	Initial current	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current	Y	Y
POWER	Power absorption	Y	Y
For KEYOPT(1) = 2: Capacitor			
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
CAP	Capacitance	Y	Y
VC0	Initial voltage	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current	Y	Y
POWER	Power absorption	Y	Y
For KEYOPT(1) = 3: Independent Current Source			
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
CURRENT SOURCE	Real or imaginary component of applied current	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current	Y	Y
POWER	Power (loss if positive, output if negative)	Y	Y
For KEYOPT(1) = 4: Independent Voltage Source			

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K	Y	Y
VOLTAGE SOURCE	Real or imaginary component of applied voltage	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current at node K	Y	Y
POWER	Power (loss if positive, output if negative)	Y	Y
For KEYOPT(1) = 5: Stranded Coil Current Source			
EL	Element Number	Y	Y
NODES	Nodes - I, J, K	Y	Y
SCAL	Scaling factor defining voltage symmetry in 2-D or 3-D analyses	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current at node K	Y	Y
POWER	Power loss or absorption	Y	Y
For KEYOPT(1) = 6: 2-D Massive Conductor Voltage Source			
EL	Element Number	Y	Y
NODES	Nodes - I, J, K	Y	Y
SCAL	Scaling factor defining voltage symmetry in 2-D or 3-D analyses	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current at node K	Y	Y
POWER	Power loss or absorption	Y	Y
For KEYOPT(1) = 7: 3-D Massive Conductor Voltage Source			
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
SCAL	Scaling factor defining voltage symmetry in 2-D or 3-D analyses	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current at node K and L	Y	Y
CONTROL VOLT	Voltage drop between node K and node L	Y	Y
POWER	Power loss or absorption	Y	Y
For KEYOPT(1) = 8: 3-D Mutual Inductor (Transformer)			
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
IND1	Primary inductance	Y	Y
IND2	Secondary inductance	Y	Y
INDM	Mutual inductance	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
CURRENT	Current in I-J branch	Y	Y
CONTROL VOLT	Voltage drop between node K and node L	Y	Y
CONTROL CURR	Current in K-L branch	Y	Y
POWER	Power absorption	Y	Y
For KEYOPT(1) = 9: Voltage Controlled Current Source			
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
GT	Transconductance	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current in I-J branch	Y	Y
CONTROL VOLT	Voltage drop between node K and node L	Y	Y
POWER	Power (loss if positive, output if negative)	Y	Y
For KEYOPT(1) = 10: Voltage Controlled Voltage Source			
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M	Y	Y
AV	Voltage gain	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current at node K	Y	Y
CONTROL VOLT	Voltage drop between node L and node M	Y	Y
POWER	Power (loss if positive, output if negative)	Y	Y
For KEYOPT(1) = 11: Current Controlled Voltage Source			
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N	Y	Y
GT	Transresistance	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current at node K	Y	Y
CONTROL VOLT	Voltage drop between node L and node M	Y	Y
CONTROL CURR	Current at node N	Y	Y
POWER	Power (loss if positive, output if negative)	Y	Y
For KEYOPT(1) = 12: Current Controlled Current Source			
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N	Y	Y
AI	Current gain	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current at node K	Y	Y
CONTROL VOLT	Voltage drop between node L and node M	Y	Y
CONTROL CURR	Current at node N	Y	Y
POWER	Power (loss if positive, output if negative)	Y	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Table 124.3: CIRCU124 Item and Sequence Numbers (p. 441) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 124.3: CIRCU124 Item and Sequence Numbers (p. 441):

Name
output quantity as defined in the Table 124.2: CIRCU124 Element Output Definitions (p. 438)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 124.3: CIRCU124 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
VOLTAGE	SMISC	1
CURRENT	SMISC	2
CONTROL VOLT	SMISC	3
CONTROL CURR	SMISC	4
POWER	NMISC	1
SOURCE (real)	NMISC	2
SOURCE (imaginary)	NMISC	3

CIRCU124 Assumptions and Restrictions

- For static analyses, a capacitor circuit element is treated as an open-circuit and an inductor circuit element is treated as a short-circuit.
- Only MKS units are allowed (**EMUNIT** command).
- The resistor, inductor, capacitor, independent current source, and mutual inductor circuit options produce symmetric coefficient matrices while the remaining options produce unsymmetric matrices.
- The sparse solver is the default for problems using the CIRCU124 element. Even if you choose a different solver, ANSYS will switch to the sparse solver when CIRCU124 elements are present.
- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, the elements must have the same reaction force (see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*).
- KEYOPT(1) = 5, 6 or 7 should not be used with PLANE233, SOLID236, or SOLID227. These elements can be coupled to CIRCU124 components directly through the VOLT degree of freedom.

CIRCU124 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

CIRCU125

Diode

MP ME <> <> <> <> EM <> PP EME MFS
Product Restrictions

CIRCU125 Element Description

CIRCU125 is a diode element normally used in electric circuit analysis. The element may also interface with electromagnetic and mechanical finite elements to simulate fully coupled electromechanical analyses at the lumped parameter level. The element has 2 nodes to define the circuit component and one degree of freedom per node to model the circuit response. The element may interface with the electric circuit element CIRCU124, with the mechanical elements MASS21, COMBIN14, and COMBIN39, and with the electromechanical transducer element TRANS126. CIRCU125 is applicable to static analyses and transient analyses with restart.

CIRCU125 Input Data

The geometry, node definition, and degree of freedom options are shown in Figure 125.1: CIRCU125 Element Options (p. 443).

The diode element is defined by the KEYOPT(1) setting and its corresponding real constants. Real constant input is dependent on the diode option used. A summary of the element input options is given in "CIRCU125 Input Summary" (p. 444). Real constants numbers 1 and 2 are created by the GUI Circuit Builder (see the *Modeling and Meshing Guide*), and are not required input for analysis purposes. The element is characterized by one degree of freedom, VOLT (voltage).

Figure 125.1: CIRCU125 Element Options

The I-U characteristics of the diodes are approximated by the piecewise linear functions shown in Figure 125.2: CIRCU125 I-U Characteristics (p. 444). The characteristic of a common (non-Zener) diode consists of line segments corresponding to the closed and open states. The characteristic of a Zener diode consists of three segments corresponding to the Zener, closed, and open states. The diode characteristic can be ideal or lossy depending on the values of the real constants.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 125.2: CIRCU125 I-U Characteristics

Legend
VLTF - Forward voltage
VLTZ - Zener voltage
RESF - Slope is forward resistance
RESB - Slope is blocking resistance
RESZ - Slope is Zener resistance

CIRCU125 Input Summary

Nodes

I, J

Degrees of Freedom

VOLT

Real Constants

Dependent on KEYOPT(1) settings.
For KEYOPT(1) = 0:
GOFFST, ID, (blank), RESF, VLTF, RESB,
(blank), (blank)
For KEYOPT(1) = 1:
GOFFST, ID, (blank), RESF, VLTF, RESB,
RESZ, VLTZ
See Table 125.1: CIRCU125 Real Constants (p. 445).

Material Properties

None

Surface Loads

None

Body Loads

None

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Special Features

None

KEYOPT(1)

Select diode options:

0 --

Common Diode

1 --

Zener Diode

If you are using the Circuit Builder to construct your model, the real constants GOFFST and ID are provided automatically.

Note

The real constant default values may not be appropriate to analyze micro devices (i.e., devices with extremely small dimensions) in MKSA units.

Table 125.1: CIRCU125 Real Constants

Real Constant No.	Name	Description
Common Diode (D) (KEYOPT(1) = 0)		
1	GOFFST	Graphical offset
2	ID	Element identification number
3	--	(blank)
4	RESF	Forward resistance (if not entered, defaults to 1.0e-12 Ohm)
5	VLTF	Forward voltage (if not entered, defaults to 0.0e0 Volt)
6	RESB	Blocking resistance (if not entered, defaults to 1.0e+12 Ohm)
7, 8	--	(blank)
Zener Diode (Z) (KEYOPT(1) = 2) - use real constants 1 through 6 as above, then:		
7	RESZ	Zener resistance (if not entered, defaults to 1.0e+12 Ohm)
8	VLTZ	Zener voltage (if not entered, defaults to 1.0e-12 Volt)

CIRCU125 Solution Considerations

CIRCU125 is a highly nonlinear element. To obtain convergence, you may have to define convergence criteria, instead of using the default values. Use **CNVTOL**,VOLT,,0.001,2,1.0E-6 if you need to change the convergence criteria.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CIRCU125 Output Data

The element output for this element is dependent on the circuit option selected. Table 125.2: CIRCU125 Element Output Definitions (p. 446) summarizes the element output data.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 125.2: CIRCU125 Element Output Definitions

Name	Definition	O	R
For KEYOPT(1) = 0: Common Diode			
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
REST	Tangent Resistance	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current	Y	Y
POWER	Power loss	Y	Y
STAT	Diode status	1	1
DYNRES	Dynamic resistance at operating point	Y	Y
AMPGEN	Norton equivalent current generator	Y	Y
For KEYOPT(1) = 1: Zener Diode			
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
REST	Tangent resistance	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current	Y	Y
POWER	Power loss	Y	Y
STAT	Diode status	2	2
DYNRES	Dynamic resistance at operating point	Y	Y
AMPGEN	Norton equivalent current generator	Y	Y

1. Common Diode Status Values

   1 - Forward, open

   2 - Reverse, blocked

2. Zener Diode Status Values

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 - Forward, open

2 - Reverse, blocked

3 - Zener, breakdown

Table 125.3: CIRCU125 Item and Sequence Numbers (p. 447) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 125.3: CIRCU125 Item and Sequence Numbers (p. 447):

Name
output quantity as defined in Table 125.2: CIRCU125 Element Output Definitions (p. 446)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 125.3: CIRCU125 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
VOLTAGE	SMISC	1
CURRENT	SMISC	2
POWER	NMISC	1
blank	NMISC	2
blank	NMISC	3
DYNRES	NMISC	4
AMPGEN	NMISC	5
STAT	NMISC	6

CIRCU125 Assumptions and Restrictions

- If either the Zener voltage or Zener resistance is blank or very small, the Zener diode will be replaced with a common diode, and a warning will be issued.
- Only MKS units are allowed (**EMUNIT** command).
- If the Zener Voltage is entered as a positive number, the element will negate the value that is entered. If the Forward Voltage is entered as a negative number, the element will replace it with its absolute value. All resistance must be positive. Any negative resistance value is replaced by its absolute value.
- The element issues an error message if applied in harmonic analysis.
- This element does not work with the CIRCU94 piezoelectric element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, the elements must have the same reaction force (see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*).
- This element cannot be used in a distributed solution.

CIRCU125 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TRANS126

Electromechanical Transducer

MP <> <> <> <> <> <> <> PP EME <>
Product Restrictions

TRANS126 Element Description

TRANS126 represents a transducer element that converts energy from an electrostatic domain into a structural domain (and vice versa), while also allowing for energy storage. The element fully couples the electromechanical domains and represents a reduced-order model suitable for use in structural finite element analysis as well as electromechanical circuit simulation. The element has up to two degrees of freedom at each node: translation in the nodal x, y, or z direction and electric potential (VOLT). The element is suitable for simulating the electromechanical response of micro-electromechanical devices (MEMS) such as electrostatic comb drives, capacitive transducers, and RF switches for example.

The characteristics of the element are derived from electrostatic field simulations of the electromechanical device using the electrostatic elements PLANE121, SOLID122, and SOLID123, as well as the **CMATRIX** macro. The TRANS126 element represents the capacitive response of the device to motion in one direction. Running a series of electrostatic simulations and extracting capacitance (**CMATRIX** command) as a function of stroke (or deflection) provides the necessary input for this element. The capacitance versus stroke represents a "reduced-order" characterization of the device suitable for simulation in this transducer element. Up to three characterizations (in X, Y, or Z) can be made from sets of electrostatic simulations to create three independent transducer elements to characterize a full translational response of the device. See TRANS126 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 126.1: TRANS126 Geometry

TRANS126 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 126.1: TRANS126 Geometry (p. 449). Nodes I and J define the element. The nodes need not be coin-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

cident. The element may lie along any one of the three global Cartesian axes as shown in Figure 126.1: TRANS126 Geometry (p. 449), or it may exist in any arbitrary coordinate system as long as the nodes are rotated into the arbitrary coordinate system in such a manner that one of the axes lies along the element's I-J direction. Use the degree of freedom option (KEYOPT(2)) to select the appropriate structural displacement degree of freedom (corresponding to the element's I-J direction) and electric potential. Orientation of the element with respect to nodal displacements (node J relative to node I) is critical. Orient the element such that a positive movement of node J relative to node I produces a positive displacement (see Figure 126.1: TRANS126 Geometry (p. 449)). Figure 126.4: TRANS126 Valid/Invalid Orientations (p. 457) illustrates valid and invalid orientations of the element for a UX-VOLT degree of freedom set.

The capacitance vs. stroke data for the element is entered through the real constant table. Use KEYOPT(3) to select from two different methods of input. For KEYOPT(3) = 0, the real constant data (R7-R11) represent the coefficients of an equation (see Figure 126.2: TRANS126 Capacitance Relationship (p. 450)). Use as many terms as are required to represent the curve. For KEYOPT(3) = 1, the real constant data (R7-R46) represent discrete pairs of capacitance and stroke data. Up to 20 pairs of data may be input. The minimum required is 5 data point sets. A curve is fit to the discrete data sets represented by the equation shown in Figure 126.2: TRANS126 Capacitance Relationship (p. 450).

Figure 126.2: TRANS126 Capacitance Relationship

Capacitance vs. Stroke Representation

The initial gap distance GAP (R3) represents the initial distance between conducting walls of the electromechanical device (that is, plates of a parallel capacitor, beams of a comb drive, etc.). The initial gap value should fall within the range of the capacitance vs. stroke data as shown in Figure 126.2: TRANS126 Capacitance Relationship (p. 450). The minimum gap distance GAPMIN (R4) represents the physical location where the gap is closed. If the gap closes to GAPMIN, the element behaves like a contact element with a normal stiffness KN represented by real constant R5. GAP and GAPMIN default to near-zero if not defined. Figure 126.3: TRANS126 Force Relationship (p. 451) illustrates the force vs. stroke for the transducer element. The curve highlights the capacitive force (which is compressive and acts to close

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the gap), and the contact force (which restrains the motion once the gap reaches GAPMIN). KN defaults to a stiffness represented by the slope from the capacitive force at GAPMIN to the origin as shown in Figure 126.3: TRANS126 Force Relationship (p. 451).

Figure 126.3: TRANS126 Force Relationship

The element supports nodal voltage and displacements (**D**) as well as nodal current and force (**F**). Use **IC** to input an initial starting value of voltage or displacement for a transient analysis, or an initial guess for a static analysis. The element produces an unsymmetric matrix. The element supports static, prestressed harmonic, transient, and prestressed modal analysis. Prestress effects must be applied for both modal and harmonic analysis (for example, a prestress static analysis with an applied DC voltage, followed by a small-signal (AC voltage) harmonic analysis, or a prestress static analysis with an applied DC voltage followed by a modal analysis). The element is nonlinear for static and transient analysis and requires an iterative solution to converge. The element supports tension only.

The transducer element by nature has both stable and unstable solutions. If the system stiffness is negative, convergence problems can occur near unstable solutions. This typically occurs at small gap distances near GAPMIN. Use KEYOPT(6) = 1 to select the augmented stiffness method if you encounter convergence problems. In this method, the electrostatic stiffness is set to zero to guarantee a positive system stiffness. After convergence is reached, the electrostatic stiffness is automatically reestablished for postprocessing and subsequent analyses.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The next table summarizes the element input. Element Input (p. 43) gives a general description of element input.

TRANS126 Input Summary

Nodes
I, J

Degrees of Freedom
UX-VOLT, UY-VOLT, OR UZ-VOLT

Real Constants

If KEYOPT(3) = 0, then:
GOFFST, EID, GAP, GAPMIN, KN, (Blank),
C0, C1, C2, C3, C4

If KEYOPT(3) = 1, then:
GOFFST, EID, GAP, GAPMIN, KN, (Blank),
GAP1, CAP1, GAP2, CAP2, ..., GAP20, CAP20

See Table 126.1: TRANS126 Real Constants (p. 453) for details.

Material Properties
None

Surface Loads
None

Body Loads
None

Special Features

Nonlinearity
Prestress

KEYOPT(2)
Select degree-of-freedom set:

0,1 --
UX-VOLT

2 --
UY-VOLT

3 --
UZ-VOLT

KEYOPT(3)
Capacitance-Gap option:

0 --
Use capacitance-gap curve input coefficients: C0, C1, C2, C3, and C4

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
Use capacitance versus gap data points: GAP1, CAP1, GAP2, CAP2 ... GAP20, CAP20

KEYOPT(6)
Stiffness method:

0 --
Full stiffness method (default)

1 --
Augmented stiffness method

The first six real constants for this element are the same, whether you set KEYOPT(3) = 0 or 1. From number 7 on, the real constants differ between the two settings, as shown in the table below.

Table 126.1: TRANS126 Real Constants

Number	Name	Description
Basic Set		
1	GOFFST	Graphical offset
2	EID	ID number
3	GAP	Initial gap
4	GAPMIN	Minimal gap
5	KN	Gap Normal Stiffness
6	(blank)	unused
For KEYOPT(3) = 0; Capacitance (Cap) vs. gap (x) function: Cap = C0/x + C1 + C2*x + C3*x**2 + C4*x**3		
7	C0	Equation constant C0
8	C1	Equation constant C1
9	C2	Equation constant C2
10	C3	Equation constant C3
11	C4	Equation constant C4
For KEYOPT(3) = 1 (Capacitance-gap curve data)		
7	GAP1	Gap 1
8	CAP1	Capacitance 1
9, ..., 46	GAP2, CAP2, ..., GAP20, CAP20	Gap2 and Capacitance 2 through Gap 20 and Capacitance 20

TRANS126 Output Data

The solution output associated with the element is shown in Table 126.2: TRANS126 Element Output Definitions (p. 454).

The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50) in the *Element Reference*. See the *Basic Analysis Guide* for ways to view results.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If this element is used in a harmonic analysis, all variables will be stored in two-column arrays as complex variables. The first column will be titled real component and the second column will be titled imaginary component. If the variable is not complex, the same value will be stored in both columns.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 126.2: TRANS126 Element Output Definitions

Name	Definition	O	R
EL	Element number	Y	Y
NODES	Nodes - I, J	Y	Y
EFORCE	Electrostatic Force	Y	Y
ESTIFF	Electrostatic stiffness (dEFORCE/dU)	Y	Y
CONDUCT	Motion conductance (dCap/dU) (RELVEL)	Y	Y
DVDT	Time rate of change of Voltage (dVOLT/dt)	Y	Y
RELDISP	Relative displacement node I to node J	Y	Y
RELVEL	Relative velocity node I to node J	Y	Y
VOLTAGE	Voltage drop between node I and node J	Y	Y
CURRENT	Current	Y	Y
CAP	Capacitance	Y	Y
MECHPOWER	Mechanical power, (force x velocity)	Y	Y
ELECPOWER	Electrical power, (voltage drop x current)	Y	Y
CENERGY	Electrostatic energy stored in capacitor	Y	Y
GAP	Actual gap, U_J - U_I + GAP (nominal) (real constant input)	Y	Y
KUU	Coupled system stiffness, dF/dU	Y	Y
KUV	Coupled system stiffness, dF/dV	Y	Y
KVU	Coupled system stiffness, dI/dU	Y	Y
KVV	Coupled system stiffness, dI/dV	Y	Y
DUU	Coupled system damping, dF/dVEL	Y	Y
DUV	Coupled system damping, dF/dVRATE	Y	Y
DVU	Coupled system damping, dI/dVEL	Y	Y
DVV	Coupled system damping, dI/dVRATE	Y	Y
DISPR, DISPI	Real and imaginary components of displacement	1	1
FORCR, FORCI	Real and imaginary components of electrostatic force	1	1
VOLTR, VOLTI	Real and imaginary components of voltage drop	1	1
CURRR, CURRI	Real and imaginary components of current	1	1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. The item is only available for prestress harmonic analysis.

Table 126.3: TRANS126 Item and Sequence Numbers (p. 455) lists output available through **ETABLE** using the Sequence Number method. See Element Table for Variables Identified By Sequence Number in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 126.3: TRANS126 Item and Sequence Numbers (p. 455):

Name
output quantity as defined in the Table 126.3: TRANS126 Item and Sequence Numbers (p. 455)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 126.3: TRANS126 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
MECH-POWER	SMISC	1
ELECPOWER	SMISC	2
CENERGY	SMISC	3
GAP	NMISC	1
RELVEL	NMISC	2
EFORCE	NMISC	3
VOLTAGE	NMISC	4
DVDT	NMISC	5
CURRENT	NMISC	6
CAP	NMISC	7
ESTIFF	NMISC	8
UCT	NMISC	9
KUU	NMISC	10
KUV	NMISC	11
KVU	NMISC	12
KVV	NMISC	13
DUU	NMISC	14
DUV	NMISC	15
DVU	NMISC	16
DVV	NMISC	17
DISPR	NMISC	18
DISPI	NMISC	19
FORCR	NMISC	20

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FORCI	NMISC	21
VOLTR	NMISC	22
VOLTI	NMISC	23
CURRR	NMISC	24
CURRI	NMISC	25

TRANS126 Assumptions and Restrictions

- The transducer element must be aligned such that the element I-J direction points along the active structural degree of freedom in the nodal coordinate system. In addition, a positive movement in the nodal coordinate system of node J relative to node I should act to open the gap (Stroke = GAP + U_j - U_i). Figure 126.4: TRANS126 Valid/Invalid Orientations (p. 457) illustrates valid and invalid orientations of the element for a UX-VOLT degree of freedom set.

- Nodes I and J may be coincident since the orientation is defined by the relative motion of node J to node I. No moment effects due to noncoincident nodes are included. That is, if the nodes are offset from a line perpendicular to the element axis, moment equilibrium may not be satisfied.

- Unreasonable high stiffness (KN) values should be avoided. The rate of convergence decreases as the stiffness increases.

- The element may not be deactivated with **EKILL**.

- Harmonic and modal analyses are valid only for small-signal analyses after a static prestress calculation.

- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, the elements must have the same reaction force (see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*).

- A minimum of two load steps must be used to obtain valid electrostatic force calculations.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 126.4: TRANS126 Valid/Invalid Orientations

Valid orientations for UX,VOLT DOF set:

Invalid orientations for UX,VOLT DOF set:

TRANS126 Product Restrictions

The TRANS126 element is only available in the ANSYS Multiphysics and ANSYS PrepPost products.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

FLUID129

2-D Infinite Acoustic

MP ME <> <> <> <> <> <> PP EME MFS
Product Restrictions

FLUID129 Element Description

FLUID129 has been developed as a companion element to FLUID29. It is intended to be used as an envelope to a model made of FLUID29 finite elements. It simulates the absorbing effects of a fluid domain that extends to infinity beyond the boundary of FLUID29 finite element domain. FLUID129 realizes a second-order absorbing boundary condition so that an outgoing pressure wave reaching the boundary of the model is "absorbed" with minimal reflections back into the fluid domain. The element can be used to model the boundary of 2-D (planar or axisymmetric) fluid regions and as such, it is a line element; it has two nodes with one pressure degree of freedom per node. FLUID129 may be used in transient, harmonic, and modal analyses. Typical applications include structural acoustics, noise control, underwater acoustics, etc. See FLUID129 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 129.1: FLUID129 Geometry

FLUID129 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 129.1: FLUID129 Geometry (p. 459). The element is defined by two nodes (I, J), the material properties and the real constants (defined in "FLUID129 Input Summary" (p. 460)). The element must be circular with radius RAD and center located at or near the center of the structure. The radius RAD should be supplied through the real constants.

The element is characterized by a pair of symmetric stiffness and damping matrices.

In a typical meshing procedure, you should mesh the interior fluid domain that is bounded by a circular boundary with FLUID29 elements, select the nodes on the circular boundary, select the type associated with the FLUID129 and then issue the **ESURF** command. The latter will automatically add the FLUID129 elements on the boundary of the finite domain.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FLUID129 Input Summary

Nodes
I, J

Degrees of Freedom
PRES

Real Constants

RAD - Radius
X_0 - Center of enclosing circle, X value
Y_0 - Center of enclosing circle, Y value

Material Properties
MP command: SONC (velocity of sound)

Surface Loads
None

Body Loads
None

Special Features
None

KEYOPT(3)
Element behavior:

0 --
Planar

1 --
Axisymmetric

FLUID129 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in Table 129.1: FLUID129 Element Output Definitions (p. 461)

A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 129.1: FLUID129 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
MAT	Material number	Y	Y
LINE:	Length	Y	Y
XC, YC	Location where results are reported	Y	1
SONC	Speed of sound	Y	Y

1. Available only at centroid as a ***GET** item.

Table 129.2: FLUID129 Item and Sequence Numbers (p. 461) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 129.2: FLUID129 Item and Sequence Numbers (p. 461):

Name
output quantity as defined in the Table 129.1: FLUID129 Element Output Definitions (p. 461)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 129.2: FLUID129 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
SONC	NMISC	1

FLUID129 Assumptions and Restrictions

- FLUID129 must lie on a boundary circular in shape and should completely enclose the domain meshed with FLUID29 elements.
- The radius RAD of the circular boundary of the finite domain should be specified as a real constant. If the coordinates (X_0, Y_0) of the center of the circle are not supplied through the real constant input, the center will be assumed to be at the origin. The center of the circle should be as close to the center of the model as possible.
- It is recommended that the enclosing circular boundary is placed at a distance of at least 0.2*lambda from the boundary of any structure that may be submerged in the fluid, where lambda = c/f is the dominant wavelength of the pressure waves; c is the speed of sound (SONC) in the fluid, and f is the dominant fre-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

quency of the pressure wave. For example, in the case of a submerged circular cylindrical shell of diameter D, the radius of the enclosing boundary, RAD, should be at least (D/2) + 0.2*lambda.

- FLUID129 uses an extra DOF, labeled XTR1, that is not available to the user. This DOF is solely for ANSYS' internal use, although it may appear in DOF listings or in program messages.
- The only applicable modal analysis method is the Damped method.

FLUID129 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FLUID130

3-D Infinite Acoustic

MP ME <> <> <> <> <> <> PP EME MFS
Product Restrictions

FLUID130 Element Description

FLUID130 has been developed as a companion element to FLUID30, FLUID220, and FLUID221. It is intended to be used as an envelope to a model made of FLUID30, FLUID220, and FLUID221 finite elements. It simulates the absorbing effects of a fluid domain that extends to infinity beyond the boundary of the finite element domain that is made of FLUID30, FLUID220, and FLUID221 elements. FLUID130 realizes a second-order absorbing boundary condition so that an outgoing pressure wave reaching the boundary of the model is "absorbed" with minimal reflections back into the fluid domain. The element can be used to model the boundary of 3-D fluid regions and as such, it is a plane surface element; it has four nodes with one pressure degrees of freedom per node. FLUID130 may be used in transient, harmonic, and modal analyses. Typical applications include structural acoustics, noise control, underwater acoustics, etc. See FLUID30, FLUID220, and FLUID221 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 130.1: FLUID130 Geometry

FLUID130 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 130.1: FLUID130 Geometry (p. 463). The element is defined by four nodes (I, J, K, L) or eight nodes (I, J, K, L, M, N, O, P), the material property SONC (speed of sound), and the real constants shown in "FLUID130 Input Summary" (p. 464). A triangular element may be formed by defining duplicate K and L node numbers. The element must be at the spherical boundary of an acoustic fluid domain, meshed using FLUID30, FLUID220, and FLUID221 elements, with radius RAD and center located at or near the center of the structure. The radius RAD should be supplied through the real constants.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The element is characterized by a symmetric stiffness and a damping matrix.

In a typical meshing procedure the user should mesh the interior fluid domain that is bounded by a spherical boundary with FLUID30, FLUID220, and FLUID221 elements, select the nodes on the spherical boundary, select the type associated with the FLUID130 and then issue the **ESURF** command. The latter will automatically add the FLUID130 elements on the boundary of the finite domain.

FLUID130 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
PRES

Real Constants

RAD - Radius
X_0 - Center of enclosing circle, X value
Y_0 - Center of enclosing circle, Y value
Z_0 - Center of enclosing circle, Z value

Material Properties
MP command: SONC (velocity of sound), DENS (mass density)

Surface Loads
None

Body Loads
None

Special Features
None

KEYOPT(1)

0 --
Low-order element (on FLUID30 exterior surface)

2 --
High-order element (on FLUID220 or FLUID221 exterior surface)

FLUID130 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in Table 130.1: FLUID130 Element Output Definitions (p. 465)

A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 130.1: FLUID130 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
AREA:	AREA	Y	Y
XC, YC	Location where results are reported	Y	1
SONC	Speed of sound	Y	Y

1. Available only at centroid as a ***GET** item.

Table 130.2: FLUID130 Item and Sequence Numbers (p. 465) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 130.2: FLUID130 Item and Sequence Numbers (p. 465):

Name
output quantity as defined in the Table 130.1: FLUID130 Element Output Definitions (p. 465)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 130.2: FLUID130 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
SONC	NMISC	1

FLUID130 Assumptions and Restrictions

- FLUID130 must lie on a boundary spherical in shape and should completely enclose the domain meshed with FLUID30, FLUID220, and FLUID221 elements.
- The radius RAD of the spherical boundary of the finite domain should be specified as a real constant. If the coordinates (X_0, Y_0, Z_0) of the center of the sphere are not supplied through the real constant input, the center will be assumed to be at the origin of the global coordinate system. The center of the sphere should be as close to the center of the model as possible.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- It is recommended that the enclosing spherical boundary is placed at a distance of at least 0.2*lambda from the boundary of any structure that may be submerged in the fluid, where lambda = c/f is the dominant wavelength of the pressure waves. c is the speed of sound (SONC) in the fluid and f is the dominant frequency of the pressure wave. For example, in the case of a submerged spherical shell of diameter D, the radius of the enclosing boundary, RAD, should be at least (D/2) + 0.2*lambda.
- FLUID130 uses extra DOFs, labeled XTR1 and XTR2, that are not available to the user. These DOFs are solely for ANSYS' internal use, although they may appear in DOF listings or in program messages.
- The only applicable modal analysis method is the Damped method.
- FLUID130 assumes constant density and speed of sound values without losses in the "infinite" domain. Consequently, the acoustic fluid elements adjacent to FLUID130 must not have viscosity (**MP**,VISC), non-uniform acoustic media, or the Johnson-Champoux-Allard model defined.

FLUID130 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SHELL131

4-Node Thermal Shell

MP ME <> PR PRN <> <> <> PP EME MFS
Product Restrictions

SHELL131 Element Description

SHELL131 is a 3-D layered shell element having in-plane and through-thickness thermal conduction capability. The element has four nodes with up to 32 temperature degrees of freedom at each node. The conducting shell element is applicable to a 3-D, steady-state or transient thermal analysis. SHELL131 generates temperatures that can be passed to structural shell elements in order to model thermal bending. See SHELL131 in the *Mechanical APDL Theory Reference* for more details about this element.

If the model containing the conducting shell element is to be analyzed *structurally*, use an equivalent structural shell element instead, such as SHELL181 or SHELL281.

Figure 131.1: SHELL131 Geometry

x_o = element x-axis if ESYS is not supplied.

x = element x-axis if ESYS is supplied.

SHELL131 Input Data

The geometry, node locations, and coordinates systems for this element are shown in Figure 131.1: SHELL131 Geometry (p. 467). The element is defined by four nodes, one thickness per layer, a material angle for each layer, and the material properties. If the material is uniform and the analysis has no transient effects, only one layer is needed with a linear temperature variation through the thickness.

The cross-sectional properties are input using the **SECTYPE**,,SHELL and **SECDATA** commands. These properties are the thickness, material number, and orientation of each layer. Tapered thicknesses may be input using the **SECFUNCTION** command. The number of integration points from the **SECDATA** command is not used; rather it is determined for all layers with KEYOPT(3). In the GUI, the ShellTool

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

provides a convenient way to define section data for this element (see Shell Analysis and Cross Sections in the *Structural Analysis Guide*). Real constants are not used for this element.

Generally, the quadratic variation in temperature through each layer (KEYOPT(3) = 0) is used for transient analysis or for strongly temperature dependent materials, and the linear variation in temperature through each layer (KEYOPT(3) = 1) is used for steady state analysis with materials that are either not temperature dependent or weakly temperature dependent. Layers may be used to model the physical changes of properties through the thickness or the effect of a through-thickness transient in greater detail.

KEYOPT(4) duplicates the number of layers input on the **SECDATA** commands. If KEYOPT(4) is 0 or blank, the program will query each element during definition in PREP7 as to which section information is being used, and then reassign the element to a different type. More element types are created as needed. The result can be seen using **ETLIST** and **ELIST** after all elements are defined. To ensure that the program can do this redefinition, the user is required to define the section information before the element is defined.

If KEYOPT(6) (also referred to as the paint option) is used, TBOT is replaced with TEMP, allowing the element to be directly attached to an underlying solid to avoid the use of constraint equations. When this option is used, surface loads cannot be applied to face 1.

As this is a thermal shell element, the direction of the element z-axis and the presence of the **SECOFFSET** command have no effect on the solution. However, to get correct plots when using the **/ESHAPE** command:

- The element z-axis should be defined with the same care as for a structural shell element.
- If KEYOPT(6) = 1 (the paint option) is set, **SECOFFSET**,BOT should be input.

Element loads are described in Nodal Loading (p. 49). Convection or heat flux (but not both) and radiation (using the RDSF surface load label) may be input as surface loads at the element faces as shown by the circled numbers on Figure 131.1: SHELL131 Geometry (p. 467). Because shell edge convection and flux loads are input on a per-unit-length basis, per-unit-area quantities must be multiplied by the total shell thickness. Radiation is not available on the edges. You can also generate film coefficients and bulk temperatures using the surface effect element SURF152. SURF152 can also be used with FLUID116.

Heat generation rates may be input as element body loads on a per layer basis. One heat generation value is applied to the entire layer. If the first layer heat generation rate HG(1) is input, and all others are unspecified, they default to HG(1). Nodal values are averaged over the entire element.

A summary of the element input is given in "SHELL131 Input Summary" (p. 468). A general description of element input is given in Element Input (p. 43).

SHELL131 Input Summary

Nodes

I, J, K, L

Degrees of Freedom

Quadratic:

If KEYOPT(3) = 0
If KEYOPT(4) = 0 or 1: TBOT, TE2, TTOP
If KEYOPT(4) = 2: TBOT, TE2, TE3, TE4, TTOP
If KEYOPT(4) = 3: TBOT, TE2, TE3, TE4, TE5, TE6, TTOP
Etc.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If KEYOPT(4) = 15: TBOT, TE2, TE3, TE4, TE5, TE6, TE7, TE8, TE9, TE10, TE11, TE12, TE13, TE14, TE15, TE16, TE17, TE18, TE19, TE20, TE21, TE22, TE23, TE24, TE25, TE26, TE27, TE28, TE29, TE30, TTOP

Linear:

If KEYOPT(3) = 1
If KEYOPT(4) = 0 or 1: TBOT, TTOP
If KEYOPT(4) = 2: TBOT, TE2, TTOP
If KEYOPT(4) = 3: TBOT, TE2, TE3, TTOP
Etc.
If KEYOPT(4) = 31: TBOT, TE2, TE3, TE4, TE5, TE6, TE7, TE8, TE9, TE10, TE11, TE12, TE13, TE14, TE15, TE16, TE17, TE18, TE19, TE20, TE21, TE22, TE23, TE24, TE25, TE26, TE27, TE28, TE29, TE30, TE31, TTOP

Constant:

If KEYOPT(3) = 2: TEMP (one layer only)

Real Constants

None

Material Properties

MP command: KXX, KYY, KZZ, DENS, C, ENTH

Surface Loads

Convections --

Face 1 (I-J-K-L) (bottom, -z side)
Face 2 (I-J-K-L) (top, +z side)
Face 3 (J-I), Face 4 (K-J), Face 5 (L-K), Face 6 (I-L)

Heat Fluxes --

Face 1 (I-J-K-L) (bottom, -z side)
Face 2 (I-J-K-L) (top, +z side)
Face 3 (J-I), Face 4 (K-J), Face 5 (L-K), Face 6 (I-L)

Radiation --

Face 1 (I-J-K-L) (bottom, -z side)
Face 2 (I-J-K-L) (top, +z side)

Body Loads

Heat Generations --

HG(1), HG(2), HG(3), ..., HG(KEYOPT(4))

Special Features

Birth and death

KEYOPT(2)

Film coefficient evaluation (if any):

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0 --
Evaluate at an average film temperature, (TS+TB)/2

1 --
Evaluate at element surface temperature, TS

2 --
Evaluate at fluid bulk temperature, TB

3 --
Evaluate at differential temperature, |TS-TB|

KEYOPT(3)
Temperature variation through layer:

0 --
Quadratic temperature variation through-layer (maximum number of layers = 15)

1 --
Linear temperature variation through-layer (maximum number of layers = 31)

2 --
No temperature variation through-layer (number of layers = 1)

KEYOPT(4)
Number of layers (input a value to match **SECDATA** commands, or leave blank to default). Maximum number of layers allowed depends on KEYOPT(3) setting (see above).

KEYOPT(6)
Application:

0 --
Thermal shell application

1 --
Paint application

SHELL131 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output shown in Table 131.1: SHELL131 Element Output Definitions (p. 471)

Output temperatures may be read by structural shell elements using the **LDREAD**,TEMP command.

If the structural shell element uses *only one* temperature through the thickness, such as SHELL41, only TEMP can be used.

If the structural shell element uses *two* temperatures through the thickness, such as for SHELL181 (with only one layer) or SHELL281 (with only one layer), only TBOT and TTOP are used and any internal temperatures such as TE2 are ignored.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the structural shell element uses *more than two* temperatures through the thickness, such as for SHELL181 (with multiple layers), all temperatures are transferred over. In this case, the corner nodes of each SHELL131 element must have identical temperature degrees of freedom.

The number of temperature points at a node generated in the thermal shell must match the number of temperature points at a node needed by the structural shell. For example, a two-layer SHELL181 element using the same material and thickness for both layers can get its temperatures from a SHELL131 element using either two layers with KEYOPT(3) = 1 (linear variation) or one layer with KEYOPT(3) = 0 (quadratic variation). Temperatures passed from this element to the stress analysis via **LDREAD**,TEMP can be viewed using **BFELIST**, as opposed to the usual **BFLIST**.

Heat flowing out of the element is considered to be positive. Heat flows are labeled HBOT, HE2, . . . HTOP, similar to the temperature labels. Gradient and flux information is provided at the midthickness of each layer. The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

To see the temperature distribution through the thickness for this element as well as all other thermal elements, use **/GRAPHICS**,POWER and **/ESHAPE**,1 followed by **PLNSOL**,TEMP.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 131.1: SHELL131 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Element material number (from **MAT** command)	Y	Y
AREA	Area of element	Y	Y
XC, YC, ZC	Location where results are reported	Y	2
HGEN	Heat generations: HG(1), HG(2), HG(3), . . .	Y	-
TG:X, Y, Z	Thermal gradient components at midlayer	Y	Y
TF:X, Y, Z	Thermal flux components at midlayer	Y	Y
FACE	Face label	1	1
AREA	Face area (same as element area)	1	1
NODES	Face nodes (same as element nodes)	1	1
HFILM	Face film coefficient	1	1
TAVG	Average face temperature	1	1
TBULK	Fluid bulk temperature	1	-
HEAT RATE	Heat flow rate across face by convection	1	1
HFAVG	Average film coefficient of the face	-	1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
TBAVG	Average face bulk temperature	-	1
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	1
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	1	-
HEAT FLUX	Heat flux at each node of the face	1	-

1. If a surface load is input.

2. Available only at the centroid as a ***GET** item.

Table 131.2: SHELL131 Item and Sequence Numbers (p. 472) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) of this manual for more information. The following notation is used in Table 131.2: SHELL131 Item and Sequence Numbers (p. 472):

Name
output quantity as defined in Table 131.1: SHELL131 Element Output Definitions (p. 471)

Item
predetermined Item label for **ETABLE** command

Table 131.2: SHELL131 Item and Sequence Numbers

Output Quantity Name	Item	Face 1	Face 2	I	J	K	L
		Bottom	Top	Corners			
AREA	NMISC	1	7	--	--	--	--
HFAVG	NMISC	2	8	--	--	--	--
TAVG	NMISC	3	9	--	--	--	--
TBAVG	NMISC	4	10	--	--	--	--
HEAT RATE	NMISC	5	11	--	--	--	--
HFLXAVG	NMISC	6	12	--	--	--	--
THICKNESS	NMISC	--	--	37	38	39	40

SHELL131 Assumptions and Restrictions

- Zero area elements are not allowed. This occurs most frequently when the element is not numbered properly.
- Zero thickness layers are not allowed.
- A triangular element may be formed by defining duplicate K and L node numbers as described in Degenerated Shape Elements (p. 39).
- The cut boundary interpolation command (**CBDOF**) does not work with this element.
- When using thermal contact, the TEMP degree of freedom must be present (KEYOPT(3) = 2 or KEYOPT(6) = 1).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- There should not be a large variation in the ratio of through-thickness conductivity (KZZ) to layer thickness for all layers within the element. If the highest and lowest values for this ratio differ by a large factor (for example, 1e5), then the results for the element may be unreliable.

- No check is made to ensure either that the number of layers between adjacent elements match or that the effective location of a degree of freedom (for example, TE7 from a 10 layer element) between elements sharing the same node is the same to a tolerance. If this is a concern, study the area using the **/ESHAPE** command. For cases where the layering intentionally changes, such as at a joint or at the runout of a tapered layer, use constraint equations (**CE** family of commands) with or without double nodes to connect the two sides.

- The program removes all imposed degrees of freedom and nodal loads (i.e., internally issues **DDELE**,all,all and **FDELE**,all,all commands) when elements that use TTOP, TBOT, etc. as degrees of freedom:

  – are defined or redefined using the **ET** or **KEYOPT** commands.

  – are changed (or deleted) using the **ET**, **ETCHG**, or **ETDELE** commands to an element type that does not use these degrees of freedom.

  If your model contained SHELL131 elements with **D** and **F** loads, and you deleted these elements via **ETDELE**, the **D** and **F** loads will automatically be deleted and reapplied to the new DOF list. You do, however, need to check other loads and verify if they need to be deleted and reapplied.

SHELL131 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The birth and death special feature is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

SHELL132

8-Node Thermal Shell

MP ME <> PR PRN <> <> <> PP EME MFS
Product Restrictions

SHELL132 Element Description

SHELL132 is a 3-D layered shell element having in-plane and through-thickness thermal conduction capability. The element has eight nodes with up to 32 temperature degrees of freedom at each node. The conducting shell element is applicable to a 3-D, steady-state or transient thermal analysis. SHELL132 generates temperatures that can be passed to structural shell elements in order to model thermal bending. See SHELL132 in the *Mechanical APDL Theory Reference* for more details about this element.

If the model containing the conducting shell element is to be analyzed *structurally*, use an equivalent structural shell element instead, such as SHELL281.

Figure 132.1: SHELL132 Geometry

x_o = element x-axis if ESYS is not supplied.

x = element x-axis if ESYS is supplied.

SHELL132 Input Data

The geometry, node locations, and coordinates systems for this element are shown in Figure 132.1: SHELL132 Geometry (p. 475). The element is defined by four/eight nodes, one thickness per layer, a material angle for each layer, and the material properties. If the material is uniform and the analysis has no transient effects, only one layer is needed with a linear temperature variation through the thickness.

The cross-sectional properties are input using the **SECTYPE**,,SHELL and **SECDATA** commands. These properties are the thickness, material number, and orientation of each layer. Tapered thicknesses may

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

be input using the **SECFUNCTION** command. The number of integration points from the **SECDATA** command is not used; rather it is determined for all layers with KEYOPT(3). In the GUI, the ShellTool provides a convenient way to define section data for this element (see Shell Analysis and Cross Sections in the *Structural Analysis Guide*). Real constants are not used for this element.

Generally, the quadratic variation in temperature through each layer (KEYOPT(3) = 0) is used for transient analysis or for strongly temperature dependent materials, and the linear variation in temperature through each layer (KEYOPT(3) = 1) is used for steady state analysis with materials that are either not temperature dependent or weakly temperature dependent. Layers may be used to model the physical changes of properties through the thickness or the effect of a thru-thickness transient in greater detail.

KEYOPT(4) duplicates the number of layers input on the **SECDATA** commands. If KEYOPT(4) is 0 or blank, the program will query each element during definition in PREP7 as to which section information is being used, and then reassign the element to a different type. More element types are created as needed. The result can be seen using **ETLIST** and **ELIST** after all elements are defined. To ensure that the program can do this redefinition, the user is required to define the section information before the element is defined.

If KEYOPT(6) (also referred to as the paint option) is used, TBOT is replaced with TEMP, allowing the element to be directly attached to an underlying solid to avoid the use of constraint equations. When this option is used, surface loads cannot be applied to face 1.

As this is a thermal shell element, the direction of the element z-axis and the presence of the **SECOFFSET** command have no effect on the solution. However, to get correct plots when using the **/ESHAPE** command:

- The element z-axis should be defined with the same care as for a structural shell element.
- If KEYOPT(6) = 1 (the paint option) is set, **SECOFFSET**,BOT should be input.

Element loads are described in Nodal Loading (p. 49). Convection or heat flux (but not both) and radiation (using the RDSF surface load label) may be input as surface loads at the element faces as shown by the circled numbers on Figure 132.1: SHELL132 Geometry (p. 475). Because shell edge convection and flux loads are input on a per-unit-length basis, per-unit-area quantities must be multiplied by the total shell thickness. Radiation is not available on the edges. You can also generate film coefficients and bulk temperatures using the surface effect element SURF152. SURF152 can also be used with FLUID116.

Heat generation rates may be input as element body loads on a per layer basis. One heat generation value is applied to the entire layer. If the first layer heat generation rate HG(1) is input, and all others are unspecified, they default to HG(1). Nodal values are averaged over the entire element.

A summary of the element input is given in "SHELL132 Input Summary" (p. 476). A general description of element input is given in Element Input (p. 43).

SHELL132 Input Summary

Nodes

I, J, K, L, M, N, O, P

Degrees of Freedom

Quadratic:

If KEYOPT(3) = 0
If KEYOPT(4) = 0 or 1: TBOT, TE2, TTOP
If KEYOPT(4) = 2: TBOT, TE2, TE3, TE4, TTOP

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If KEYOPT(4) = 3: TBOT, TE2, TE3, TE4, TE5, TE6, TTOP
Etc.
If KEYOPT(4) = 15: TBOT, TE2, TE3, TE4, TE5, TE6, TE7, TE8, TE9, TE10, TE11, TE12, TE13, TE14, TE15, TE16, TE17, TE18, TE19, TE20, TE21, TE22, TE23, TE24, TE25, TE26, TE27, TE28, TE29, TE30, TTOP

Linear:

If KEYOPT(3) = 1
If KEYOPT(4) = 0 or 1: TBOT, TTOP
If KEYOPT(4) = 2: TBOT, TE2, TTOP
If KEYOPT(4) = 3: TBOT, TE2, TE3, TTOP
Etc.
If KEYOPT(4) = 31: TBOT, TE2, TE3, TE4, TE5, TE6, TE7, TE8, TE9, TE10, TE11, TE12, TE13, TE14, TE15, TE16, TE17, TE18, TE19, TE20, TE21, TE22, TE23, TE24, TE25, TE26, TE27, TE28, TE29, TE30, TE31, TTOP

Constant:

If KEYOPT(3) = 2: TEMP (one layer only)

Real Constants

None

Material Properties

MP command: KXX, KYY, KZZ, DENS, C, ENTH

Surface Loads

Convections --

Face 1 (I-J-K-L) (bottom, -z side)
Face 2 (I-J-K-L) (top, +z side)
Face 3 (J-I), Face 4 (K-J), Face 5 (L-K), Face 6 (I-L)

Heat Fluxes --

Face 1 (I-J-K-L) (bottom, -z side)
Face 2 (I-J-K-L) (top, +z side)
Face 3 (J-I), Face 4 (K-J), Face 5 (L-K), Face 6 (I-L)

Radiation --

Face 1 (I-J-K-L) (bottom, -z side)
Face 2 (I-J-K-L) (top, +z side)

Body Loads

Heat Generations --

HG(1), HG(2), HG(3), ..., HG(KEYOPT(4))

Special Features

Birth and death

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(2)
Film coefficient evaluation (if any):

0 --
Evaluate at an average film temperature, (TS+TB)/2

1 --
Evaluate at element surface temperature, TS

2 --
Evaluate at fluid bulk temperature, TB

3 --
Evaluate at differential temperature, |TS-TB|

KEYOPT(3)
Temperature variation through layer:

0 --
Quadratic temperature variation through layer (maximum number of layers = 15)

1 --
Linear temperature variation through layer (maximum number of layers = 31)

2 --
No temperature variation through layer (number of layers = 1)

KEYOPT(4)
Number of layers (input a value to match **SECDATA** commands, or leave blank to default). Maximum number of layers allowed depends on KEYOPT(3) setting (see above).

KEYOPT(6)
Application:

0 --
Thermal shell application

1 --
Paint application

SHELL132 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output as shown in Table 132.1: SHELL132 Element Output Definitions (p. 479).

Output nodal temperatures may be read by structural shell elements (such as SHELL281) using the **LDREAD**,TEMP capability. If the structural shell element uses two temperatures through the thickness, only TBOT and TTOP are used, and any internal temperatures such as TE2 are ignored. If the structural shell element uses more than two temperatures through the thickness, all temperatures are transferred over. In this case, the corner nodes of each SHELL132 element must have identical temperature degrees of freedom. Also, the number of temperature points at a node generated in the thermal shell must match the number of temperature points at a node needed by the structural shell. For example, a two

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

layer shell element using the same material and thickness for both layers can get its temperatures from a SHELL132 element using either two layers with KEYOPT(3) = 1 (linear variation) or one layer with KEYOPT(3) = 0 (quadratic variation). Temperatures passed from this element to the stress analysis via **LDREAD**,TEMP can be viewed using **BFELIST**, as opposed to the usual **BFLIST**.

Heat flowing out of the element is considered to be positive. Heat flows are labeled HBOT, HE2, . . . HTOP, similar to the temperature labels. Gradient and flux information is provided at the midthickness of each layer. The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

To see the temperature distribution thru the thickness for this element as well as all other thermal elements, use **/GRAPHICS**,POWER and **/ESHAPE**,1 followed by **PLNSOL**L,TEMP.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 132.1: SHELL132 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Element material number (from **MAT** command)	Y	Y
AREA	Area of element	Y	Y
XC, YC, ZC	Location where results are reported	Y	2
HGEN	Heat generations: HG(1), HG(2), HG(3), . . .	Y	-
TG:X, Y, Z	Thermal gradient components at midlayer	Y	Y
TF:X, Y, Z	Thermal flux components at midlayer	Y	Y
FACE	Face label	1	1
AREA	Face area (same as element area)	1	1
NODES	Face nodes (same as element nodes)	1	1
HFILM	Face film coefficient	1	1
TAVG	Average face temperature	1	1
TBULK	Fluid bulk temperature	1	-
HEAT RATE	Heat flow rate across face by convection	1	1
HFAVG	Average film coefficient of the face	-	1
TBAVG	Average face bulk temperature	-	1
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	1
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	1	-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
HEAT FLUX	Heat flux at each node of the face	1	-

1. If a surface load is input.

2. Available only at the centroid as a *GET item.

Table 132.2: SHELL132 Item and Sequence Numbers (p. 480) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) of this manual for more information. The following notation is used in Table 132.2: SHELL132 Item and Sequence Numbers (p. 480):

Name
output quantity as defined in the Table 132.1: SHELL132 Element Output Definitions (p. 479)

Item
predetermined Item label for **ETABLE** command

Table 132.2: SHELL132 Item and Sequence Numbers

Output Quantity Name	Item	Face 1	Face 2	I	J	K	L
		Bottom	Top	Corners			
AREA	NMISC	1	7	--	--	--	--
HFAVG	NMISC	2	8	--	--	--	--
TAVG	NMISC	3	9	--	--	--	--
TBAVG	NMISC	4	10	--	--	--	--
HEAT RATE	NMISC	5	11	--	--	--	--
HFLXAVG	NMISC	6	12	--	--	--	--
THICKNESS	NMISC	--	--	37	38	39	40

SHELL132 Assumptions and Restrictions

- Zero area elements are not allowed. This occurs most frequently when the element is not numbered properly.
- Zero thickness layers are not allowed.
- A triangular element may be formed by defining duplicate K, L, and O node numbers as described in Degenerated Shape Elements (p. 39).
- Midside nodes may not be dropped.
- The cut boundary interpolation command (**CBDOF**) does not work with this element.
- When using thermal contact, the TEMP degree of freedom must be present (KEYOPT(3) = 2 or KEYOPT(6) = 1).
- There should not be a large variation in the ratio of through-thickness conductivity (KZZ) to layer thickness for all layers within the element. If the highest and lowest values for this ratio differ by a large factor (for example, 1e5), then the results for the element may be unreliable.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- No check is made to ensure either that the number of layers between adjacent elements match or that the effective location of a degree of freedom (for example, TE7 from a 10 layer element) between elements sharing the same node is the same to a tolerance. If this is a concern, study the area using the **/ESHAPE** command. For cases where the layering intentionally changes, such as at a joint or at the runout of a tapered layer, use constraint equations (**CE** family of commands) with or without double nodes to connect the two sides.
- This element may not be used with **THOPT**,QUASI if convection or radiation surfaces are present.
- This element may not be used with the **/EFACET** command for PowerGraphics displays.
- The program removes all imposed degrees of freedom and nodal loads (i.e., internally issues **DDELE**,all,all and **FDELE**,all,all commands) when elements that use TTOP, TBOT, etc. as degrees of freedom:
  - are defined or redefined using the **ET** or **KEYOPT** commands.
  - are changed (or deleted) using the **ET**, **ETCHG**, or **ETDELE** commands to an element type that does not use these degrees of freedom.

    If your model contained SHELL132 elements with **D** and **F** loads, and you deleted these elements via **ETDELE**, the **D** and **F** loads will automatically be deleted and reapplied to the new DOF list. You do, however, need to check other loads and verify if they need to be deleted and reapplied.

SHELL132 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The birth and death special feature is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

FLUID136

3-D Squeeze Film Fluid Element

MP ME <> <> <> <> <> <> PP EME MFS
Product Restrictions

FLUID136 Element Description

FLUID136 models viscous fluid flow behavior in small gaps between fixed surfaces and structures moving perpendicular to the fixed surfaces. The element behavior is based on the Reynolds squeeze film theory and the theory of rarefied gases. As such, it is limited to structures with lateral dimensions much greater than the gap size. In addition, the pressure change must be small relative to the ambient pressure, and any viscous heating is neglected. FLUID136 is particularly applicable to modeling squeeze-film effects in microstructures. However, it can also model thin-film fluid behavior in macrostructures.

As a fluid-only element (PRES dof), the element can be used to determine the stiffening and damping effects that a fluid exerts on a moving structure by applying a known normal velocity. The velocity normal to the element surface is specified as a body force. If the velocity of the moving surface is not known, FLUID136 can determine the fluid response from the eigenmodes of the structure using the Modal Projection Method.

FLUID136 is applicable to static, harmonic, and transient analyses. A static analysis is used to determine the damping effects for low operating frequencies where fluid stiffening effects are negligible. A harmonic analysis is used to determine the fluid stiffening and damping effects for high operating frequencies where fluid stiffening effects are not negligible. A transient analysis is used to determine the fluid stiffening and damping effects for non-harmonic loadings. The Modal Projection Method can also be used to extract frequency-dependent damping ratios for use with the **MDAMP** and **DMPRAT** commands; and Alpha and Beta damping parameters for use with the **ALPHAD** and **BETAD** commands.

As a fluid-structure element (PRES, UX, UY, UZ), the element can be combined with solid structural elements in a coupled-field solution where pressure effects are computed from the structure's motion. In this mode, the element is applicable to a static or transient analysis. Compressibility options are available when considering large displacements and/or large pressure changes. Contact options are also available when the structural degrees of freedom are active in order to model opening and closing contact conditions.

FLUID136 can be used to model three different flow regimes: continuum theory, high Knudsen number, and high Knudsen number with accommodation factors.

See FLUID136 in the *Mechanical APDL Theory Reference* for more details about this element.

FLUID136 Input Data

The element is defined by four corner nodes with an option to include mid-side nodes (KEYOPT(2) = 1). The element should be oriented such that the element normal is pointing toward the fluid domain. If solid elements are used for the structural domain, the fluid element normal vector is automatically computed. If necessary, the fluid element normal vector can be flipped using **ENSYM**.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 136.1: FLUID136 Geometry

KEYOPT (1) specifies the flow regime. The Knudsen number can be calculated from the mean free fluid path at a reference pressure, the operating or absolute pressure, and the gap.

For a PRES degree of freedom (KEYOPT(3) = 0) and a linearized Reynolds equation (KEYOPT(4) = 0 or 2),

$$K_n = (MFP*PREF) / (PAMB*GAP)$$

For PRES, UX, UY, UZ degrees of freedom (KEYOPT(3) = 1 or 2) and a nonlinear Reynolds equation (KEYOPT(4) = 1),

$$K_n = (MFP*PREF) / (P_{abs}*GAP) \text{ if } P_{abs} > minpabs$$

$$K_n = (MFP*PREF) / (minpabs*GAP) \text{ if } P_{abs} < minpabs$$

where:

P_{abs} = PAMB + PRES
minpabs = minimum absolute pressure determined by real constant MINPABSF

For continuum theory to be valid (KEYOPT(1) = 0), the Knudsen number should be less than 0.01. If the Knudsen number is greater than 0.01 (KEYOPT(1) = 1 or 2), the dynamic viscosity is adjusted to account for the slip flow boundary. See Flow Regime Considerations in the *Fluids Analysis Guide* for a complete discussion of flow regimes and calculation of the Knudsen number.

The type of reflection of the gas molecules at the wall interface is specified using accommodation factors. Squeeze film models assume diffuse reflection of the gas molecules at the wall interface (accommodation factor = 1). This assumption is valid for most metals, but is less accurate for micromachined surfaces, particularly those fabricated from silicon. Materials, such as silicon, cause specular reflection. Typical accommodation factors for silicon are between 0.80 and 0.90.

KEYOPT (3) sets the element degrees of freedom. Setting KEYOPT (3) to 1 or 2 activates the displacement degrees of freedom. When displacement DOFs are active both fluidic and mechanical contact pressures can be generated. FLUID136 can only be used for static and transient analyses when the displacement DOFs are activated.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If KEYOPT(5) = 2, the element is ignored from a fluid pressure standpoint when the fluid gap goes below a specified minimum fluid gap (fluid_mingap). If KEYOPT(6) = 1 or 2, mechanical contact pressure is applied to a structure if the fluid height goes below a specified minimum mechanical gap (mech_mingap).

For the fluid-only option (PRES dof), the fluid velocity normal to the surface may be specified using nodal or element loading with the FLUE body load label on the **BF** or **BFE** commands. If FLUID136 is used in conjunction with the Modal Projection Method, the fluid velocities are obtained from the modal displacements and applied using the **DMPEXT** command.

FLUID136 Input Summary

Nodes

I, J, K, L (KEYOPT(2) = 0)

I, J, K, L, M, N, O, P (KEYOPT(2) = 1)

Degrees of Freedom

See KEYOPT(3)

Real Constants

See Table 136.1: FLUID136 Real Constants (p. 487).

Material Properties

MP command: VISC (dynamic viscosity)

Surface Loads

None

Body Loads

FLUE (velocity) (For KEYOPT(3) = 0 only)

Special Features

None

KEYOPT(1)

Continuous flow options

0 --

Continuum theory

1 --

High Knudsen numbers (greater than 0.01)

2 --

High Knudsen numbers and accommodation factors

KEYOPT(2)

Element geometry

0 --

Four node element

1 --

Eight node element (not available if KEYOPT(3) = 1 or 2)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(3)

Degrees of Freedom

0 --

PRES (Valid for static, harmonic, and transient analyses.)

1 --

PRES, UX, UY, UZ - explicit treatment of cross-coupling terms. Produces a symmetric matrix. Valid for static and transient analyses only. Convergence issues may be experienced if the fluid gap approaches zero.

2 --

PRES, UX, UY, UZ - implicit treatment of cross-coupling terms. Produces an unsymmetric matrix. Valid for static and transient analyses only.

KEYOPT(4)

Compressibility. If PRES is the only degree of freedom (KEYOPT(3) = 0), the compressible linearized Reynold equation is used (KEYOPT(4) = 0). The following are valid when degrees of freedom are PRES, UX, UY, and UZ (KEYOPT(3) = 1 or 2).

0 --

Compressible linearized Reynolds equation. (large displacement and small pressure changes)

1 --

Compressible nonlinear Reynolds equation. (large displacement and large pressure changes)

2 --

Incompressible linearized Reynolds equation. (large displacement and small pressure changes)

For more information on the linearized Reynolds equation, refer to Flow Between Flat Surfaces in the *Mechanical APDL Theory Reference*.

KEYOPT(5)

If the element gap goes below fluid_mingap:

0 --

Trap it as an error.

1 --

Reset it to fluid_mingap.

2 --

Ignore this element from a fluid pressure standpoint. This element is considered dead from a fluids standpoint. However, for postprocessing, a fluid pressure can be specified. See real constants PENP and SPRES.

For KEYOPT(5) = 1 or 2, mechanical contact may be included by KEYOPT(6) or TARGE170 and CONTA174 elements.

If the element gap is above fluid_mingap, fluid pressure is applied on the structure.

KEYOPT(6)

If the element gap goes below mech_mingap:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0 --
Do not apply mechanical contact pressure on the structure. This element is considered mechanically dead.

1 --
Apply mechanical contact pressure on the structure using the penalty method. Specify a stiffness parameter (real constant STIFFP). Damping is input by real constant DAMPP and it defaults to zero.

2 --
Apply mechanical contact pressure on the structure using the augmented Lagrangian method. Specify an initial stiffness (real constant STIFFP) and a penetration tolerance (real constant MPTF). Damping is input by real constant DAMPP and it defaults to zero.

The fluid environment is defined by the following set of real constants.

Table 136.1: FLUID136 Real Constants

No.	Name	Description
1	GAP	Element gap separation
2	blank	—
3	blank	—
4	PAMB	Ambient (i.e., surrounding) pressure
5	ACF1	Accommodation factor for top moving surface.
6	ACF2	Accommodation factor for bottom fixed surface.
7	PREF	Reference pressure for the mean free fluid path
8	MFP	Mean free fluid path at reference pressure PREF
9	GAPX	Gap vector global Cartesian component X
10	GAPY	Gap vector global Cartesian component Y
11	GAPZ	Gap vector global Cartesian component Z
12	MMGF	mech_mingapf (minimum mechanical gap as a fraction of GAP)
13	FMGF	fluid_mingapf (minimum fluid gap as a fraction of GAP)
14	PENP	Penalty parameter for fluid dead element (KEYOPT(5) = 2)
15	SPRES	Specified pressure for fluid dead element (KEYOPT(5) = 2)
16	STIFFP	Stiffness parameter for mechanical contact (KEYOPT(6) = 1 or 2)
17	DAMPP	Damping parameter for mechanical contact (KEYOPT(6) = 1 or 2)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

No.	Name	Description
18	MPTF	mech_pen_tolf (KEYOPT(6) = 2) (mechanical penetration tolerance as a fraction of mech_mingap)
19	MINPABSF	minpabsf (minimum absolute pressure as a fraction of P_{amb})

For continuum theory (KEYOPT(1) = 1), GAP and PAMB must be specified.

For high Knudsen numbers (KEYOPT(1) = 1), GAP, PAMB, PREF, and MFP must be specified. PREF and MFP are used to adjust the dynamic viscosity. ACF1 and ACF2 are assumed to be 1.

For high Knudsen numbers with accommodation factors (KEYOPT(1) = 2), GAP, PAMB, PREF, MFP, ACF1, and ACF2 must be specified. Different accommodation factors may be specified for each surface.

For small deflections, GAP is assumed to be constant. For the fluid-only option (PRES dof) and large deflections, GAP can be updated using **SETFGAP**.

Real constants GAPX, GAPY, and GAPZ are the unit vector components of the normal gap vector g in the global Cartesian system (see figure below).

Figure 136.2: Moving Structure with Thin Film of FLUID136 Elements

Real constants FMGF and MMGF determine the minimum fluid gap (fluid_mingap) and minimum mechanical gap (mech_mingap) as shown below:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 136.3: Minimum Fluid and Minimum Mechanical Gaps

$$\text{fluid_mingap} = \begin{cases} \text{GAP*FMGF} & \text{if FMGF} > 0 \\ |\text{FMGF}| & \text{if FMGF} < 0 \end{cases}$$

$$\text{mech_mingap} = \begin{cases} \text{GAP*MMGF} & \text{if MMGF} > 0 \\ |\text{MMGF}| & \text{if MMGF} < 0 \end{cases}$$

Real constant MPTF determines the mechanical penetration tolerance as shown below:

Figure 136.4: Mechanical Penetration

$$\text{mech_pen_tol} = \begin{cases} \text{mech_mingap*MPTF} & \text{if MPTF} > 0 \\ |\text{MPTF}| & \text{if MPTF} < 0 \end{cases}$$

Figure 136.5: Fluid Penetration

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Real constant MINPABSFA determines the minimum absolute pressure as shown below. The minimum absolute pressure is used in the definition of Knudson number.

$$\text{minpabs} = \begin{cases} P_{amb} * \text{MINPABSF} & \text{if MINPABSF} > 0 \\ |\text{MINPABSF}| & \text{if MINPABSF} < 0 \end{cases}$$

Stiffness is input by real constant STIFFP and it is typically large. Damping is input by real constant DAMPP and it is typically zero.

FLUID136 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in Table 136.2: FLUID136 Element Output Definitions (p. 490)

A general description of solution output is given in Table 136.2: FLUID136 Element Output Definitions (p. 490). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 136.2: FLUID136 Element Output Definitions

Name	Definition	O	R
PRES	Pressure change with regard to ambient temperature		Y
PG (X, Y, Z)	Mid-surface fluid velocity	Y	Y
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
AREA:	Area	Y	Y
FLUE	Velocity (normal to surface)	Y	Y
SNORMAL(YX, Y, Z)Y	Components of unit surface normal n	-	-
VELC(X, Y, Z)	Components of mechanical velocity at centroid	-	-
DISPC(X, Y, Z)	Components of displacement at centroid	-	-
PRESC	Fluid pressure at centroid	-	-
GAPDIR(X, Y, Z)	Components of gap vector g	-	-
FLUIDDEAD	Fluid alive or dead (1 = alive; 0 = dead)	-	-
FLUIDPEN	Fluid penetration	-	-
MECHDEAD	Mechanical alive or dead (1 = alive; 0 = dead)	-	-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
MECHPEN	Mechanical penetration at centroid	-	-
STIFF	Element stiffness	-	-
CONTPRES	Contact pressure at centroid	-	-
KN	Knudsen number	-	-

Contact pressure is computed as an element centroidal quantity:

CONTPRES= STIFFP*mech_penetration + DAMPP*mech_velocity

Table 136.3: FLUID136 Item and Sequence Numbers (p. 491) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 136.3: FLUID136 Item and Sequence Numbers (p. 491):

Name
output quantity as defined in the Table 136.2: FLUID136 Element Output Definitions (p. 490)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 136.3: FLUID136 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
Effective viscosity	NMISC	1
GAP	NMISC	2
KEYOPT(3) = 1 or 2		
AREA	NMISC	3
SNORMALX	NMISC	4
SNORMALY	NMISC	5
SNORMALZ	NMISC	6
VELCX	NMISC	7
VELCX	NMISC	8
VELCX	NMISC	9
DISPCX	NMISC	10
DISPCY	NMISC	11
DISPCZ	NMISC	12
PRESC (Zero if KEY-OPT(5) = 2)	NMISC	13

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
GAPDIRCX	NMISC	14
GAPDIRCY	NMISC	15
GAPDIRCZ	NMISC	16
FLUIDDEAD (0 if KEYOPT(5) = 2) (1 if KEYOPT(5) ≠ 2)	NMISC	17
FLUIDPEN (0 if KEYOPT(5) ≠ 2)	NMISC	18
MECHDEAD (0 if KEYOPT(6) = 0) (1 if KEYOPT(6) ≠ 0)	NMISC	19
MECHPEN (0 if KEYOPT(6) = 0)	NMISC	20
STIFF (0 if KEYOPT(6) = 0)	NMISC	21
CONTPRES (0 if KEYOPT(6) = 0)	NMISC	22
KN	NMISC	23

FLUID136 Assumptions and Restrictions

- Knudsen numbers larger than 880 are not supported.
- The gas flow is assumed to be isothermal.
- The fluid gap is small compared to the lateral width of the underlying structure.
- The element assumes isothermal viscous flow. All the fluid properties are at a constant temperature (**TUNIF**) within a load step, even if you specify material properties with temperature dependencies (using **MP**).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See Squeeze Film Theory in the *Mechanical APDL Theory Reference* for more information on the governing equations.

- This element cannot be used in a distributed solution.

FLUID136 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

FLUID138

3-D Viscous Fluid Link Element

MP ME <> <> <> <> <> <> PP EME MFS
Product Restrictions

FLUID138 Element Description

FLUID138 models the viscous fluid flow behavior through short channels (i.e., holes) in microstructures moving perpendicular to fixed surfaces. FLUID138 can be used in conjunction with FLUID136 elements to determine the stiffening and damping effects that the fluid exerts on the moving perforated microstructure.

FLUID138 assumes isothermal flow at low Reynolds numbers. The channel length must be small relative to the acoustic wave length, and the pressure change must be small relative to the ambient pressure. FLUID138 accounts for gas rarefaction effects and fringe effects due to the short channel length.

As with FLUID136, FLUID138 is applicable to static, harmonic, and transient analyses. FLUID138 can be used to model two different flow regimes: continuum theory and high Knudsen number.

In contrast to FLUID116, this element is more accurate for channels of rectangular cross section, allows channel dimensions to be small compared to the mean free path, allows modeling of evacuated systems, and considers fringe effects at the inlet and outlet. These effects can considerably increase the damping force in the case of short channel length. See FLUID138 - 3-D Viscous Fluid Link Element in the *Mechanical APDL Theory Reference* for more details about this element.

FLUID138 Input Data

The element is defined by two nodes. The I node is located at the center of the cross-section of the hole region on the same plane as the nodes used to model the squeeze film fluid region (FLUID136 elements). The J node is located at the opposite face of the structure through the channel depth.

Figure 138.1: FLUID138 Geometry

KEYOPT(1) specifies the flow regime. The Knudsen number can be calculated from the mean free fluid path at a reference pressure, the operating pressure, and the lateral dimensions.

$$K_n = (MFP*PREF) / (PAMB*DIM)$$

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For rectangular channels, DIM is the smallest lateral dimension. For circular channels, DIM is the radius.

For continuum theory to be valid (KEYOPT(1) = 0), the Knudsen number should be less than 0.01. If the Knudsen number is greater than 0.01 (KEYOPT(1) = 1 or 2), the dynamic viscosity is adjusted to account for the slip flow boundary.

The fluid environment is defined by a set of real constants:

For rectangular channels, DIM1 and DIM2 specify the lateral dimensions of the channel. For circular channels, DIM1 specifies the radius of the channel and DIM2 is not used, PAMB specifies the ambient (i.e., surrounding) pressure, PREF specifies the reference pressure for the mean free fluid path, and MFP specifies the mean free fluid path at reference pressure PREF.

For continuum theory (KEYOPT(1) = 1), DIM1, DIM2 (if rectangular channel), and PAMB must be specified.

For high Knudsen numbers (KEYOPT(1) = 1), DIM1, DIM2 (if rectangular channel), PAMB, PREF and MFP must be specified. PREF and MFP are used to adjust the dynamic viscosity.

FLUID138 does not support any loadings. To preserve the pressure drop through the hole, the PRES degree of freedom for the nodes of the FLUID136 elements at the periphery of the hole must be coupled to the PRES degree of freedom for node I of the FLUID138 element representing the hole, and the pressure degree of freedom for node J must be set to the surrounding ambient pressure.

FLUID138 Input Summary

Nodes
I, J

Degrees of Freedom
PRES

Real Constants
DIM1, DIM2, (blank), PAMB, (blank), (blank),

PREF, MFP

Material Properties
MP command: VISC (dynamic viscosity)

Surface Loads
None

Body Loads
None

Special Features
None

KEYOPT(1)
Continuous flow options

0 --
Continuum theory

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
High Knudsen numbers

KEYOPT(3)
Cross section definition

0 --
Circular cross section

1 --
Rectangular cross section

FLUID138 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in Table 129.1: FLUID129 Element Output Definitions (p. 461)

A general description of solution output is given in Table 136.2: FLUID136 Element Output Definitions (p. 490). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 138.1: FLUID138 Element Output Definitions

Name	Definition	O	R
PRES	Pressure change with regard to ambient pressure		Y
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
MAT	Material number	Y	Y
VOL	Volume	Y	Y
FLUE	Fluences	Y	Y
LENGTH	Channel Length	Y	Y
AREA	Area	Y	Y
PRES (I, J)	P1 at node I, P2 at node J	Y	Y
FLOW	Flow rate	Y	
VELOCITY	Average velocity	Y	

Table 138.2: FLUID138 Item and Sequence Numbers (p. 498) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic*

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Analysis Guide and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 129.2: FLUID129 Item and Sequence Numbers (p. 461):

Name
output quantity as defined in the Table 129.1: FLUID129 Element Output Definitions (p. 461)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 138.2: FLUID138 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
Effective viscosity	NMISC	1
Effective length	NMISC	2
Fluid resistance	NMISC	3
Cross sectional area	NMISC	4

FLUID138 Assumptions and Restrictions

- Knudsen numbers larger than 880 are not supported.
- The gas flow is assumed to be isothermal.
- The pressure change must be small compared to ambient pressure.
- The element assumes isothermal viscous flow. All the fluid properties are at a constant temperature (**TUNIF**) within a load step, even if you specify material properties with temperature dependencies (using **MP**). See Squeeze Film Theory in the *Mechanical APDL Theory Reference* for more information on the governing equations.

FLUID138 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FLUID139

3-D Slide Film Fluid Element

MP ME <> <> <> <> <> <> PP EME MFS
Product Restrictions

FLUID139 Element Description

FLUID139 is a uniaxial element which models the fluid behavior between a sliding surface and a fixed wall. The viscous flow between surfaces is represented by a series connection of mass-damper elements whereby each node corresponds to a local fluid layer. The element has applications for modeling the fluid damping effects in microsystems such as comb drive fingers, large horizontally moving plates in seismic devices, etc. The element can be used in conjunction with other elements to model complete structural-fluid damping interaction, or stand-alone to add damping effects in a lumped sense to a structure. For low frequency applications, Couette flow assumptions is used. At higher frequencies where inertial effects become important, Stokes flow theory is used. First and second order slip flow models can be activated for systems which operate at high Knudsen numbers. The element is applicable to large deflection cases where the surface area exposed to a fixed wall changes with displacement (such as in comb fingers). See FLUID139 in the *Mechanical APDL Theory Reference* for more details about this element.

FLUID139 Input Data

The element is defined by two nodes. The I node is connected to the first "wall" and the J (or I+32) node is attached to the second "wall". Either wall may be constrained from moving, or both walls may move with respect to one another.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 139.1: FLUID139 Geometry

The 2-node option (KEYOPT(2) = 0) is recommended for systems which operate at frequencies below the cut-off frequency.

The 32-node option (KEYOPT(2) = 1) is necessary for Stokes flow models where only a small fluid layer at the wall is accelerated due to fluid inertia. For the 32 node option (KEYOPT(2) = 1), the first node is node I, and the 32nd node is node J. The intermediate node numbers (2-31) must be defined, but their location may be arbitrary. The geometric location of node I and J is not important as their separation distance is computed from the real constant gap separation GAP.

The fluid environment is defined by the real constants.

Gap is the local gap separation (i.e., width of fluid domain). AREA is the surface area. DADU is the change in the overlap area with respect to the surface displacement. That is, DADU is the first derivative of AREA with respect to displacement. If the surface area is constant, then DADU is the width of the overlap surface. PAMB is the ambient (i.e., surrounding) pressure. PREF is the reference pressure for which the mean free path of the fluid is defined. MFP is the mean free path of the fluid at PREF.

FLUID139 can be used to model continuous flow or slip flow boundary conditions. If the Knudsen number is less than 0.01, then continuous flow boundary conditions are valid. If the Knudsen number is greater than 0.01, but not near 1, then first order slip flow boundary conditions are valid. If the Knudsen number is near 1, then extended slip flow boundary conditions are valid. KEYOPT(3) is used to specify fluid blow boundary conditions. KEYOPT(3) = 0 specifies continuous flow. KEYOPT(3) = 1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

specifies first order slip flow boundary conditions. KEYOPT(3) = 2 specifies extended slip flow boundary conditions. See Flow Regime Considerations in the *Fluids Analysis Guide* for a complete discussion of flow regimes and calculation of the Knudsen number.

FLUID139 can be loaded by nodal displacements at the interface nodes using the **D** command or by nodal forces using the **F** command. A combination of FLUID139 and structural elements allows a simultaneous fluid-structure domain simulation.

FLUID139 Input Summary

Nodes

I, J (KEYOPT(2) = 0)

I, J, node 32 (KEYOPT(2) = 1)

Degrees of Freedom

UX, UY, UZ (Depending on KEYOPT(1))

Real Constants

GAP, AREA, DADU, PAMB, (blank), (blank)

PREF, MFP

Material Properties

MP command: DENS (density), VISC (dynamic viscosity)

Surface Loads

None

Body Loads

None

Special Features

None

KEYOPT(1)

Operating Directions

0,1 --
x-direction (UX DOF)

2 --
y-direction (UY DOF)

3 --
z-direction (UZ DOF)

KEYOPT(2)

Flow model

0 --
2-node element (Couette flow)

1 --
32-node element (Stokes flow)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(3)
Continuous flow options

0 --
Continuum theory

1 --
First order slip flow

2 --
Extended slip flow theory

FLUID139 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in Table 129.1: FLUID129 Element Output Definitions (p. 461)

A general description of solution output is given in Table 136.2: FLUID136 Element Output Definitions (p. 490). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 139.1: FLUID139 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
MAT	Material number	Y	Y
VOL	Volume	Y	Y
GAP	Gap separation	Y	Y
AREA	Area	Y	Y
PRES (I, J)	P1 at node I, P2 at node J	Y	Y

Table 138.2: FLUID138 Item and Sequence Numbers (p. 498) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 129.2: FLUID129 Item and Sequence Numbers (p. 461):

Name
output quantity as defined in the Table 129.1: FLUID129 Element Output Definitions (p. 461)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Item

predetermined Item label for **ETABLE** command

E

sequence number for single-valued or constant element data

Table 139.2: FLUID139 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
Effective viscosity	NMISC	1
GAP	NMISC	2
AREA	NMISC	3

FLUID139 Assumptions and Restrictions

The element assumes isothermal viscous flow. All the fluid properties are at a constant temperature (**TUNIF**) within a load step, even if you specify material properties with temperature dependencies (using **MP**). See Slide Film Theory in the *Mechanical APDL Theory Reference* for more information on the governing equations.

This element cannot be used in a distributed solution.

FLUID139 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

ROM144

Reduced Order Electrostatic-Structural

MP <> <> <> <> <> <> <> PP EME <>
Product Restrictions

ROM144 Element Description

ROM144 represents a 2-D or 3-D reduced order model of a coupled electrostatic-structural system. The element fully couples the electromechanical domains and represents a reduced order model suitable for use in finite element analysis as well as electromechanical circuit simulations. The element has ten modal degrees of freedom relating modal forces and modal displacements (EMF), ten voltage degrees of freedom relating electrical current and potential (VOLT) and, optionally, 10 master nodes relating nodal forces to nodal displacements (UX). Only nine of the 10 modal degrees of freedom and five of the 10 voltage degrees of freedom are actually used. The element is suitable for simulating the electromechanical response of micro-electromechanical devices (MEMS) such as clamped beams, micromirror actuators, and RF switches.

The element is derived from a series of uncoupled structural and electrostatic domain simulations using the electrostatic elements (such as PLANE121, SOLID122, SOLID123, and INFIN111) and structural elements which are compatible with electrostatic elements. The ROM144 element represents a complicated flexible structure whose nodes move mainly in one direction either X, Y or Z referred to the global Cartesian axes. For instance, torsional systems with angles less than ten degree or flexible bending of cantilevers or membranes obey those restrictions (pressure sensors, cantilever for AF microscopy, RF filter). Geometrical nonlinearities caused by stress stiffening or initial prestress are considered as well as multiple conductor systems. See ROM144 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 144.1: ROM144 Schematic

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ROM144 Input Data

The element is defined by 20 (KEYOPT(1) = 0) or 30 nodes (KEYOPT(1) = 1). A reduced order model file `filename.rom` and the appropriate polynomial coefficients for the strain energy and capacitance functions stored in `jobname_ijk.pcs` must be available in the working directory. Furthermore, the model database `filename.db` and the reduced solution file (.rdsp) generated by the Use Pass are required to perform an Expansion Pass.

Real constant number 1 (R1) is the element identification number (ID). It is automatically created by the circuit builder (see Using the Circuit Builder in the *Low-Frequency Electromagnetic Analysis Guide*), and is not required input for analysis purposes. The element supports nodal forces F and displacements D applied at ROM master nodes (21 to 30). The UX degree of freedom must be chosen independent from the physical direction of the original master node. Electrode current and voltage can be applied only to the first five active voltage nodes (11–15). Modal displacements may be set by the EMF degree of freedom using the **D** command. Element loads defined in the Generation Pass may be scaled and superimposed by the **RMLVSCALE** command. ROM144 can be attached to other finite elements such as COMBIN14 and COMBIN40 at the master DOF. The "reaction force" for the modal displacement degree of freedom (EMF) is a modal force, labeled CURT, and should be used when defining the solution convergence criteria (**CNVTOL** command). The "reaction force" for the electric potential degree of freedom (VOLT) is current, labeled AMPS. The element is compatible with the electric circuit elements CIRCU124 and CIRCU125 and the electromechanical transducer element TRANS126.

Modal damping ratios may be altered by the **RMMRANGE** command. Save the ROM database before using the changed data in the Use Pass.

A summary of the element input is given in "ROM144 Input Summary" (p. 506).

ROM144 Input Summary

Nodes

20 nodes if KEYOPT(1) = 0:

I, J, K, L, M, N, O, P, Q, Blank, S, T, U, V, W, Blank, Blank, Blank, Blank, Blank

30 nodes if KEYOPT(1) = 1:

I, J, K, L, M, N, O, P, Q, Blank, S, T, U, V, W, Blank, Blank, Blank, Blank, Blank, C, D, E, F, G, H, II, JJ, KK, LL

Degrees of Freedom

EMF, VOLT, UX

Real Constants

R1 - Element identification number

Material Properties

None

Surface Loads

via **RMLVSCALE** command

Body Loads

via **RMLVSCALE** command

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Special Features

Nonlinearity
Prestress

KEYOPT(1)
Select DOF set:

0 --
No ROM master nodes will be used (default).

1 --
ROM master nodes are used.

KEYOPT(2)
Select matrix option:

0 --
Unsymmetric matrix option (default).

1 --
Symmetric matrix option (must be activated in case of **ANTYPE** = MODAL).

ROM144 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution.
- Additional element output as shown in the following table.

Table 144.1: ROM144 Element Output Definitions

Name	Definition
SENG	Strain energy
CAP1	First capacitance defined by **RMCAP**
CAP2	Second capacitance defined by **RMCAP**
CAP3	Third capacitance defined by **RMCAP**
CAP4	Forth capacitance defined by **RMCAP**
CAP5	Fifth capacitance defined by **RMCAP**
CAP6	Sixth capacitance defined by **RMCAP**
CAP7	Seventh capacitance defined by **RMCAP**
CAP8	Eighth capacitance defined by **RMCAP**
CAP9	Ninth capacitance defined by **RMCAP**
CAP10	Tenth capacitance defined by **RMCAP**

Table 144.2: ROM Item and Sequence Numbers (p. 508) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table in the *Element Reference* for more information. The following notation is used in Table 144.2: ROM Item and Sequence Numbers (p. 508):

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name

output quantity as defined in the Table 144.1: ROM144 Element Output Definitions (p. 507)

Item

predetermined Item label for **ETABLE** command

E

sequence number for single-valued or constant element data

Table 144.2: ROM Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
SENG	NMISC	1
CAP1	NMISC	2
CAP2	NMISC	3
CAP3	NMISC	4
CAP4	NMISC	5
CAP5	NMISC	6
CAP6	NMISC	7
CAP7	NMISC	8
CAP8	NMISC	9
CAP9	NMISC	10
CAP10	NMISC	11

ROM144 Assumptions and Restrictions

- Modal forces may not be applied to the ROM element.
- Harmonic and modal analyses are valid only for small-signal analyses after a static prestress calculation.
- Using different ROM elements (i.e., based on different ROM database and polynomial coefficient files) in the same Use Pass is not supported.
- This element cannot be used in a distributed solution.

ROM144 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SURF151

2-D Thermal Surface Effect

MP ME <> PR PRN DS <> <> PP EME MFS
Product Restrictions

SURF151 Element Description

SURF151 may be used for various load and surface effect applications. It may be overlaid onto a face of any 2-D thermal solid element (except axisymmetric harmonic elements PLANE75 and PLANE78). The element is applicable to 2-D thermal analyses. Various loads and surface effects may exist simultaneously. See SURF151 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 151.1: SURF151 Geometry

SURF151 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 151.1: SURF151 Geometry (p. 509). The element is defined by two to five node points and the material properties. An extra node (away from the base element) may be used for convection or radiation effects. Two extra nodes (away from the base element) may be used to more accurately capture convection effects.

The mass, volume, and heat generation calculations use the in-plane element thicknesses at nodes I and J (real constants TKI and TKJ, respectively). Thickness TKI defaults to 0.0, and thickness TKJ defaults to TKI. If KEYOPT(3) = 3, the out-of-plane thickness is input as the real constant TKPS (defaults to 1.0). The mass calculation uses the density (material property DENS).

See Nodal Loading (p. 49) for a description of element loads. Convections or heat fluxes may be input as surface loads on the element.

The convection surface conductivity matrix calculation uses the film coefficient (input on the **SFE** command with *KVAL* = 0 and *CONV* as the label). If the extra node option is used, its temperature becomes the bulk temperature. If the extra node is not used, the *CONV* value input with *KVAL* = 2 becomes the bulk temperature. The convection surface heat flow vector calculation uses the bulk temperature. On a given face, either a heat flux or a convection may be specified, but not both simultaneously.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For the extra node option (KEYOPT(5) = 1), film effectiveness and free stream temperatures may also be input for convection surface loads (input on the **SFE** command with the *CONV* label and *KVAL* = 3 and 4, respectively). If film effectiveness is input, bulk temperature is ignored.

Setting KEYOPT(7) = 1 multiplies the evaluated film coefficient by the empirical term $ITS-TBI^n$, where TS is the element surface temperature, TB is the fluid bulk temperature, and n is an empirical coefficient (real constant ENN).

If KEYOPT(5) = 1 and flow information is available from FLUID116 with KEYOPT(2) = 1, the bulk temperature may be adjusted to the adiabatic wall temperature using KEYOPT(6) = 1, real constants OMEG (rotational speed) and NRF (recovery factor), and the logic described in the *Mechanical APDL Theory Reference*. For this adjustment, the global Y Cartesian coordinate axis is used as the axis of rotation (KEYOPT(3) = 1). When using the OMEG real constant, you can specify either numerical values or table inputs. If specifying table inputs, enclose the table name in % signs (for example, %`tabname`%). Rotational speed (OMEG) can vary with time and location. Use the ***DIM** command to dimension the table and identify the variables. For more information and examples on using table inputs, see Array Parameters of the *ANSYS Parametric Design Language Guide*, Applying Loads Using TABLE Type Array Parameters in the *Basic Analysis Guide* and Doing a Thermal Analysis Using Tabular Boundary Conditions in the *Thermal Analysis Guide*, as well as the description of ***DIM** in the *Command Reference*.

A film coefficient specified by the **SFE** command may be modified by activating the user subroutine USERCV with the **USRCAL** command. USERCV may be used to modify the film coefficient of a surface element with or without an extra node. It may be used if the film coefficient is a function of temperature and/or location.

If the surface element has an extra node (KEYOPT(5) = 1), the bulk temperature and/or the film coefficient may be redefined in a general way by user programmable routine USRSURF116. USRSURF116 may be used if the bulk temperature and/or the film coefficient is a function of fluid properties, velocity and/or wall temperature. If a bulk temperature is determined by USRSURF116, it overrides any value specified by **SFE** or according to KEYOPT(6). Also, if a film coefficient is determined by USRSURF116, it overrides any values specified by **SFE** or USRCAL, USERCV. USRSURF116 calculation are activated by modifying the USRSURF116 subroutine and creating a customized version of ANSYS; there will be no change in functionality without modifying USRSURF116. For more information, see User-Programmable Features (UPFs).

Heat generation rates are input on a per unit volume basis and may be input as an element body load at the nodes, using the **BFE** command. Element body loads are not applied to other elements connected at the same nodes. The node I heat generation HG(I) defaults to zero. The node J heat generation defaults to HG(I). The heat generation load vector calculation uses the heat generation rate values.

As an alternative to using the **BFE** command, you can specify heat generation rates directly at the nodes using the **BF** command. For more information on body loads, see Body Loads in the *Basic Analysis Guide*.

SURF151 allows for radiation between the surface and the extra node. The emissivity of the surface (input as material property EMIS for the material number of the element) is used for the radiation surface conductivity matrix. The form factor FORMF and the Stefan-Boltzmann constant SBCONST are also used for the radiation surface conductivity matrix. The form factor can be either input as a real constant (defaults to 1) using KEYOPT(9) = 1 or it can be calculated automatically as a cosine effect using KEYOPT(9) = 2 or 3. For information on how the cosine effect depends on basic element orientation and the extra node location, see the *Mechanical APDL Theory Reference*. There is no distance effect included in the cosine effect. For axisymmetric analyses, the automatic form factor calculation is used only with the extra node on the Y-axis. The Stefan-Boltzmann constant defaults to 0.119×10^{-10} (Btu/hr*in^2* °R^4).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When KEYOPT(4) = 0, an edge with a removed midside node implies that the temperature varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.

If a single PLANE element lies beneath SURF151, you can automatically set the element behavior (plane stress, axisymmetric, or plane stress with thickness [including TKPS if applicable]) to that of the underlying solid element using KEYOPT(3) =10. This option is valid only when a single PLANE element lies beneath the SURF element. For example, if you apply a SURF151 element over a PLANE77 (thermal) element whose nodes are also used in the definition of a PLANE183 (structural) element, a warning appears and the load is not applied to the element.

A summary of the element input is given in "SURF151 Input Summary" (p. 511). A general description of element input is given in Element Input (p. 43). For axisymmetric applications see Harmonic Axisymmetric Elements (p. 83).

SURF151 Input Summary

Nodes

I, J if KEYOPT (4) = 1, and KEYOPT (5) = 0
I, J, K if KEYOPT (4) = 1, and KEYOPT(5) = 1
I, J, K, L if KEYOPT (4) = 1, and KEYOPT(5) = 2
I, J, K if KEYOPT (4) = 0, and KEYOPT(5) = 0
I, J, K, L if KEYOPT (4) = 0, and KEYOPT(5) = 1
I, J, K, L, M if KEYOPT (4) = 0, and KEYOPT(5) = 2

Degrees of Freedom
TEMP

Real Constants

FORMF, SBCONST, (Blank), OMEG, NRF, VABS,
TKI, TKJ, (Blank), (Blank), (Blank), TKPS,
ENN, GC, JC
See Table 151.1: SURF151 Real Constants (p. 514) for a description of the real constants

Material Properties

MP command: DENS (density), EMIS (emissivity) if KEYOPT(9) > 0)

Surface Loads

Convections --
face 1 (I-J) if KEYOPT(8) > 1

Heat Fluxes --
face 1 (I-J) if KEYOPT(8) = 1

Body Loads

Heat Generation --
HG(I), HG(J); also HG(K) if KEYOPT(4) = 0

Special Features
Birth and death

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(1)

Adiabatic wall temperature option:

0, 1, 2 --

See Adiabatic Wall Temperature as Bulk Temperature for information on these options.

KEYOPT(2)

Recovery factor (F_R) option:

0, 1, or 2 --

See Adiabatic Wall Temperature as Bulk Temperature for information on these options.

KEYOPT(3)

Element behavior:

0 --

Plane

1 --

Axisymmetric

3 --

Plane with thickness input (TKPS)

10 --

Use the element behavior--plane, axisymmetric, or plane with thickness input (including TKPS if applicable)--of the underlying solid element.

KEYOPT(4)

Midside nodes:

0 --

Has midside node (that matches the adjacent solid element)

1 --

No midside node

KEYOPT(5)

Extra node for radiation and/or convection calculations:

0 --

No extra nodes

1 --

Has extra node (optional if KEYOPT(8) > 1; required if KEYOPT(9) > 0)

2 --

Two extra nodes (optional if KEYOPT (8) > 1). Only valid for convection calculations. Use this option if the bulk temperature is unknown. The extra nodes get bulk temperatures from the two nodes of a FLUID116 element. This is generally more accurate than the one extra node option.

KEYOPT(6) (used only if KEYOPT(5) = 1 and KEYOPT(8) > 1)

Use of bulk temperatures:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0 --
Extra node temperature used as bulk temperature

1 --
Adiabatic wall temperature used as bulk temperature

KEYOPT(7)
Empirical term:

0 --
Do not multiply film coefficient by empirical term.

1 --
Multiply film coefficient by empirical term $|TS-TB|^n$.

KEYOPT(8)
Heat flux and convection loads:

0 --
Ignore heat flux and convection surface loads (if any)

1 --
Include heat flux, ignore convection

Use the following to include convection (ignore heat flux):

2 --
Evaluate film coefficient h_f (if any) at average film temperature, (TS +TB)/2

3 --
Evaluate h_f at element surface temperature, TS

4 --
Evaluate h_f at fluid bulk temperature, TB

5 --
Evaluate h_f at differential temperature, | TS - TB |

KEYOPT(9)
Radiation form factor calculation:

0 --
Do not include radiation

1 --
Use radiation with the form factor real constant

2 --
Use radiation with cosine effect computed as an absolute value (ignore real constant)

3 --
Use radiation with cosine effect computed as zero if negative (ignore real constant)

KEYOPT(13)
Film coefficient matrix key:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0 --
Program determines whether to use diagonal or consistent film coefficient matrix (default).

1 --
Use a diagonal film coefficient matrix.

2 --
Use a consistent matrix for the film coefficient.

Table 151.1: SURF151 Real Constants

No.	Name	Description
1	FORMF	Form factor
2	SBCONT	Stefan-Boltzmann constant
3	(Blank)	--
4	OMEGA	Angular velocity
5	NRF	Recovery factor
6	VABS	Absolute value of fluid velocity (KEYOPT(1) = 0)
7	TKI	In-plane thickness at node I
8	TKJ	In-plane thickness at node J (defaults to TKI)
9-11	(Blank)	--
12	TKPS	Out-of-plane thickness (if KEYOPT(3) = 3)
13	ENN	Empirical coefficient
14	GC	Gravitational constant used for units consistency
15	JC	Joule constant used to convert work units to heat units

SURF151 Output Data

The solution output associated with the element is in two forms:

- Nodal degree of freedom results included in the overall nodal solution
- Additional element output as shown in Table 151.2: SURF151 Element Output Definitions (p. 515)

Convection heat flux is positive out of the element; applied heat flux is positive into the element. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 151.2: SURF151 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
SURFACE NODES	Nodes - I, J	Y	Y
EXTRA NODE	Extra node (if present)	Y	Y
MAT	Material number	Y	Y
AREA	Surface area	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	7
VN(X, Y)	Components of unit vector normal to center of element	-	Y
DENSITY	Density	-	1
MASS	Mass of Element	-	1
HGEN	Heat generations HG(I), HG(J), HG(K)	2	-
HEAT GEN. RATE	Heat generation rate over entire element (HGTOT)	2	2
HFLUX	Input heat flux at nodes I, J	3	-
HEAT FLOW RATE	Input heat flux heat flow rate over element surface area (HFCTOT)	3	3
HFILM	Film coefficient at each face node	4	4
TBULK	Bulk temperature at each face node or temperature of extra node	4	4
TAVG	Average surface temperature	4	4
TAW	Adiabatic wall temperature	5	5
RELVEL	Relative velocity	5	5
SPHTFL	Specific heat of the fluid	5	5
RECFAC	Recovery factor	5	5
CONV. HEAT RATE	Convection heat flow rate over element surface area (HFCTOT)	4	4
CONV. HEAT RATE/AREA	Average convection heat flow rate per unit area	4	-
EMISSUR	Average emissivity of surface (for element material number)	6	6
EMISEXT	Emissivity of extra node	6	6
TEMPSUR	Average temperature of surface	6	6
TEMPEXT	Temperature of extra node	6	6
FORM FACTOR	Average form factor of element	6	6
RAD. HEAT RATE	Radiation heat flow rate over entire element (HRTOT)	6	6

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
RAD. HEAT RATE/AREA	Average radiation heat flow rate per unit area	6	-

1. If dens > 0
2. If heat generation load is present
3. If KEYOPT(8) = 1
4. If KEYOPT(8) > 1
5. If KEYOPT(6) = 1 and KEYOPT(8) > 1
6. If KEYOPT(9) > 0
7. Available only at centroid as a ***GET** item.

Table 151.3: SURF151 Item and Sequence Numbers (p. 516) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 151.3: SURF151 Item and Sequence Numbers (p. 516):

Name
output quantity as defined in Table 151.2: SURF151 Element Output Definitions (p. 515)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J
sequence number for data at nodes I and J

Table 151.3: SURF151 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
HGTOT	SMISC	1
HFCTOT	SMISC	2
HRTOT	SMISC	3
AREA	NMISC	1
VNX	NMISC	2
VNY	NMISC	3
HFILM	NMISC	5
TAVG	NMISC	6
TBULK	NMISC	7

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
TAW	NMISC	8
RELVEL	NMSC	9
SPHTFL	NMSC	10
RECFAC	NMSC	11
EMISSUR	NMISC	12
EMISEXT	NMISC	13
TEMPSUR	NMISC	14
TEMPEXT	NMISC	15
FORM FACTOR	NMISC	16
DENS	NMISC	17
MASS	NMISC	18

SURF151 Assumptions and Restrictions

- The element must not have a zero length.
- If KEYOPT(9) > 0 (radiation is used):
  - element is nonlinear and requires an iterative solution
  - extra node must be present.
  - if KEYOPT(4) = 0, midside nodes may not be dropped.

SURF151 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The TKPS real constant (R12) is not applicable.
- The only allowable material property is EMIS.
- No special features are allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

SURF152

3-D Thermal Surface Effect

MP ME <> PR PRN DS <> <> PP EME MFS
Product Restrictions

SURF152 Element Description

SURF152 may be used for various load and surface effect applications. It may be overlaid onto an area face of any 3-D thermal element. The element is applicable to 3-D thermal analyses. Various loads and surface effects may exist simultaneously. See SURF152 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 152.1: SURF152 Geometry

SURF152 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 152.1: SURF152 Geometry (p. 519). The element is defined by four to ten nodes and the material properties. An extra node (away from the base element) may be used for convection or radiation effects. Two extra nodes (away from the base element) may be used to more accurately capture convection effects. A triangular element may be formed by defining duplicate K and L node numbers as described in Degenerated Shape Elements (p. 39). The element x-axis is parallel to the I-J side of the element.

The mass, volume, and heat-generation calculations use the element thicknesses at node I, J, K, and L (real constants TKI, TKJ, TKK, and TKL, respectively). Thickness TKI defaults to 0.0, and thicknesses TKJ, TKK, and TKL default to TKI. The mass calculation uses the density (material property DENS).

See Nodal Loading (p. 49) for a description of element loads. Convections or heat fluxes may be input as surface loads on the element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The convection surface conductivity matrix calculation uses the film coefficient (input on the **SFE** command with *KVAL* = 0 and *CONV* as the label). If the extra node is used, its temperature becomes the bulk temperature. If the extra node is not used, the *CONV* value input with *KVAL* = 2 becomes the bulk temperature. The convection surface heat flow vector calculation uses the bulk temperature. On a given face, either a heat flux or a convection may be specified, but not both simultaneously.

For the extra node option (KEYOPT(5) = 1), film effectiveness and free stream temperatures may also be input for convection surface loads (input on the **SFE** command with the *CONV* label and *KVAL* = 3 and 4, respectively). If film effectiveness is input, bulk temperature is ignored.

Setting KEYOPT(7) = 1 multiplies the evaluated film coefficient by the empirical term ITS - TBI^n, where TS is the element surface temperature, TB is the fluid bulk temperature, and n is an empirical coefficient (real constant ENN).

If KEYOPT(5) = 1 and flow information is available from FLUID116 with KEYOPT(2) = 1, the bulk temperature may be adjusted to the adiabatic wall temperature using KEYOPT(6) = 1, real constants OMEG (rotational speed) and NRF (recovery factor), and the logic described in the *Mechanical APDL Theory Reference*. For this adjustment, the axis of rotation may be defined as the global Cartesian X, Y or Z coordinate axis (KEYOPT(3)). When using the OMEG real constant, you can specify either numerical values or table inputs. If specifying table inputs, enclose the table name in % signs (for example, %`tabname`%). Rotational speed (OMEG) can vary with time and location. Use the ***DIM** command to dimension the table and identify the variables. For more information and examples on using table inputs, see Array Parameters of the *ANSYS Parametric Design Language Guide*, Applying Loads Using TABLE Type Array Parameters in the *Basic Analysis Guide*, and Doing a Thermal Analysis Using Tabular Boundary Conditions in the *Thermal Analysis Guide*, as well as the description of the ***DIM** command in the *Command Reference*.

A film coefficient specified by the **SFE** command may be modified by activating the user subroutine USERCV with the **USRCAL** command. USERCV may be used to modify the film coefficient of a surface element with or without an extra node. It may be used if the film coefficient is a function of temperature and/or location.

If the surface element has an extra node (KEYOPT(5) = 1), the bulk temperature and/or the film coefficient may be redefined in a general way by user programmable routine USRSURF116. USRSURF116 may be used if the bulk temperature and/or the film coefficient is a function of fluid properties, velocity and/or wall temperature. If a bulk temperature is determined by USRSURF116, it overrides any value specified by **SFE** or according to KEYOPT(6). Also, if a film coefficient is determined by USRSURF116, it overrides any values specified by **SFE** or USRCAL, USERCV. USRSURF116 calculation are activated by modifying the USRSURF116 subroutine and creating a customized version of ANSYS; there will be no change in functionality without modifying USRSURF116. For more information, see User-Programmable Features (UPFs).

Heat generation rates are input on a per unit volume basis and may be input as an element body load at the nodes, using the **BFE** command. Element body loads are not applied to other elements connected at the same nodes. The node I heat generation HG(I) defaults to zero. If all other heat generations are unspecified, they default to HG(I). If all corner node heat generations are specified, each midside node heat generation defaults to the average heat generation of its adjacent corner nodes. For any other input heat generation pattern, unspecified heat generations default to zero. The heat generation load vector calculation uses the heat generation rate values.

As an alternative to using the **BFE** command, you can specify heat generation rates directly at the nodes using the **BF** command. For more information on body loads, see Body Loads in the *Basic Analysis Guide*.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SURF152 allows for radiation between the surface and the extra node. The emissivity of the surface (input as material property EMIS for the material number of the element) is used for the radiation surface conductivity matrix. The form factor FORMF and the Stefan-Boltzmann constant SBCONST are also used for the radiation surface conductivity matrix. The form factor can be either input as a real constant (defaults to 1) using KEYOPT(9) = 1 or it can be calculated automatically as a cosine effect using KEYOPT(9) = 2 or 3. For information on how the cosine effect depends on basic element orientation and the extra node location, see the *Mechanical APDL Theory Reference*. There is no distance effect included in the cosine effect. The Stefan-Boltzmann constant defaults to 0.119×10^{-10} (Btu/hr*in^2* °R^4)).

When KEYOPT(4) = 0, an edge with a removed midside node implies that the temperature varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.

A summary of the element input is given in "SURF152 Input Summary" (p. 521). A general description of element input is given in Element Input (p. 43).

SURF152 Input Summary

Nodes

I, J, K, L if KEYOPT (4) = 1 and KEYOPT (5) = 0
I, J, K, L, M if KEYOPT (4) = 1 and KEYOPT (5) = 1
I, J, K, L, M, N if KEYOPT (4) = 1 and KEYOPT (5) = 2
I, J, K, L, M, N, O, P if KEYOPT (4) = 0 and KEYOPT (5) = 0
I, J, K, L, M, N, O, P, Q if KEYOPT (4) = 0 and KEYOPT (5) = 1
I, J, K, L, M, N, O, P, Q, R if KEYOPT (4) = 0 and KEYOPT (5) = 2

Degrees of Freedom

KEYOPT(11) Setting	DOF for all nodes except extra node(s)	DOF for extra node(s) (KEYOPT(5) = 1 or 2)
0	TEMP	TEMP
1	TTOP	TEMP
2	TBOT	TEMP

Real Constants

FORMF, SBCONST, (Blank), OMEG, NRF, VABS,
TKI, TKJ, TKK, TKL, (Blank), (Blank),
ENN, GC, JC
See Table 152.1: SURF152 Real Constants (p. 524) for a description of the real constants

Material Properties

MP command: DENS (density), EMIS (emissivity) if KEYOPT(9) > 0)

Surface Loads

Convections --
face 1 (I-J-K-L) if KEYOPT(8) > 1

Heat Fluxes --
face 1 (I-J-K-L) if KEYOPT(8) = 1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Body Loads

Heat Generation --
HG(I), HG(J), HG(K), HG(L), and, if KEYOPT(4) = 0, HG(M), HG(N), HG(O), HG(P)

Special Features
Birth and death

KEYOPT(1)
Adiabatic wall temperature option:

0, 1, 2 --
See Adiabatic Wall Temperature as Bulk Temperature for information on these options.

KEYOPT(2)
Recovery factor (F_R) option:

0, 1, or 2 --
See Adiabatic Wall Temperature as Bulk Temperature for information on these options.

KEYOPT(3)
Axis of symmetry:

0 --
OMEG used about global Cartesian X-axis

1 --
OMEG used about global Cartesian Y-axis

2 --
OMEG used about global Cartesian Z-axis

KEYOPT(4)
Midside nodes:

0 --
Has midside nodes (that match the adjacent solid element)

1 --
Does not have midside nodes

KEYOPT(5)
Extra nodes:

0 --
No extra nodes. Use this option if the bulk temperature is known.

1 --
One extra node (optional if KEYOPT (8) > 1; required if KEYOPT (9) > 0). Valid for convection and radiation calculations. Use this option if the bulk temperature is unknown. The extra node gets the bulk temperature from a FLUID116 element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2 --
Two extra nodes (optional if KEYOPT (8) > 1). Only valid for convection calculations. Use this option if the bulk temperature is unknown. The extra nodes get bulk temperatures from the two nodes of a FLUID116 element. This is generally more accurate than the one extra node option.

KEYOPT(6) (used only if KEYOPT(5) = 1 and KEYOPT(8) > 1)
Use of bulk temperature:

0 --
Extra node temperature used as bulk temperature

1 --
Adiabatic wall temperature used as bulk temperature

KEYOPT(7)
Empirical term:

0 --
Do not multiply film coefficient by empirical term.

1 --
Multiply film coefficient by empirical term $|TS\text{-}TB|^n$.

KEYOPT(8)
Heat flux and convection loads:

0 --
Ignore heat flux and convection surface loads (if any)

1 --
Include heat flux, ignore convection

Use the following to include convection (ignore heat flux):

2 --
Evaluate film coefficient h_f (if any) at average film temperature, (TS +TB)/2

3 --
Evaluate h_f at element surface temperature, TS

4 --
Evaluate h_f at fluid bulk temperature, TB

5 --
Evaluate h_f at differential temperature, | TS - TB |

KEYOPT(9)
Radiation form factor calculation:

0 --
Do not include radiation

1 --
Use radiation with the form factor real constant

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2 --
Use radiation with cosine effect calculated as an absolute value (ignore real constant)

3 --
Use radiation with cosine effect calculated as zero if negative (ignore real constant)

KEYOPT(11)
Label used for all nodal degrees of freedom (except for the extra node):

0 --
TEMP

1 --
TTOP

2 --
TBOT

The extra node, if requested with KEYOPT(5) = 1, is always TEMP.

KEYOPT(13)
Film coefficient matrix key:

0 --
Program determines whether to use diagonal or consistent film coefficient matrix (default).

1 --
Use a diagonal film coefficient matrix.

2 --
Use a consistent matrix for the film coefficient.

Table 152.1: SURF152 Real Constants

No.	Name	Description
1	FORMF	Form factor
2	SBCONT	Stefan-Boltzmann constant
3	(Blank)	--
4	OMEGA	Angular velocity (KEYOPT(6) = 1)
5	NRF	Recovery factor
6	VABS	Absolute value of fluid velocity (KEYOPT(1) = 0)
7	TKI	Thickness at node I
8	TKJ	Thickness at node J (defaults to TKI)
9	TKK	Thickness at node K (defaults to TKI)
10	TKL	Thickness at node L (defaults to TKI)
11-12	(Blank)	--
13	ENN	Empirical coefficient
14	GC	Gravitational constant used for units consistency
15	JC	Joule constant used to convert work units to heat units

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SURF152 Output Data

The solution output associated with the element is in two forms:

- Nodal degree of freedom results included in the overall nodal solution
- Additional element output as shown in Table 152.2: SURF152 Element Output Definitions (p. 525)

Convection heat flux is positive out of the element; applied heat flux is positive into the element. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 152.2: SURF152 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
SURFACE NODES	Nodes - I, J, K, L	Y	Y
EXTRA NODE	Extra node (if present)	Y	Y
MAT	Material number	Y	Y
AREA	Surface area	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	7
VN(X, Y, Z)	Components of unit vector normal to center of element	-	Y
DENSITY	Density	-	1
MASS	Mass of element	-	1
HGEN	Heat generations HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P)	2	-
HEAT GEN. RATE	Heat generation rate over entire element (HGTOT)	2	2
HFLUX	Input heat flux at nodes I, J, K, L	3	-
HEAT FLOW RATE	Input heat flux heat flow rate over element surface area (HFCTOT)	3	3
HFILM	Film coefficient at each face node	4	4
TBULK	Bulk temperature at each face node or temperature of extra node	4	4
TAVG	Average surface temperature	4	4
TAW	Adiabatic wall temperature	5	5

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
RELVEL	Relative velocity	5	5
SPHTFL	Specific heat of the fluid	5	5
RECFAC	Recovery factor	5	5
CONV. HEAT RATE	Convection heat flow rate over element surface area (HFCTOT)	4	4
CONV. HEAT RATE/AREA	Average convection heat flow rate per unit area	4	-
EMISSUR	Average emissivity of surface (for element material number)	6	6
EMISEXT	Emissivity of extra node	6	6
TEMPSUR	Average temperature of surface	6	6
TEMPEXT	Temperature of extra node	6	6
FORM FACTOR	Average form factor of element	6	6
RAD. HEAT RATE	Radiation heat flow rate over entire element (HRTOT)	6	6
RAD. HEAT RATE/AREA	Average radiation heat flow rate per unit area	6	-

1. If dens > 0
2. If heat generation load is present
3. If KEYOPT(8) = 1
4. If KEYOPT(8) > 1
5. If KEYOPT(6) = 1 and KEYOPT(8) > 1
6. If KEYOPT(9) > 0
7. Available only at centroid as a ***GET** item.

Table 152.3: SURF152 Item and Sequence Numbers (p. 527) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 152.3: SURF152 Item and Sequence Numbers (p. 527):

Name
output quantity as defined in Table 152.2: SURF152 Element Output Definitions (p. 525)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

I,J,K,L

sequence number for data at nodes I, J, K, L

Table 152.3: SURF152 Item and Sequence Numbers

Output Quant-ity Name	ETABLE and ESOL Command Input	
	Item	E
HGTOT	SMISC	1
HFCTOT	SMISC	2
HRTOT	SMISC	3
AREA	NMISC	1
VNX	NMISC	2
VNY	NMISC	3
VNZ	NMISC	4
HFILM	NMISC	5
TAVG	NMISC	6
TBULK	NMISC	7
TAW	NMISC	8
RELVEL	NMISC	9
SPHTFL	NMISC	10
RECFAC	NMISC	11
EMISSUR	NMISC	12
EMISEXT	NMISC	13
TEMPSUR	NMISC	14
TEMPEXT	NMISC	15
FORM FACTOR	NMISC	16
DENS	NMISC	17
MASS	NMISC	18

SURF152 Assumptions and Restrictions

- The element must not have a zero area.
- If KEYOPT(9) > 0 (radiation is used):
  - element is nonlinear and requires an iterative solution
  - extra node must be present.
  - if KEYOPT(4) = 0, midside nodes may not be dropped.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SURF152 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The only allowable material property is EMIS.
- No special features are allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SURF153

2-D Structural Surface Effect

MP ME ST PR PRN DS <> <> PP EME MFS
Product Restrictions

SURF153 Element Description

SURF153 may be used for various load and surface effect applications. It may be overlaid onto a face of any 2-D structural solid element (except axisymmetric harmonic elements PLANE25 and PLANE83). The element is applicable to 2-D structural analyses. See SURF153 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 153.1: SURF153 Geometry

SURF153 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 153.1: SURF153 Geometry (p. 529). The element is defined by two or three node points and the material properties. The element x-axis is along to the I-J line of the element.

The mass and volume calculations use the in-plane element thicknesses at nodes I and J (real constants TKI and TKJ, respectively). Thickness TKI defaults to 0.0, and thickness TKJ defaults to TKI. If KEYOPT(3) = 3, the out-of-plane thickness is input as the real constant TKPS (defaults to 1.0). The mass calculation uses the density (material property DENS, mass per unit volume) and the real constant ADMSUA, the added mass per unit area.

The stress stiffness matrix and load vector calculations use the in-plane force per unit length (input as real constant SURT) and the elastic foundation stiffness (input as real constant EFS); the EFS uses pressure-per-length (or force-per-length-cubed) units. The foundation stiffness can be damped, either by using the material property BETD as a multiplier on the stiffness or by directly using the material property VISC.

See Nodal Loading (p. 49) for a description of element loads. Pressures may be input as surface loads as force-per-length-squared on the element faces as shown by the circled numbers on Figure 153.2: Pressures (p. 530). SURF153 allows complex pressure loads.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 153.2: Pressures

Faces 1 and 2 [KEYOPT(2) = 0] Positive values of pressure on the first two faces act in the positive element coordinate directions (except for the normal pressure which acts in the negative z direction). For face 1, positive or negative values may be removed as requested with KEYOPT(6) to simulate the discontinuity at the free surface of a contained fluid.

Faces 1 and 2 [KEYOPT(2) = 1] Pressure loads are applied to the element faces according to the local coordinate system as follows: face 1 in the local x direction and face 2 in the local y direction. A local coordinate system must be defined, and the element must be set to that coordinate system via the **ESYS** command; however, the loads specified in that local coordinate system do not follow the large displacements and/or rotations of the element. KEYOPT(6) does not apply.

Face 3 The magnitude of the pressure at each integration point is $P_I + XP_J + YP_K$, where P_I through P_K are input as *VAL1* through *VAL3* on the **SFE** command, and X and Y are the global Cartesian coordinates at the current location of the point. No input values can be blank. The **SFFUN** and **SFGRAD** commands do not work with face 3.

Face 4 The magnitude of the pressure is P_I, and the direction is $(P_J i + P_K j)/(P_J^2 + P_K^2)^{1/2}$ where i and j are unit vectors in the global Cartesian directions. The load magnitude can be adjusted with KEYOPTs(11) and (12). No input values can be blank. When using the **SFFUN** or **SFGRAD** commands, the load direction is not altered, but the load magnitude is the average of the computed corner node magnitudes. **SFCUM**,ADD should be used with caution, as this command also causes the load direction components to be added.

The effects of pressure load stiffness are automatically included for this element for real pressure on face 1 if KEYOPT(2) = 0 or on face 3. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

Temperatures may be input as element body loads at the nodes. Element body load temperatures are not applied to other elements connected at the same nodes. The node I temperature T(I) defaults to TUNIF. The node J temperature defaults to T(I). Temperatures are used for material property evaluation only.

When KEYOPT(4) = 0, a removed midside node implies that the displacement varies linearly, rather than parabolically. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If a single PLANE element lies beneath SURF153, you can automatically set the element behavior (plane stress, axisymmetric, or plane stress with thickness [including TKPS if applicable]) to that of the underlying solid element using KEYOPT(3) =10. This option is valid only when a single PLANE element lies beneath the SURF element. For example, if you apply a SURF153 element over a PLANE77 (thermal) element whose nodes are also used in the definition of a PLANE183 (structural) element, a warning appears and the load is not applied to the element.

KEYOPT(7) = 1 is useful when the element is used to represent a force. When KEYOPT(7) = 0, the force is input as a pressure times an area; however, if the area changes due to large deflections, the force also changes. When KEYOPT(7) = 1, the force remains unchanged even if the area changes.

A summary of the element input is given in "SURF153 Input Summary" (p. 531). A general description of element input is given in Element Input (p. 43). For axisymmetric applications see Harmonic Axisymmetric Elements (p. 83).

SURF153 Input Summary

Nodes

I, J if KEYOPT (4) = 1,
I, J, K if KEYOPT(4) = 0

Degrees of Freedom
UX, UY

Real Constants

(Blank), (Blank), (Blank), EFS, SURT, ADMSUA,
TKI, TKJ, (Blank), (Blank), (Blank), TKPS
See Table 153.1: SURF153 Real Constants (p. 533) for a description of the real constants

Material Properties
MP command: DENS, VISC, ALPD, BETD

Surface Loads

Pressures --

face 1 (I-J) (in -y normal direction)
face 2 (I-J) (in +x tangential direction)
face 3 (I-J) (in -y normal direction, global taper)
face 4 (I-J) (oriented by input vector)

Body Loads

Temperatures --
T(I), T(J); also T(K) if KEYOPT(4) = 0

Special Features

Birth and death
Large deflection
Linear perturbation
Rezoning (see "SURF153 Assumptions and Restrictions" (p. 536))
Stress stiffening

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(2)

Pressure applied to faces 1 and 2 according to coordinate system:

0 --
Apply face loads in the element coordinate system

1 --
Apply face loads in the local coordinate system

KEYOPT(3)

Element behavior:

0 --
Plane stress

1 --
Axisymmetric

2 --
Plane strain

3 --
Plane stress with thickness input (TKPS)

5 --
Generalized plane strain

10 --
Use the element behavior--plane stress, axisymmetric, plain strain, plane stress with thickness input (including TKPS if applicable), or generalized plane strain--of the underlying solid element.

KEYOPT(4)

Midside nodes:

0 --
Has midside node (that matches the adjacent solid element)

1 --
No midside node

KEYOPT(6)

Applicable only to normal direction pressure (faces 1 and 3):

0 --
Use pressures as calculated (positive and negative)

1 --
Use positive pressures only (negative set to zero)

2 --
Use negative pressures only (positive set to zero)

KEYOPT(7)

Loaded area during large-deflection analyses:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0 --
Use new area

1 --
Use original area

KEYOPT(11)
Pressure applied by vector orientation (face 4):

0 --
On projected area and includes tangential component

1 --
On projected area and does not include tangential component

2 --
On full area and includes the tangential component

KEYOPT(12)
Effect of the direction of the element normal (element y-axis) on vector-oriented (face 4) pressure:

0 --
Pressure load is applied regardless of the element normal orientation

1 --
Pressure load is not used if the element normal is oriented in the same general direction as the pressure vector

Table 153.1: SURF153 Real Constants

No.	Name	Description
1 ... 3	(Blank)	--
4	EFS	Foundation stiffness
5	SURT	Surface tension
6	ADMSUA	Added mass/unit area
7	TKI	In-plane thickness at node I
8	TKJ	In-plane thickness at node J (defaults to TKI)
9 ... 11	(Blank)	--
12	TKPS	Out-of-plane thickness if KEYOPT(3) = 3 (defaults to 1.0)

SURF153 Output Data

The solution output associated with the element is in two forms:

- Nodal degree of freedom results included in the overall nodal solution
- Additional element output as shown in Table 153.2: SURF153 Element Output Definitions (p. 534)

A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 153.2: SURF153 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
SURFACE NODES	Nodes - I, J	Y	Y
EXTRA NODE	Extra node (if present)	Y	Y
MAT	Material number	Y	Y
AREA	Surface area	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	6
VN(X, Y)	Components of unit vector normal to center of element	-	Y
PRESSURES(F/L2)	Pressures P1, P2, P3, P4 at nodes I, J (Face indicated by PRES LOAD KEY)	1	-
PY, PX	Pressures at nodes in element coordinate system (P4 uses an average element coordinate system)	-	1
AVG. FACE PRESSURE	Average normal pressure (P1AVG), Average tangential pressure (P2AVG), Average tapered normal pressure (P3AVG), Effective value of vector oriented pressure (P4EFF)	1	1
DVX, DVY	Direction vector of pressure P4	1	1
TEMP	Surface temperatures T(I), T(J), T(K)	2	2
DENSITY	Density (input as DENS)	3	3
MASS	Mass of Element	3	3
FOUNDATION STIFFNESS	Foundation Stiffness (input as EFS)	4	4
FOUNDATION PRESSURE	Foundation Pressure	4	4
SURFACE TENSION	Surface Tension (input as SURT)	5	5

1. If pressure load
2. If temperature load
3. If DENS > 0
4. If EFS > 0
5. If SURT > 0
6. Available only at centroid as a ***GET** item.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Table 153.3: SURF153 Item and Sequence Numbers (p. 535) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 153.3: SURF153 Item and Sequence Numbers (p. 535):

Name
output quantity as defined in the Table 153.2: SURF153 Element Output Definitions (p. 534)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J
sequence number for data at nodes I and J

Table 153.3: SURF153 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
PY (real)	SMISC	-	1	2
PX (real)	SMISC	-	3	4
PY (imaginary)	SMISC		27	28
PX (imaginary)	SMISC		29	30
P1AVG (real)	SMISC	13	-	-
P2AVG (real)	SMISC	14	-	-
P3AVG (real)	SMISC	15	-	-
P4EFF (real)	SMISC	16	-	-
P1AVG (imaginary)	SMISC	39	-	-
P2AVG (imaginary)	SMISC	40	-	-
P3AVG (imaginary)	SMISC	41	-	-
P4EFF (imaginary)	SMISC	42	-	-
FOUNPR	SMISC	21	-	-
AREA	NMISC	1	-	-
VNX	NMISC	2	-	-
VNY	NMISC	3	-	-
EFS	NMISC	5	-	-
SURT	NMISC	6	-	-
DENS	NMISC	7	-	-
MASS	NMISC	8	-	-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
DVX	NMISC	9	-	-
DVY	NMISC	10	-	-

SURF153 Assumptions and Restrictions

- The element must not have a zero length.
- The surface tension load vector acts along the line connecting nodes I and J as a force applied to the nodes seeking to minimize the length of the line. If the nodes of the element are not coplanar when using surface tension, equilibrium may be lost.
- For structural large deflection analyses, the loads are applied to the current size of the element, not the initial size.
- Surface printout and foundation stiffness are not valid for elements deactivated [**EKILL**] and then reactivated [**EALIVE**]. Surface printout does not include large strain effects.
- For rezoning, only normal and tangential pressures applied on SURF153 are supported. (That is, rezoning support is available only for pressure on faces 1 and 2.)

SURF153 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The only special features allowed are stress stiffening and large deflections.
- The VISC and BETD material properties are not applicable.

ANSYS Structural

- The only allowable material property is DENS.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SURF154

3-D Structural Surface Effect

MP ME ST PR PRN DS <> <> PP EME MFS
Product Restrictions

SURF154 Element Description

SURF154 is used for various load and surface effect applications in 3-D structural analyses. It can be overlaid onto an area face of any 3-D element. Various loads and surface effects may exist simultaneously. See SURF154 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 154.1: SURF154 Geometry

SURF154 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 154.1: SURF154 Geometry (p. 537). The element is defined by four to eight nodes and the material properties. A triangular element may be formed by defining duplicate K and L node numbers as described in Degenerated Shape Elements (p. 39). The default element x-axis is parallel to the I-J side of the element.

The mass and volume calculations use the element thicknesses at nodes I, J, K, and L (real constants TKI, TKJ, TKK, and TKL, respectively). Thickness TKI defaults to 0.0, and thicknesses TKJ, TKK, and TKL default to TKI. The mass calculation uses the density (material property DENS, mass per unit volume) and the real constant ADMSUA, the added mass per unit area.

The stress stiffness matrix and load vector calculations use the in-plane force per unit length (input as real constant SURT) and the elastic foundation stiffness (input as real constant EFS); the EFS uses pressure-per-length (or force-per-length-cubed) units. The foundation stiffness can be damped, either by using the material property BETD as a multiplier on the stiffness or by directly using the material property VISC.

See Nodal Loading (p. 49) for a description of element loads. Pressures may be input as surface loads on the element faces as shown by the circled numbers on Figure 154.2: Pressures (p. 538). SURF154 allows complex pressure loads.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Faces 1, 2, and 3 [KEYOPT(2) = 0] Positive values of pressure on the first three faces act in the positive element coordinate directions (except for the normal pressure which acts in the negative z direction). For face 1, positive or negative values may be removed as requested with KEYOPT(6) to simulate the discontinuity at the free surface of a contained fluid. For faces 2 and 3, the direction of the load is controlled by the element coordinate system; therefore, the **ESYS** command is normally needed.

Faces 1, 2, and 3 [KEYOPT(2) = 1] Pressure loads are applied to the element faces according to the local coordinate system, as follows: face 1 in the local x direction, face 2 in the local y direction, and face 3 in the local z direction. A local coordinate system must be defined, and the element must be set to that coordinate system via the **ESYS** command; however, the loads specified in that local coordinate system do not follow the large displacements and/or rotations of the element. KEYOPT(6) does not apply.

Figure 154.2: Pressures

Face 4 The direction is normal to the element and the magnitude of the pressure at each integration point is $P_I + XP_J + YP_K + ZP_L$, where P_I through P_L are input as *VAL1* through *VAL4* on the **SFE** command, and X, Y, Z are the global Cartesian coordinates at the current location of the point. No input values can be blank. Face 4 can be used to apply hydrostatic ocean loading, with P_L = -(vertical acceleration) * (water density). Positive or negative values may be removed as requested with KEYOPT(6) to simulate the discontinuity at the free surface of a contained fluid. The **SFFUN** and **SFGRAD** commands do not work with face 4.

Face 5 The magnitude of the pressure is P_I, and the direction is $(P_J i + P_K j + P_L k)/(P_J^2 + P_K^2 + P_L^2)^{1/2}$ where i, j, and k are unit vectors in the global Cartesian directions. The load magnitude may be adjusted with KEYOPT(11) and KEYOPT(12). No input values can be blank. When using the **SFFUN** or **SFGRAD** commands, the load direction is not altered but the load magnitude is the average of the computed corner node magnitudes. **SFCUM**,ADD should be used with caution, as this command also causes the load direction components to be added.

The effects of pressure load stiffness are automatically included for this element for real pressure on face 1 if KEYOPT(2) = 0 or on face 4. If an unsymmetric matrix is needed for pressure load stiffness effects, issue a **NROPT**,UNSYM command.

Temperatures may be input as element body loads at the nodes. Element body load temperatures are not applied to other elements connected at the same nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). If all corner node temperatures are

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

specified, each midside node temperature defaults to the average temperature of its adjacent corner nodes. For any other input temperature pattern, unspecified temperatures default to TUNIF. Temperatures are used for material property evaluation only.

When KEYOPT(4) = 0, an edge with a removed midside node implies that the displacement varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.

KEYOPT(7) = 1 is useful when the element is used to represent a force. When KEYOPT(7) = 0, the force is input as a pressure times an area; however, if the area changes due to large deflections, the force also changes. When KEYOPT(7) = 1, the force remains unchanged even if the area changes.

KEYOPT(8) > 0 applies ocean loading on the element.

A summary of the element input is given in "SURF154 Input Summary" (p. 539). A general description of element input is given in Element Input (p. 43).

SURF154 Input Summary

Nodes

I, J, K, L if KEYOPT (4) = 1
I, J, K, L, M, N, O, P if KEYOPT (4) = 0

Degrees of Freedom
UX, UY, UZ

Real Constants

(Blank), (Blank), (Blank), EFS, SURT, ADMSUA,
TKI, TKJ, TKK, TKL
See Table 154.1: SURF154 Real Constants (p. 541) for a description of the real constants

Material Properties
MP command: DENS, VISC, ALPD, BETD

Surface Loads

Pressures --

face 1 (I-J-K-L) (in -z normal direction)
face 2 (I-J-K-L) (tangential (+x))
face 3 (I-J-K-L) (tangential (+y))
face 4 (I-J-K-L) (in -z normal direction, global taper)
face 5 (I-J-K-L) (oriented by input vector)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L); also T(M), T(N), T(O), T(P) if KEYOPT(4) = 0

Special Features

Birth and death
Large deflection

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Linear perturbation
Ocean loading
Rezoning (see "SURF154 Assumptions and Restrictions" (p. 544))
Stress stiffening

KEYOPT(2)
Pressure applied to faces 1, 2, and 3 according to coordinate system:

0 --
Apply face loads in the element coordinate system

1 --
Apply face loads in the local coordinate system

KEYOPT(4)
Midside nodes:

0 --
Has midside nodes (that match the adjacent solid element)

1 --
Does not have midside nodes

KEYOPT(6)
Applicable only to normal direction pressure (faces 1 and 4). This KEYOPT is valid only when KEYOPT(2) = 0.

0 --
Use pressures as calculated (positive and negative)

1 --
Use positive pressures only (negative set to zero)

2 --
Use negative pressures only (positive set to zero)

KEYOPT(7)
Loaded area during large-deflection analyses:

0 --
Use new area

1 --
Use original area

KEYOPT(8)
Specifies whether ocean loading is applied:

0 --
No ocean loading (default). This option applies even if the ocean-loading data is defined.

1 --
Ocean loading is applied if the ocean-loading data is defined. This option is valid when *KWAVE* = 8 or 101+ on the **OCDATA** command. For *KWAVE* = 101+, ocean loading is applied based on user subroutine `userPanelHydFor` computations.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(11)

Pressure applied by vector orientation (face 5):

0 --

On projected area and includes tangential component

1 --

On projected area and does not include tangential component

2 --

On full area and includes the tangential component

KEYOPT(12)

Effect of the direction of the element normal (element z-axis) on vector oriented (face 5) pressure:

0 --

Pressure load is applied regardless of the element normal orientation

1 --

Pressure load is not used if the element normal is oriented in the same general direction as the pressure vector.

Table 154.1: SURF154 Real Constants

No.	Name	Description
1 ... 3	(Blank)	--
4	EFS	Foundation stiffness
5	SURT	Surface tension
6	ADMSUA	Added mass/unit area
7	TKI	Thickness at node I
8	TKJ	Thickness at node J (defaults to TKI)
9	TKK	Thickness at node K (defaults to TKI)
10	TKL	Thickness at node L (defaults to TKI)

SURF154 Output Data

The solution output associated with the element is in two forms:

- Nodal degree of freedom results included in the overall nodal solution
- Additional element output as shown in Table 154.2: SURF154 Element Output Definitions (p. 542)

A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 154.2: SURF154 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
SURFACE NODES	Nodes - I, J, K, L	Y	Y
EXTRA NODE	Extra node (if present)	Y	Y
MAT	Material number	Y	Y
AREA	Surface area	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	1
VN(X, Y, Z)	Components of unit vector normal to center of element	-	Y
PRESSURES(F/L2)	Pressures P1 (includes hydrodynamic pressure), P2, P3, P4, and P5 at nodes I, J, K, L (Face indicated by PRES LOAD KEY)	2	-
PZ, PX, PY	Pressures at nodes in element coordinate system (P5 uses an average element coordinate system)	-	2
DVX, DVY, DVZ	Direction vector of pressure P5	2	2
AVERAGE FACE PRESSURES	- Average normal pressure (P1AVG), including hydrodynamic ocean pressure when KEYOPT(8) > 1 - Average tangential-X pressure (P2AVG) - Average tangential-Y pressure (P3AVG) - Average tapered normal pressure (P4AVG) - Effective value of vector oriented pressure (P5EFF)	2	2
AVERAGE OCEAN PRESSURE	Average normal hydrodynamic ocean pressure	3	3
TEMP	Surface temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	4	4
DENSITY	Density	5	5
MASS	Mass of element	5	5
FOUNDATION STIFFNESS	Foundation Stiffness (input as EFS)	6	6
FOUNDATION PRESSURE	Foundation Pressure	6	6
SURFACE TENSION	Surface Tension (input as SURT)	7	7

1. Available only at centroid as a ***GET** item
2. If pressure load
3. Only if KEYOPT(8) > 0
4. If temperature load

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. If dens > 0

6. If EFS > 0

7. If SURT > 0

Table 154.3: SURF154 Item and Sequence Numbers (p. 543) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (**/POST1**) of the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) of this manual for more information. The following notation is used in Table 154.3: SURF154 Item and Sequence Numbers (p. 543):

Name
output quantity as defined in the Table 154.2: SURF154 Element Output Definitions (p. 542)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J,K,L
sequence number for data at nodes I, J, K, L

Table 154.3: SURF154 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
PZ (real)	SMISC	-	1	2	3	4
PX (real)	SMISC	-	5	6	7	8
PY (real)	SMISC	-	9	10	11	12
PZ (imaginary)	SMISC	-	27	28	29	30
PX (imaginary)	SMISC	-	31	32	33	34
PY (imaginary)	SMISC	-	35	36	37	38
P1AVG (real)	SMISC	13	-	-	-	-
P2AVG (real)	SMISC	14	-	-	-	-
P3AVG (real)	SMISC	15	-	-	-	-
P4AVG (real)	SMISC	16	-	-	-	-
P5EFF (real)	SMISC	17	-	-	-	-
OCEAN PRESS (real)	SMISC	18	-	-	-	-
P1AVG (imaginary)	SMISC	39	-	-	-	-
P2AVG (imaginary)	SMISC	40	-	-	-	-
P3AVG (imaginary)	SMISC	41	-	-	-	-
P4AVG (imaginary)	SMISC	42	-	-	-	-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
P5EFF (imaginary)	SMISC	43	-	-	-	-
OCEAN PRESS (imaginary)	SMISC	44	-	-	-	-
FOUNPR	SMISC	21	-	-	-	-
AREA	NMISC	1	-	-	-	-
VNX	NMISC	2	-	-	-	-
VNY	NMISC	3	-	-	-	-
VNZ	NMISC	4	-	-	-	-
EFS	NMISC	5	-	-	-	-
SURT	NMISC	6	-	-	-	-
DENS	NMISC	7	-	-	-	-
MASS	NMISC	8	-	-	-	-
DVX	NMISC	9	-	-	-	-
DVY	NMISC	10	-	-	-	-
DVZ	NMISC	11	-	-	-	-

SURF154 Assumptions and Restrictions

- The element must not have a zero area.
- The surface-tension load vector acts in the plane of the element as a constant force applied to the nodes seeking to minimize the area of the surface. If the nodes of the element are not coplanar when using surface tension, equilibrium may be lost.
- For structural large-deflection analyses, the loads are applied to the current size of the element, not the initial size.
- Surface printout and foundation stiffness are not valid for elements deactivated (**EKILL**) and then reactivated (**EALIVE**). Surface printout does not include large strain effects.
- Nodal hydrodynamic pressure output is not available.
- For rezoning, only normal and tangential pressures applied on SURF154 are supported. (That is, rezoning support is available only for pressure on faces 1, 2 and 3.)

SURF154 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The only special features allowed are stress stiffening and large deflections.
- The VISC and BETD material properties are not applicable.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS Structural

- The only allowable material property is DENS.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

SURF156

3-D Structural Surface Line Load Effect

MP ME ST PR PRN DS <> <> PP EME MFS
Product Restrictions

SURF156 Element Description

SURF156 may be used for applying line pressure loads on structures. It may be overlaid onto the edge of any 3-D element. The element is applicable to 3-D structural analyses. Various loads and surface effects may exist simultaneously. See SURF156 - 3-D Structural Surface Line Load Effect in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 156.1: SURF156 Geometry

SURF156 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 156.1: SURF156 Geometry (p. 547). The element is defined by two to four nodes (KEYOPT(4) = 0 or 1). The orientation node lies in the element x-z plane and is required for orientation of the element loads. The element x-axis is parallel to the line connecting nodes I and J of the element.

See Nodal Loading (p. 49) for a description of element loads. Pressures may be input as surface loads on the element faces as shown by the circled numbers on Figure 156.2: Pressures (p. 548). SURF156 allows complex pressure loads. The input units are force per length.

Faces 1, 2, and 3 [KEYOPT(2) = 0] Positive values of pressure on the first three faces act in the positive element coordinate directions. For faces 2 and 3, the direction of the load is controlled by the element coordinate system which is oriented via the orientation node; therefore, the **ESYS** command has no effect. When using large deflection (**NLGEOM**,ON), the orientation of the loads may change based on the new location of the nodes. If the orientation node is on another element that moves, the orientation node will move with it. If the orientation node is not on another element, the node cannot move.

Faces 1, 2, and 3 [KEYOPT(2) = 1] Pressure loads are applied to the element faces according to the local coordinate system, as follows: face 1 in the x direction, face 2 in the local y direction, and face 3 in the local z direction. A local coordinate system must be defined, and the element must be set to that coordinate system via the **ESYS** command. When using large deflection (**NLGEOM**,ON), the orientation of the loads does not change.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 156.2: Pressures

Face 4 The magnitude of the pressure is P_I, and the direction is $(P_J i + P_K j + P_L k)/(P_J^2 + P_K^2 + P_L^2)^{1/2}$ where i, j, and k are unit vectors in the global Cartesian directions. No input values can be blank. When using the **SFFUN** or **SFGRAD** commands, the load direction is not altered but the load magnitude is the average of the computed corner node magnitudes. **SFCUM**,ADD should be used with caution, as this command also causes the load direction components to be added.

Face 5 The magnitude of the pressure is P_I, the load point is node I, and the direction is the element x-axis.

Face 6 The magnitude of the pressure is P_I, the load point is node J, and the direction is the element negative x-axis.

The effects of pressure load stiffness are automatically included for this element for real pressure on faces 2 and 3 if KEYOPT(2) = 0. If an unsymmetric matrix is needed for pressure load stiffness effects, issue a **NROPT**,UNSYM command.

KEYOPT(7) = 1 is useful when the element is used to represent a force. When KEYOPT(7) = 0, the force is input as a pressure times a unit length; however, if the length changes due to large deflections, the force also changes. When KEYOPT(7) = 1, the force remains unchanged even if the length changes.

A summary of the element input is given in "SURF156 Input Summary" (p. 548). A general description of element input is given in Element Input (p. 43).

SURF156 Input Summary

Nodes

I, J, K, L, if KEYOPT (4) = 0 and KEYOPT(5) = 0
I, J, K, if KEYOPT (4) = 1 and KEYOPT(5) = 0, or if KEYOPT(4) = 0 and KEYOPT(5) = 1
I, J, if KEYOPT(4) = 1 and KEYOPT(5) = 1

Degrees of Freedom
UX, UY, UZ

Real Constants
None

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Material Properties

None

Surface Loads

Pressures --

face 1 (parallel to x direction)
face 2 (parallel to y direction)
face 3 (parallel to z direction)
face 4 (oriented by input vector)
face 5 (parallel to x direction)
face 6 (parallel to x direction)

Body Loads

None

Special Features

Large deflection
Linear perturbation
Stress stiffening

KEYOPT(2)

Pressure applied to faces 1, 2, and 3 according to coordinate system:

0 --
Apply face loads in the element coordinate system

1 --
Apply face loads in the local coordinate system

KEYOPT(4)

Midside node:

0 --
Has a midside node

1 --
Does not have a midside node

KEYOPT(5)

Orientation node:

0 --
Has an orientation node

1 --
Does not have an orientation node. Use only for load on face 1 or face 4, or if KEYOPT(2) = 1.

KEYOPT(7)

Loaded area during large-deflection analyses:

0 --
Use new area

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
Use original area

SURF156 Output Data

The solution output associated with the element is in two forms:

- Nodal degree of freedom results included in the overall nodal solution
- Additional element output as shown in Table 156.1: SURF156 Element Output Definitions (p. 550)

A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 156.1: SURF156 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K	Y	Y
ORIENTATION NODE	Orientation node	Y	Y
PRESSURES	Pressures P1, P2, P3, P4 at nodes I, J	1	-
VECTOR DIRECTION	Direction vector of pressure P4	1	1

1. If pressure load

Table 156.2: SURF156 Item and Sequence Numbers (p. 551) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (**/POST1**) of the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 156.2: SURF156 Item and Sequence Numbers (p. 551):

Name
output quantity as defined in the Table 156.1: SURF156 Element Output Definitions (p. 550)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

I, J

sequence number for data at nodes I, J

Table 156.2: SURF156 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
P1 (real)	SMISC	-	1	2
P2 (real)	SMISC	-	3	4
P3 (real)	SMISC	-	5	6
P4 (real)	SMISC	7	-	-
P1 (imaginary)	SMISC	-	8	9
P2 (imaginary)	SMISC	-	10	11
P3 (imaginary)	SMISC	-	12	13
P4 (imaginary)	SMISC	14	-	-
P4 (real) VECTOR DIRECTION	NMISC	1 - 3	-	-
P4 (imaginary) VECTOR DIRECTION	NMISC	4 - 6	-	-

SURF156 Assumptions and Restrictions

- The element must not have a zero length, and the orientation node (when used) cannot be colinear with nodes I and J.

SURF156 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The only special features allowed are stress stiffening and large deflections.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

SHELL157

Thermal-Electric Shell

MP ME <> PR PRN <> <> <> PP EME MFS
Product Restrictions

SHELL157 Element Description

SHELL157 is a 3-D element having in-plane thermal and electrical conduction capability. The element has four nodes with two degrees of freedom, temperature and voltage, at each node. The element applies to a 3-D, steady-state or transient thermal analysis, although the element includes no transient electrical capacitance or inductance effects. The element requires an iterative solution to include the Joule heating effect in the thermal solution. See SHELL157 in the *Mechanical APDL Theory Reference* for more details about this element. If no electrical effects are present, the 3-D thermal shell (SHELL131) may be used.

If the model containing the thermal-electrical element is also to be analyzed structurally, replace the element with an equivalent structural element (such as SHELL181). If both in-plane and transverse thermal-electric conduction are needed, use a thermal-electric solid element (SOLID226 with KEYOPT(1) = 110).

Figure 157.1: SHELL157 Geometry

SHELL157 Input Data

The geometry, node locations, and coordinate systems for this element are shown in Figure 157.1: SHELL157 Geometry (p. 553). The element is defined by four nodes, four thicknesses, a material direction angle, and the orthotropic material properties.

The element may have variable thickness. The thickness is assumed to vary smoothly over the area of the element, with the thickness input at the four nodes. If the element has a constant thickness, you need to specify only TK(I) . If the thickness is not constant, you must specify all four thicknesses.

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems (p. 55). The element x-axis may be rotated by an angle THETA (in degrees). You can assign the specific heat and density any values for steady-state solutions. The electrical material property, RSV_, is the resistivity of the material. You can specify the

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

resistivity, like any other material property, as a function of temperature. Properties not specified default as described in Linear Material Properties in the *Material Reference*.

Specify the word VOLT for the *Lab* variable on the **D** command and the voltage input for the value. Specify the word AMPS for the *Lab* variable on the **F** command and the current into the node input for the value.

Element loads are described in Nodal Loading (p. 49). Convection or heat flux (but not both) and radiation may be specified as surface loads at the element faces as shown by the circled numbers on Figure 157.1: SHELL157 Geometry (p. 553). Because shell edge convection and flux loads are input on a per-unit-length basis, per-unit-area quantities must be multiplied by the shell thickness.

Heat generation rates may be specified as element body loads at the nodes. If the node I heat generation rate HG(I) is input, and all others are unspecified, they default to HG(I). This rate is in addition to the Joule heat generated by the current flow.

"SHELL157 Input Summary" (p. 554) summarizes the element input. A general description of element input appears in Element Input (p. 43).

SHELL157 Input Summary

Nodes

I, J, K, L

Degrees of Freedom

TEMP, VOLT

Real Constants

TK(I) - Shell thickness at node I
TK(J) - Shell thickness at node J; defaults to TK(I)
TK(K) - Shell thickness at node K; defaults to TK(I)
TK(L) - Shell thickness at node L; defaults to TK(I)
THETA - Element X-axis rotation

Material Properties

MP command: KXX, KYY, DENS, C, ENTH, RSVX, RSVY

Surface Loads

Convection or Heat Flux (but not both) and Radiation (using Lab = RDSF)--

face 1 (I-J-K-L) (bottom, -Z side), face 2 (I-J-K-L) (top, +Z side),
face 3 (J-I), face 4 (K-J), face 5 (L-K), face 6 (I-L)

Body Loads

Heat Generations --

HG(I), HG(J), HG(K), HG(L)

Special Features

Birth and death

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(2)

Evaluation of film coefficient:

0 --

Evaluate film coefficient (if any) at average film temperature, (TS + TB)/2

1 --

Evaluate at element surface temperature, TS

2 --

Evaluate at fluid bulk temperature, TB

3 --

Evaluate at differential temperature, |TS - TB|

SHELL157 Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures and voltages included in the overall nodal solution
- Additional element output as shown in Table 157.1: SHELL157 Element Output Definitions (p. 555)

Heat flowing out of the element is considered to be positive. The element output directions are parallel to the element coordinate system. The heat flow and the current flow into the nodes may be printed with the **OUTPR** command. The Joule heat generated this substep is used in the temperature distribution calculated for the next substep. A general description of solution output is given in Solution Output (p. 50). See The General Postprocessor (POST1) in the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 157.1: SHELL157 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
AREA	Convection face area	Y	Y
XC, YC, ZC	Location where results are reported	Y	2
HGEN	Heat generations HG(I), HG(J), HG(K), HG(L)	Y	-
TG:X, Y, SUM	Thermal gradient components and vector sum at centroid	Y	Y
TF:X, Y, SUM	Thermal flux (heat flow rate/cross-sectional area) components and vector sum at centroid	Y	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
EF:X, Y, SUM	Component electric fields and vector sum	Y	Y
JS:X, Y	Component current densities	Y	Y
JSSUM	Component current density vector sum	Y	-
JHEAT:	Joule heat generation per unit volume	Y	Y
FACE	Face label	1	1
AREA	Face area	1	1
NODES	Face nodes	1	1
HFILM	Film coefficient	1	1
TAVG	Average face temperature	1	1
TBULK	Fluid bulk temperature	1	-
HEAT RATE	Heat flow rate across face by convection	1	1
HFAVG	Average film coefficient of the face	-	1
TBAVG	Average face bulk temperature	-	1
HFLXAVG	Heat flow rate across face caused by input heat flux	-	1
HEAT RATE/AREA	Heat flow rate/area across face by convection	1	-
HEAT FLUX	Heat flux at each node of face	1	-

1. If a surface load is input

2. Available only at centroid as a ***GET** item.

Table 157.2: SHELL157 Item and Sequence Numbers (p. 556) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 157.2: SHELL157 Item and Sequence Numbers (p. 556):

Name
output quantity as defined in the Table 157.1: SHELL157 Element Output Definitions (p. 555)

Item
predetermined Item label for **ETABLE** command

Table 157.2: SHELL157 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input						
	Item	FACE 1 (BOT)	FACE 2 (TOP)	FACE 3 (J-I)	FACE 4 (K-J)	FACE 5 (L-K)	FACE 6 (I-L)
AREA	NMISC	1	7	13	19	25	31
HFAVG	NMISC	2	8	14	20	26	32
TAVG	NMISC	3	9	15	21	27	33
TBAVG	NMISC	4	10	16	22	28	34

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input						
	Item	FACE 1 (BOT)	FACE 2 (TOP)	FACE 3 (J-I)	FACE 4 (K-J)	FACE 5 (L-K)	FACE 6 (I-L)
HEAT RATE	NMISC	5	11	17	23	29	35
HFLXAVG	NMISC	6	12	18	24	30	36

SHELL157 Assumptions and Restrictions

- The element requires an iterative solution for electrical-thermal coupling.
- Zero area elements are not allowed. This occurs most frequently when the elements are not numbered properly. The element must not taper down to a zero thickness at any corner. A triangular element may be formed by defining duplicate K and L node numbers as described in Degenerated Shape Elements (p. 39). The specific heat and enthalpy are evaluated at each integration point to allow for abrupt changes (such as for melting) within a coarse grid. If a current is specified at the same node that a voltage is specified, the current is ignored. The electrical and the thermal solutions are coupled through an iterative procedure.
- No conversion is included between electrical heat units and mechanical heat units. The resistivity may be divided by a conversion factor, such as 3.415 BTU/Hr per watt, to get Joule heat in mechanical units. Current (input and output) should also be converted for consistent units.
- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, the elements must have the same reaction force (see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*).
- This element cannot be used in a distributed solution.

SHELL157 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The birth and death special feature is not allowed.

ANSYS Emag

- This element has only electric field capability, and does not have thermal capability.
- The element may only be used in a steady-state electric analysis.
- The only valid degree of freedom is VOLT.
- The only allowable material properties are RSVX and RSVY.
- No surface loads or body loads are applicable.
- The birth and death special feature is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

SURF159

General Axisymmetric Surface

MP ME ST <> <> <> <> <> PP EME MFS
Product Restrictions

SURF159 Element Description

Use SURF159 to model axisymmetric solid surface loads acting on general axisymmetric solid elements (SOLID272 or SOLID273). The element has linear or quadratic displacement behavior on the master plane and is well suited to modeling irregular meshes on the master plane. It is defined by two or three nodes on the master plane, and nodes created automatically in the circumferential direction based on the master plane nodes. The total number of nodes depends on the number of nodal planes (KEYOPT(2)). The element area between nodal planes is called a *facet*. Each node has three degrees of freedom: translations in the nodal x, y and z directions. Various loads and surface effects can exist simultaneously.

In addition to the SURF159 element information provided here, the following topics are available in the *Mechanical APDL Theory Reference*:

- SURF159 - General Axisymmetric Surface with 2 or 3 Nodes
- General Axisymmetric Surfaces

For related information (concerning the elements used with SURF159), see General Axisymmetric Elements (p. 76) in this document.

Figure 159.1: SURF159 Geometry (KEYOPT(2) = 3)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SURF159 Input Data

The geometry and node locations for this element (when KEYOPT(2) = 3) are shown in Figure 159.1: SURF159 Geometry (KEYOPT(2) = 3) (p. 559). The element input data includes nodes, real constants, and the material properties. The total number of nodes is the two or three base nodes times the number of nodal planes. (For information about how Fourier nodes are generated, see the **NAXIS** command documentation.) SURF159 elements can be generated via the **ESURF** command *before* generating nodes for general axisymmetric element sections (**NAXIS**).

The element has the x-axis as normal to the element surface, the y-axis in the meridional direction, and the z-axis in the circumferential direction.

The mass and volume calculations use the element thicknesses (real constants TKI, and TKJ). Thickness TKJ defaults to TKI. The mass calculation uses the density (material property DENS, mass per unit volume) and the real constant ADMSUA, the added mass per unit area.

The stress stiffness matrix and load vector calculations use the in-plane force per unit length (input as real constant SURT) and the elastic foundation stiffness (input as real constant EFS); the EFS uses pressure-per-length (or force-per-length-cubed) units. The foundation stiffness can be damped, either by using the material property BETD as a multiplier on the stiffness or by directly using the material property VISC.

"SURF159 Input Summary" (p. 562) contains a summary of the element input. See Nodal Loading (p. 49) for a general description of element input.

Pressure Loads

Pressures must be input as element surface loads (as force-per-length squared) on the element faces as shown by the circled numbers in Figure 159.2: SURF159 Pressures (p. 561). For SURF159, applying surface loads to the element (**SFE**) or nodes (**SF**) is very different in comparison to the SOLID272 and SOLID273 elements. An **SF** command, if issued, applies loads to faces 1 through K only (where K = KEYOPT(2), the number of nodal planes).

SURF159 allows complex loads.

Defaults for Pressures on Faces 1 through 3K

If the first pressure for a face is specified (**SFE**) and the other three remain blank, the other three pressures default to the first.

If values are specified (and perhaps supplemented by the aforementioned default) to the first face of each of the three face families (1 through K, K+1 through 2K, or 2K+1 to 3K), the same values are assigned to *all other faces* around the circumference. (The **SFELIST** command does not reflect this defaulting behavior.)

Example 1 To obtain a uniform pressure across the element and around the circumference, only one specified pressure is necessary on the first face of the face family.

Example 2 To obtain a pressure that tapers in the radial direction and is constant around the circumference, all four values must be specified on the first face of the face family.

Faces 1 through 3K [KEYOPT(3) = 0] Positive values of pressure on these faces act in the positive element coordinate directions, as follows: faces 1 through K in the element x (normal) direction, faces

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

K+1 through 2K in the element y (tangent, meridional) direction, and faces 2K+1 through 3K in the element z (tangent, circumferential) direction. For faces 1 through K, positive or negative values may be removed as requested via KEYOPT(6) to simulate the discontinuity at the free surface of a contained fluid. For faces K+1 through 3K, the direction of the load is controlled by the element coordinate system; therefore, the **ESYS** command may be needed.

Faces 1 through 3K [KEYOPT(3) = 1] Pressure loads are applied to the element faces according to the local coordinate system, as follows: faces 1 through K in the local x direction, faces K+1 through 2K in the local y direction, and faces 2K+1 through 3K in the local z direction. A local coordinate system must be defined, and the element must be set to that coordinate system via the **ESYS** command. KEYOPT(6) does not apply.

Figure 159.2: SURF159 Pressures

where K = KEYOPT (2)

Face 3K+1 The direction is normal to the element and the magnitude of the pressure at each integration point is $P_I + XP_J + YP_K + ZP_L$, where P_I through P_L are input as *VAL1* through *VAL4* on the **SFE** command, and X, Y, Z are the global Cartesian coordinates at the current location of the integration point. No input values can be blank. Positive or negative values may be removed as requested with KEYOPT(6) to simulate the discontinuity at the free surface of a contained fluid. The **SFFUN** and **SFGRAD** commands do not work with face 3K+1.

Face 3K+2 The magnitude of the pressure is P_I, and the direction is $(P_J i + P_K j + P_L k)/(P_J^2 + P_K^2 + P_L^2)^{1/2}$ where i, j, and k are unit vectors in the global Cartesian directions. The load magnitude may be adjusted with KEYOPT(11) and KEYOPT(12). No input values can be blank.

Temperatures may be input as element body loads at the nodes. For the nodes on the master plane, the node I_1 temperature $T(I_1)$ defaults to TUNIF. If all other temperatures are unspecified, they default to $T(I_1)$. If both corner node temperatures are specified, the midside node temperature defaults to the average temperature of the adjacent corner nodes. For any other input pattern, unspecified temperatures default to TUNIF. For the nodes generated in the circumferential direction based on the master node, if all other temperatures are unspecified, they default to the value of their base nodes ($T(I_1)$, $T(J_1)$, and $T(K_1)$, depending on their location). For any other input pattern, unspecified temperatures default to TUNIF.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, issue a **NROPT**,UNSYM command. For a geometric nonlinear analysis when convergence is an issue, use an unsymmetric matrix.

SURF159 Input Summary

Nodes

I_1, J_1, K_1, I_2, J_2, K_2, . . . , I_n, J_n, K_n (where n = KEYOPT(2), the number of nodal planes)

Degrees of Freedom

UX, UY, UZ

Real Constants

(Blank), (Blank), (Blank), EFS, SURT, ADMSUA,
TKI, TKJ

Material Properties

MP command: DENS, VISC, ALPD, BETD

Surface Loads

Pressures --

In the following table, K represents the number of nodal planes (specified via KEYOPT(2)):

Pressure Face	Nodes	Pressure Load Type	
		KEYOPT(3) = 0	KEYOPT(3) = 1
face 1	J_1, I_1, I_2, J_2	Normal	x direction per **ESYS**
face 2	J_2, I_2, I_3, J_3		
. . .	. . .		
face K	J_K, I_K, I_1, J_1		
face K+1	J_1, I_1, I_2, J_2	Meridional tangent	y direction per **ESYS**
face K+2	J_2, I_2, I_3, J_3		
. . .	. . .		
face 2K	J_K, I_K, I_1, J_1		
face 2K+1	J_1, I_1, I_2, J_2	Circumferential tangent	z direction per **ESYS**
face 2K+2	J_2, I_2, I_3, J_3		
. . .	. . .		
face 3K	J_K, I_K, I_1, J_1		
face 3K+1	All	Global taper	
face 3K+2	All	Oriented vector	

Body Loads

Temperatures --

$T(I_1)$, $T(J_1)$, $T(K_1)$, $T(I_2)$, $T(J_2)$, $T(K_2)$, . . . , $T(I_n)$, $T(J_n)$, $T(K_n)$

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Special Features

Birth and death
Large deflection
Stress stiffening

KEYOPT(2)

Number of Fourier nodes in the circumferential direction (that is, the number of nodal planes):

1 --
Axisymmetric deformation (with or without torsion).

3 - 12 --
General 3-D deformation.

This KEYOPT has no default. You must specify a valid value. (0 is not valid.) The value must match the KEYOPT(2) value of the underlying SOLID272 or SOLID273 element.

For information about specifying the number Fourier nodes, see General Axisymmetric Elements in this document.

For information about how Fourier nodes are generated, see the **NAXIS** command documentation.

KEYOPT(3)

Pressure applied to faces 1 through 3K (3 x KEYOPT(2)), according to the coordinate system:

0 --
Apply face loads in the default element coordinate system (for example, the first K faces act normal to the surface).

1 --
Apply face loads in the specified element coordinate system (for example, the first K faces act in the x direction as defined via the **ESYS** command).

KEYOPT(4)

Midside nodes:

0 --
Has midside nodes. Use with SOLID273. This value is the default.

1 --
No midside nodes. Use with SOLID272.

KEYOPT(6)

Applicable only to normal direction pressure (faces 1 through K and 3K+1):

0 --
Use pressures as calculated (positive and negative).

1 --
Use positive pressures only (negative set to zero).

2 --
Use negative pressures only (positive set to zero).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To use KEYOPT(6), KEYOPT(3) must be set to 0.

KEYOPT(7)
Loaded area during large-deflection analyses:

0 --
Use new area.

1 --
Use original area.

KEYOPT(11)
Pressure applied by vector orientation (face 3K+2):

0 --
On projected area and includes tangential component.

1 --
On projected area and does not include tangential component.

2 --
On full area and includes the tangential component.

KEYOPT(12)
Effect of the direction of the element normal (element x-axis) on vector oriented (face 3K+2) pressure:

0 --
Pressure load is applied regardless of the element normal orientation.

1 --
Pressure load is not used if the element normal is oriented in the same general direction as the pressure vector.

Table 159.1: SURF159 Real Constants

No.	Name	Description
1 ... 3	(Blank)	--
4	EFS	Foundation stiffness
5	SURT	Surface tension
6	ADMSUA	Added mass/unit area
7	TKI	In-plane thickness at node I
8	TKJ	In-plane thickness at node J (defaults to TKI)

SURF159 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 159.2: SURF159 Element Output Definitions (p. 565)

A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To view 3-D mode shapes for a modal or eigenvalue buckling analysis, expand the modes with element results calculation active (via the **MXPAND** command's `Elcalc` = YES option).

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 159.2: SURF159 Element Output Definitions

Name	Definition	O	R
ELEMID	Element number	Y	Y
NODES	Nodes - I, J, . . . , K (nodes in master plane)	Y	Y
NNP	Number of nodal planes	Y	Y
AREA	Total area	Y	Y
VOLU	Total volume	Y	Y
MAT	Material number	Y	Y
DENS	Density	Y	Y
ADMSUA	Added mass per unit area	Y	Y
MASS	Element mass	Y	Y
EFS	Elastic foundation stiffness	Y	Y
SURT	Surface tension	Y	Y
PRES	Pressures	Y	Y

Table 159.3: SURF159 Item and Sequence Numbers (p. 565) lists output available through **ETABLE** using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this document for more information. The following notation is used in the output table:

Name
output quantity as defined in the Table 159.2: SURF159 Element Output Definitions (p. 565)

Item
predetermined Item label for **ETABLE**

I_1, J_1, I_2, J_2, . . .
sequence number for data at nodes I_1, J_1, I_2, J_2, . . .

Table 159.3: SURF159 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input (K = KEYOPT(2))										
	Item	E	I_1	J_1	I_2	J_2	I_3	J_3	...	I_K	J_K
P_1	SMISC	-	2	1	3	4	-	-	...	-	-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input (K = KEYOPT(2))										
	Item	E	I_1	J_1	I_2	J_2	I_3	J_3	...	I_K	J_K
P_2	SMISC	-	-	-	6	5	7	8	...	-	-
...	...	...	...	...	...	...	...	...	...	...	...
P_K	SMISC	-	4K-1	4K	-	-	-	-	...	4K-2	4K-3
P_{K+1}	SMISC	-	4K+2	4K+1	4K+3	4K+4	-	-	...	-	-
P_{K+2}	SMISC	-	-	-	4K+6	4K+5	4K+7	4K+8	...	-	-
...	...	...	...	...	...	...	...	...	...	...	...
P_{2K}	SMISC	-	8K-1	8K	-	-	-	-	...	8K-2	8K-3
P_{2K+1}	SMISC	-	8K+2	8K+1	8K+3	8K+4	-	-	...	-	-
P_{2K+2}	SMISC	-	-	-	8K+6	8K+5	8K+7	8K+8	...	-	-
...	...	...	...	...	...	...	...	...	...	...	...
P_{3K}	SMISC	-	12K-1	12K	-	-	-	-	...	12K-2	12K-3
P_{3K+1} (magnitude)	SMISC	12K+1	-	-	-	-	-	-	...	-	-
X gradient	SMISC	12K+2	-	-	-	-	-	-	...	-	-
Y gradient	SMISC	12K+3	-	-	-	-	-	-	...	-	-
Z gradient	SMISC	12K+4	-	-	-	-	-	-	...	-	-
P_{3K+2} (magnitude)	SMISC	12K+5	-	-	-	-	-	-	...	-	-
X component	SMISC	12K+6	-	-	-	-	-	-	...	-	-
Y component	SMISC	12K+7	-	-	-	-	-	-	...	-	-
Z component	SMISC	12K+8	-	-	-	-	-	-	...	-	-

The item and sequence numbers shown are for real pressures. Imaginary pressures are represented in the same way but have 12K+8 added to them.

SURF159 Assumptions and Restrictions

- The length of the base element must be nonzero.
- The base element must lie on one side of the axisymmetric axis, and the axisymmetric axis must be on the same plane as the base element (master plane).
- A base element or base node must be associated with one axisymmetric axis (defined via **SECTYPE** and **SECDATA** commands) *before* generating nodes for general axisymmetric element sections (**NAXIS**) or defining an element by node connectivity (**E**).
- An edge with a removed midside node implies that the displacement varies linearly, rather than parabolically, along that edge. For more information about the use of midside nodes, see Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide*.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You cannot apply a pressure load by applying a surface load to selected areas (**SFA**).
- The contribution of the element to the mass moment inertia of the whole model is calculated by element mass multiplied by the square of the coordinates of the elemental centroid. The moment of inertia may therefore be inaccurate.
- The element does not support the expansion pass of a superelement with large rotation.
- Issuing an **/ESHAPE**,1 command while PowerGraphics is active causes the program to plot the elements in 3-D and the results on both nodal planes and all integration planes in the circumferential direction; otherwise, the program plots the elements in 2-D and the results on the master plane.
- You cannot display surface load symbols (**/PSF**) for this element.
- When listing the surface loads for elements (**SFELIST**), only information for the *first* facet is returned.
- When plotting contour values via **/ESHAPE**,1, expanded SURF159 elements are assigned zeroes; you should therefore deselect SURF159 elements when plotting results such as stresses and strains.
- In postprocessing, print commands return the nodal plane results only.
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). Prestress effects can be activated via the **PSTRES** command.

SURF159 Product Restrictions

No product-specific restrictions exist for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

LINK160

Explicit 3-D Spar (or Truss)

<> <> <> <> <> <> <>DY <> <> <>
Product Restrictions

LINK160 Element Description

LINK160 has three degrees of freedom at each node and carries an axial force.

This element is used in explicit dynamic analyses only. Refer to the *LS-DYNA Theoretical Manual* for more information.

Figure 160.1: LINK160 Geometry

LINK160 Input Data

The geometry and node locations are shown in Figure 160.1: LINK160 Geometry (p. 569). Node K determines the initial orientation of the cross section. For this element, you can choose three materials: isotropic elastic, plastic kinematic, and bilinear kinematic.

The element is defined by nodes I and J in the global coordinate system. Node K defines a plane (with I and J) containing the element s-axis. The element r-axis runs parallel to the length of the element and through nodes I and J. Node K is always required to define the element axis system and it must not be colinear with nodes I and J. The location of node K is used only to *initially* orient the element.

Use the **EDLOAD** command to apply nodal loads (displacements, forces, etc.). Also use **EDLOAD** to apply loads on rigid bodies. For more information on how to apply loads in an explicit dynamic analysis, see Loading in the *ANSYS LS-DYNA User's Guide*.

A summary of the element input is given in "LINK160 Input Summary" (p. 570). A general description of element input is given in Element Input (p. 43).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LINK160 Input Summary

Nodes

I, J, K (K is the orientation node)

Degrees of Freedom

UX, UY, UZ, VX, VY, VZ, AX, AY, AZ

For explicit dynamics analyses, V (X, Y, Z) refers to nodal velocity, and A (X, Y, Z) refers to nodal acceleration. Although V (X, Y, Z) and A (X, Y, Z) appear as degrees of freedom, they are not actually physical degrees of freedom; however, these quantities are computed as degree-of-freedom solutions and stored for postprocessing.

Real Constants

Area - Cross-sectional area

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, NUXY, DENS, ALPD, BETD
EDMP command: RIGID

Surface Loads

None

Body Loads

None

Special Features

All nonlinear features allowed for an explicit dynamic analysis.

KEYOPTS

None

LINK160 Output Data

Output data for LINK160 consists of the following: Axial force

To output the data, you must use the **ETABLE** command. For the ITEM label, specify SMISC. For the COMP label, specify 1 for axial force. Then, you can use the **PRETAB** command to print the output data.

LINK160 Assumptions and Restrictions

- The spar element assumes a straight bar, axially loaded at its ends with uniform properties from end to end.
- The length of the spar must be greater than zero, so nodes I and J must not be coincident.
- The cross-sectional area must be greater than zero.
- The displacement shape function implies a uniform stress in the spar.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LINK160 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

BEAM161

Explicit 3-D Beam

<> <> <> <> <> <> <>DY <> <> <>
Product Restrictions

BEAM161 Element Description

BEAM161 has several characteristics:

- It is incrementally objective (rigid body rotations do not generate strains), allowing for the treatment of finite strains that occur in many practical applications.
- It is simple for computational efficiency and robustness.
- It is compatible with the brick elements.
- It includes finite transverse shear strains. However, the added computations needed to retain this strain component, compared to those for the assumption of no transverse shear strain, are significant.

The Belytschko beam element formulation (KEYOPT(1) = 2, 4, 5) is part of a family of structural finite elements that use a "co-rotational technique" for treating large rotation.

This element is used in explicit dynamic analyses only. Refer to the *LS-DYNA Theoretical Manual* for more information.

Figure 161.1: BEAM161 Geometry

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BEAM161 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 161.1: BEAM161 Geometry (p. 573). Node K determines the initial orientation of the cross section.

The element is defined by nodes I and J in the global coordinate system. Node K defines a plane (with I and J) containing the element s-axis. The element r-axis runs parallel to the centroidal line of the element and through nodes I and J. Node K is always required to define the element axis system and it must not be colinear with nodes I and J. The location of node K is used only to *initially* orient the element. (For information about orientation nodes and beam meshing, see Meshing Your Solid Model in the *Modeling and Meshing Guide*.)

Use the **EDLOAD** command to apply nodal loads and other load types described below. For detailed information on how to apply loads in an explicit dynamic analysis, see the *ANSYS LS-DYNA User's Guide*.

Pressures can be input as surface loads on the element faces as shown by the circled numbers in Figure 161.1: BEAM161 Geometry (p. 573). Note, however, that pressure *is actually a traction load applied to the center line of the element*. Use the **EDLOAD** command to apply the pressure load, and input the pressure as a force per unit length value. Positive normal pressures act into the element.

Base accelerations and angular velocities in the x, y, and z directions can be applied at the nodes using the **EDLOAD** command. To apply these loads, you need to first select the nodes and create a component. The load is then applied to that component.

You can also use the **EDLOAD** command to apply loads (displacements, forces, etc.) on rigid bodies.

You can choose from the following materials when working with BEAM161, with the restrictions as noted:

- Isotropic Elastic
- Bilinear Kinematic (Except KEYOPT(1) = 2)
- Plastic Kinematic (Except KEYOPT(1) = 2)
- Viscoelastic (KEYOPT(1) = 1 only)
- Power Law Plasticity (KEYOPT(1) = 1 only)
- Piecewise Linear Plasticity (KEYOPT(1) = 1 only)

KEYOPT(1) allows you to specify one of four element formulations for BEAM161 (see "BEAM161 Input Summary" (p. 577)). For details of real constants to be specified for each element formulation, see Table 161.1: BEAM161 Real Constants (p. 578).

KEYOPT(2) is valid only with rectangular element formulations (KEYOPT(1) = 0, 1, 4).

The following illustrations show the valid standard beam cross sections when KEYOPT(4)>0, and KEYOPT(5) = 2 (standard beam cross section).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 161.2: Standard Beam Cross Sections

w = flange width
t_f = flange thickness
d = depth
t_w = web thickness
s_{ref} = location of reference surface normal to s, Hughes-Liu beam only
t_{ref} = location of reference surface normal to t, Hughes-Liu beam only

Type 1: W-section

Type 2: C-section

Type 3: Angle section

Type 4: T-section

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 161.3: Standard Beam Cross Sections (continued)

Type 5: Rectangular tubing

Type 6: Z-section

Type 7: Trapezoidal section

KEYOPT(5) is not valid when KEYOPT(1) = 2.

A summary of the element input is given in "BEAM161 Input Summary" (p. 577). Additional information about real constants for this element is provided in Table 161.1: BEAM161 Real Constants (p. 578). For more information about this element, see the *ANSYS LS-DYNA User's Guide*.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BEAM161 Input Summary

Nodes

I, J, K (K is the orientation node)

Degrees of Freedom

UX, UY, UZ, VX, VY, VZ, AX, AY, AZ, ROTX, ROTY, ROTZ

Note

For explicit dynamics analyses, V(X, Y, Z) refers to nodal velocity, and A(X, Y, Z) refers to nodal acceleration. Although V(X, Y, Z) and A(X, Y, Z) appear as DOFs, they are not actually physical DOFs. However, these quantities are computed as DOF solutions and stored for postprocessing.

Real Constants

See Table 161.1: BEAM161 Real Constants (p. 578) for a description of the real constants.

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, NUXY, DENS, ALPD, BETD
EDMP command: RIGID (KEYOPT(1) = 1,2)

Surface Loads

Pressure --
face 1 (I-J) (+r tangential direction), face 2 (I-J) (-s normal direction), face 3 (I) (-t normal direction)

Body Loads

None

Special Features

All nonlinear features allowed for an explicit dynamic analysis.

KEYOPT(1)

Element formulation:

0, 1 --
Hughes-Liu with cross section integration (default)

2 --
Belytschko-Schwer resultant beam (resultant)

4 --
Belytschko-Schwer full cross section integration

5 --
Belytschko-Schwer circular beam with cross section integration

KEYOPT(2)

Quadrature rule:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
One integration point

0, 2 --
2 x 2 Gauss quadrature (default)

3 --
3 x 3 Gauss quadrature

4 --
3 x 3 Lobatto quadrature

5 --
4 x 4 Gauss quadrature

Note

KEYOPT(2) is valid only with rectangular element formulations (KEYOPT(1) = 0, 1, 4).

KEYOPT(4)
Integration rule for section:

0 --
Standard integration option

n --
User-defined integration rule ID (valid range: 1 to 9999)

KEYOPT(5)
Cross section type:

0 --
Rectangular cross section

1 --
Circular cross section

2 --
Arbitrary cross section (user defined integration rule) or standard beam cross section, if KEYOPT (4) > 0.

Table 161.1: BEAM161 Real Constants

No.	Name	Description	Use if...
1	SHRF	Shear factor. Default = 1.0 Recommended for rectangular sections = 5/6.	KEYOPT (1) = 0,1, 4, or 5
2	TS1	Beam thickness in s direction at node 1;	KEYOPT (1) = 0, 1, or 4 KEYOPT (5) = 0 or 2

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

No.	Name	Description	Use if...
		if KEYOPT (5) = 2, then use for arbitrary cross section only.	
3	TS2	Beam thickness in s direction at node 2; if KEYOPT (5) = 2, then use for arbitrary cross section only.	KEYOPT (1) = 0, 1, or 4 KEYOPT (5) = 0 or 2
4	TT1	Beam thickness in t direction at node 1; if KEYOPT (5) = 2, then use for arbitrary cross section only.	KEYOPT (1) = 0, 1, or 4 KEYOPT (5) = 0 or 2
5	TT2	Beam thickness in t direction at node 2; if KEYOPT (5) = 2, then use for arbitrary cross section only.	KEYOPT (1) = 0, 1, or 4 KEYOPT (5) = 0 or 2
2	DS1	Beam outer diameter at node 1[1]	KEYOPT (1) = 0, 1, or 5 KEYOPT (4) = 0 KEYOPT (5) = 1
3	DS2	Beam outer diameter at node 2[1]	KEYOPT (1) = 0, 1, or 5 KEYOPT (4) = 0 KEYOPT (5) = 1
4	DT1	Beam inner diameter at node 1[1]	KEYOPT (1) = 0, 1, or 5 KEYOPT (4) = 0 KEYOPT (5) = 1
5	DT2	Beam inner diameter at node 2[1]	KEYOPT (1) = 0, 1, or 5 KEYOPT (4) = 0 KEYOPT (5) = 1
6	NSLOC	Location of reference surface normal to s-axis = 1 side at s = 1 = 0 center = -1 side at s = -1	KEYOPT (1) = 0, 1, 4, or 5 KEYOPT (4) = 0
7	NTLOC	Location of reference surface normal to t-axis = 1 side at t = 1	KEYOPT (1) = 0, 1, 4, or 5 KEYOPT (4) = 0

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

No.	Name	Description	Use if...
		= 0 center = -1 side at t = -1	
8	A	Cross sectional area See Table 161.2: Properties of Beam Cross Sections for Several Common Cross Sections (p. 582)	KEYOPT (4) = 0 KEYOPT (1) = 2
9	ISS	Moment of inertia about s-axis See Table 161.2: Properties of Beam Cross Sections for Several Common Cross Sections (p. 582)	KEYOPT (4) = 0 KEYOPT (1) = 2
10	ITT	Moment of inertia about t-axis See Table 161.2: Properties of Beam Cross Sections for Several Common Cross Sections (p. 582)	KEYOPT (4) = 0 KEYOPT (1) = 2
11	IRR	Polar moment of inertia See Table 161.2: Properties of Beam Cross Sections for Several Common Cross Sections (p. 582)	KEYOPT (4) = 0 KEYOPT (1) = 2
12	SA	Shear area See Table 161.2: Properties of Beam Cross Sections for Several Common Cross Sections (p. 582)	KEYOPT (4) = 0 KEYOPT (1) = 2
13	NIP	Number of integration points See Figure 161.5: Definition of Integration Points for User Defined Integration Rule (p. 584)	KEYOPT (4) > 0 and KEYOPT (5) = 2
14	RA	Relative area of cross section; that is, the actual cross-sectional area divided by the area defined by the product of the specified thickness in the s direction and the thickness in the t direction. See Figure 161.4: Definition of Relative Area for User Defined Integration Rule (p. 584).	KEYOPT (4) > 0 and KEYOPT (5) = 2

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

No.	Name	Description	Use if...
15	ICST	Standard cross section type. **Note** If this type is nonzero, then NIP and RA should be zero. Cross section types are: 1 - W-section 2 - C-section 3 - Angle section 4 - T-section 5 - Rectangular tubing 6 - Z-section 7 - Trapezoidal section See Figure 161.2: Standard Beam Cross Sections (p. 575), Figure 161.3: Standard Beam Cross Sections (continued) (p. 576).	KEYOPT (4) > 0[2] and KEYOPT (5) = 2 (standard cross section only)
16	W	Flange width	ICST > 0, and NIP = RA = 0
17	TF	Flange thickness	ICST > 0, and NIP = RA = 0
18	D	Depth	ICST > 0, and NIP = RA = 0
19	TW	Web thickness	ICST > 0, and NIP = RA = 0
20	SREF	Location of reference surface normal to s **Note** If KEYOPT (1) = 1 only	ICST > 0, and NIP = RA = 0
21	TREF	Location of reference surface normal to t **Note** If KEYOPT (1) = 1 only	ICST > 0, and NIP = RA = 0
22, 25, 28, ...79	S(i)	s coordinate of integration point	KEYOPT (4) > 0

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

No.	Name	Description	Use if...
		i = 1, NIP (NIP = 20 max)[3]	KEYOPT (5) = 2, arbitrary cross section only NIP > 0, RA > 0, ICST = 0
23, 26, 29, ...80	T(i)	t coordinate of integration point i = 1, NIP (NIP = 20 max)[3]	KEYOPT (4) > 0 KEYOPT (5) = 2, arbitrary cross section only NIP > 0, RA > 0, ICST = 0
24, 27, 30, ...81	WF(i)	Weighting factor; that is, the area associated with the integration point divided by the actual cross-section area. i = 1, NIP (NIP = 20 max)[3] See Table 161.2: Properties of Beam Cross Sections for Several Common Cross Sections (p. 582)	KEYOPT (4) > 0 KEYOPT (5) = 2, arbitrary cross section only NIP > 0, RA > 0, ICST = 0

1. DS1, DS2, DT1, and DT2 are used only if KEYOPT (5) = 1. If KEYOPT (5) = 0 or 2, then use TS1, TS2, TT1, and TT2.

2. For KEYOPT (5) = 2, standard cross-section type, the integration point ID (KEYOPT (4) > 0) is not used since NIP = RA = 0. However, you must provide this input in any case.

3. Specify S(i), T(i), and WF(i) for each integration point. For example, for 20 integration points, specify S(1), T(1), WF(1), S(2), T(2), WF(2), ... S(20), T(20), WF(20).

Table 161.2: Properties of Beam Cross Sections for Several Common Cross Sections

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$I_{tt} = I_{ss} \cong \pi r^3 h$$

$$J \cong 2\pi r^3 h$$

$$f_{tt} = f_{ss} = 2$$

$$A \cong 2\pi r h$$

$$I_{tt} = \frac{bh^3}{12}$$

$$I_{ss} = \frac{b^3 h}{12}$$

$$J \cong \left[\frac{1}{3} - 0.21\left(\frac{b}{h}\right)\left(1 - \frac{b^4}{12h^4}\right)\right] hb^3$$

$$f_{tt} = f_{ss} = \frac{6}{5}$$

$$A = bh$$

$$I_{tt} \cong \left(\frac{h^2}{6}\right)(ht_w + 3bt_f)$$

$$I_{ss} \cong \left(\frac{b^2}{6}\right)(bt_f + 3ht_w)$$

$$J \cong \frac{2b^2h^2(t_w t_f)}{(bt_w + ht_f)}$$

$$f_{tt} = \frac{A}{[2(b + t_w)t_f]}$$

$$f_{ss} = \frac{A}{[2(h + t_f)t_w]}$$

$$A = 2(bt_f + ht_w)$$

$$\text{Shear area} = \frac{A}{f} = \mu A$$

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 161.4: Definition of Relative Area for User Defined Integration Rule

$$\text{Relative area} = \frac{A}{S_t\, t_t}$$

t_t = input as TT1 or TT2 on R command
S_t = input as TS1 or TS2 on R command

Figure 161.5: Definition of Integration Points for User Defined Integration Rule

BEAM161 Output Data

To store output data for this element, you first need to specify the number of integration points for which you want output data. Use the **EDINT**,,*BEAMIP* command during the solution phase of your analysis to specify the number of integration points. By default, output is written for 4 integration points. For the resultant beam formulation (KEYOPT(1) = 2), there is no stress output (regardless of the *BEAMIP*

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

setting). If you set *BEAMIP* = 0, no stress output is written for any of the beam elements. In this case, the beams will not appear in any POST1 plots because the program assumes they are failed elements.

To display the data for BEAM161, you must use the **ETABLE** command. Then, you can use the **PRETAB** command to print the output data. The **RSYS** command has no effect when postprocessing output for this element.

The following items are available on the results file.

Table 161.3: BEAM161 Element Output Definitions

Name	Definition
S (r, rs, rt)	Stresses
EPEQ	Equivalent plastic strain
EPTO	Axial strain
MFORr	Member force in the element coordinate system, r direction
N (s, t)	Out-of-plane (s, t) shear
M (s, t)	Element (s, t) moments
TORQ	Torsional resultant

For each of these output data, one set of values, given at the centroid, is output for the entire beam.

Table 161.4: BEAM161 Item and Sequence Numbers (p. 585) lists output available through the **ETABLE** and **ESOL** commands using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 161.4: BEAM161 Item and Sequence Numbers (p. 585):

Name
output quantity as defined in the Table 161.3: BEAM161 Element Output Definitions (p. 585)

Item
predetermined Item label for **ETABLE** or **ESOL** command

E
sequence number for single-valued or constant element data

1st IP
sequence number for the first integration point

***n*th IP**
sequence number for the *n*th integration point as defined by the **EDINT** command.

Table 161.4: BEAM161 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	1st IP	*n*th IP
MFORr	SMISC	1	-	-
Ns	SMISC	2	-	-
Nt	SMISC	3	-	-
Ms	SMISC	4	-	-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	1st IP	*n*th IP
Mt	SMISC	5	-	-
TORQ	SMISC	6	-	-
Sr	LS	-	1	5 x (*n*-1) +1
Srs	LS	-	2	5 x (*n*-1) +2
Srt	LS	-	3	5 x (*n*-1) +3
EPEQ	LS	-	4	5 x (*n*-1) +4
EPTO	LS	-	5	5 x (*n*-1) +5

1. In this table, *n* refers to the current integration point for which you want output data.

BEAM161 Assumptions and Restrictions

- The beam must not have a zero length.
- The beam can have any open or single-cell closed cross-sectional shape for which the area and moments of inertia are nonzero.
- Warping torsion is assumed negligible and the warping moment of inertia is not used in the stiffness computation.
- Warping of the cross section is unconstrained and is the same for all cross-sections; therefore, the torsional rotation of the cross-section is assumed to vary linearly along the length. However, warping is not applicable to the resultant beam formulation (KEYOPT(1) = 2).

BEAM161 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PLANE162

Explicit 2-D Structural Solid

<> <> <> <> <> <> <>DY <> <> <>
Product Restrictions

PLANE162 Element Description

PLANE162 is used for modeling 2-D solid structures in ANSYS LS-DYNA. The element can be used either as a planer or as an axisymmetric element. The element is defined by four nodes having six degrees of freedom at each node: translations, velocities, and accelerations in the nodal x and y directions. A three-node triangle option is also available, but not recommended.

The element is used in explicit dynamic analyses only. When using this element, the model must only contain PLANE162 elements - you cannot mix 2-D and 3-D explicit elements in the same model. Furthermore, all PLANE162 elements in the model must be the same type (plane stress, plane strain, or axisymmetric). Refer to the *LS-DYNA Theoretical Manual* for more information.

Figure 162.1: PLANE162 Geometry

PLANE162 Input Data

The geometry, node locations, and coordinate system for this element are shown in Figure 162.1: PLANE162 Geometry (p. 587). Use KEYOPT(3) to specify whether the element is a plane stress, plane strain, or axisymmetric element. For the axisymmetric option (KEYOPT(3) = 1), you may also use KEYOPT(2) to specify either area or volume weighted axisymmetric elements.

KEYOPT(5) defines the element continuum treatment. Two different formulations are available: Lagrangian (default) and Arbitrary Lagrangian-Eulerian (ALE). In addition to setting KEYOPT(5) = 1, you must also set appropriate parameters on the **EDALE** and **EDGCALE** commands in order for the ALE formulation to take affect. See Arbitrary Lagrangian-Eulerian Formulation in the *ANSYS LS-DYNA User's Guide* for more information.

Use the **EDLOAD** command to apply nodal loads and other types of loads described below. For detailed information on how to apply loads in an explicit dynamic analysis, see Loading in the *ANSYS LS-DYNA User's Guide*. Note that when the axisymmetric option (KEYOPT(3) = 1) is selected and KEYOPT(2) = 0 (area weighted option), nodal loads should be input per unit length of circumference. Likewise, when KEYOPT(3) = 1 and KEYOPT(2) = 1 (volume weighted option), nodal loads should be input per radian.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Other aspects of axisymmetric elements are covered in Harmonic Axisymmetric Elements (p. 83). Pressures are always on a 360° basis, irrespective of the KEYOPT(2) setting.

Pressures can be input as surface loads on the element faces (edges) as shown by the circled numbers in Figure 162.1: PLANE162 Geometry (p. 587). Positive normal pressures act into the element.

Other loads that can be applied using the **EDLOAD** command include base accelerations and angular velocities in the x and y directions, and displacements and forces on rigid bodies.

Several types of temperature loading are also available for this element. See Temperature Loading in the *ANSYS LS-DYNA User's Guide*.

The material models available to use with this element will depend on the KEYOPT(3) setting. KEYOPT(3) controls whether the element is a plane stress, plane strain, or axisymmetric element. For all three of these options (KEYOPT(3) = 0, 1, or 2), you can choose the following materials:

- Isotropic Elastic
- Orthotropic Elastic
- Elastic Fluid
- Viscoelastic
- Bilinear Isotropic
- Temperature Dependent Bilinear Isotropic
- Bilinear Kinematic
- Plastic Kinematic
- Power Law Plasticity
- Rate Sensitive Power Law Plasticity
- Strain Rate Dependent Plasticity
- Piecewise Linear Plasticity
- Composite Damage
- Johnson-Cook Plasticity
- Bamman

For the plane stress option (KEYOPT(3) = 0), you can also choose the following materials:

- 3-Parameter Barlat Plasticity
- Barlat Anisotropic Plasticity
- Transversely Anisotropic Elastic Plastic
- Transversely Anisotropic FLD

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For the axisymmetric and plane strain options (KEYOPT(3) = 1 or 2), you can also choose the following materials:

- Blatz-Ko Rubber
- Mooney-Rivlin Rubber
- Elastic-Plastic Hydrodynamic
- Closed Cell Foam
- Low Density Foam
- Crushable Foam
- Honeycomb
- Null
- Zerilli-Armstrong
- Steinberg

PLANE162 Input Summary

Nodes

I, J, K, L

Degrees of Freedom

UX, UY, VX, VY, AX, AY

Note

For explicit dynamic analyses, V(X, Y) refers to nodal velocity, and A(X, Y) refers to nodal acceleration. Although V(X, Y) and A(X, Y) appear as DOFs, they are not actually physical DOFs. However, these quantities are computed as DOF solutions and stored for postprocessing.

Real Constants

None

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, EY, PRXY or NUXY,
ALPX (or CTEX *or* THSX),
DENS, GXY, ALPD, BETD
EDMP command: RIGID, HGLS, ORTHO, FLUID

Surface Loads

Pressures --

face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Body Loads

Temperatures (see Temperature Loading in the *ANSYS LS-DYNA User's Guide*.

Special Features

All nonlinear features allowed for an explicit dynamic analysis.

KEYOPT(2)

Weighting option (used for axisymmetric elements, KEYOPT(3) = 1):

0 --
Area weighted axisymmetric element

1 --
Volume weighted axisymmetric element

KEYOPT(3)

Element behavior:

0 --
Plane stress

1 --
Axisymmetric

2 --
Plane strain (Z strain = 0.0)

KEYOPT(5)

Element continuum treatment:

0 --
Lagrangian (default)

1 --
ALE (Arbitrary Lagrangian-Eulerian)

PLANE162 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 162.1: PLANE162 Element Output Definitions (p. 591)

Several items are illustrated in Figure 162.2: PLANE162 Stress Output (p. 591). The element stresses are output in terms of the global Cartesian coordinate system by default. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 162.2: PLANE162 Stress Output

You can rotate stress results for PLANE162 into a defined coordinate system using the **RSYS** command. However, **RSYS** cannot be used to rotate strain results for this element type.

The following items are available on the results file.

Table 162.1: PLANE162 Element Output Definitions

Name	Definition
S(X, Y, XY)	Stresses
S(1, 2, 3)	Principal stresses
SINT	Stress intensity
SEQV	Equivalent stress
EPTO(X, Y, XY)	Total strains
EPTO(1, 2, 3)	Total principal strains
EPTO(INT)	Total strain intensity
EPTO(EQV)	Total equivalent strain
EPEL(X, Y, XY)	Elastic strains
EPEL(1, 2, 3)	Principal elastic strains
EPEL(INT)	Elastic strain intensity
EPEL(EQV)	Equivalent elastic strain
EPPL(EQV)	Equivalent plastic strain

Note

Stress and total strain are always available. Some components of stress and strain (for example, yz and zx components) are always zero. The availability of elastic strain and equivalent plastic strain depends on the material model used for the element (see Element Output Data in the *ANSYS LS-DYNA User's Guide* for details).

Table 162.2: PLANE162 Item and Sequence Numbers (p. 592) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 162.2: PLANE162 Item and Sequence Numbers (p. 592):

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name
output quantity

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 162.2: PLANE162 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
EPEQ (equivalent plastic strain)	NMISC	1

PLANE162 Assumptions and Restrictions

- The area of the element must be nonzero.
- The element must lie in the global X-Y plane as shown in Figure 162.1: PLANE162 Geometry (p. 587), and the Y-axis must be the axis of symmetry for axisymmetric analyses.
- An axisymmetric structure should be modeled in the +X quadrants.
- A triangular element may be formed by defining duplicate K and L node numbers (see Degenerated Shape Elements (p. 39)).

PLANE162 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SHELL163

Explicit Thin Structural Shell

<> <> <> <> <> <> <>DY <> <> <>
Product Restrictions

SHELL163 Element Description

SHELL163 is a 4-node element with both bending and membrane capabilities. Both in-plane and normal loads are permitted. The element has 12 degrees of freedom at each node: translations, accelerations, and velocities in the nodal x, y, and z directions and rotations about the nodal x, y, and z-axes.

This element is used in explicit dynamic analyses only. Refer to the *LS-DYNA Theoretical Manual* for more information.

Figure 163.1: SHELL163 Geometry

SHELL163 Input Data

The following real constants are provided for SHELL163. SHRF is the shear factor. NIP is the number of integration points through the thickness of the element, up to a maximum of 100. If NIP is input as 0 or blank, ANSYS defaults the value to 2. T1 - T4 indicate the shell thickness at each of the 4 nodes. NLOC specifies the location of the reference surface for KEYOPT(1) = 1, 6, or 7. The reference surface is used in the formulation of the element stiffness matrix. (NLOC does not define the location of the contact surface.) If you set NLOC = 1 or -1 (top or bottom surface), you must set *SHNU* = -2 on the **EDSHELL** command.

ESOP is the option for the spacing of integration points, and can be either 0 or 1. ESOP is used only if KEYOPT(4) > 0. If you set ESOP = 0, you must define real constants S(i), and WF(i) to define the integration point locations. If KEYOPT(3) = 1, then you must also define BETA(i) and MAT(i) for each integration point. Set ESOP = 1 if the integration points are equally spaced through the thickness such that the shell is subdivided into NIP layers of equal thickness (up to 100 layers).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The thickness is assumed to vary smoothly over the area of the element, with the thickness input at the 4 nodes. If the element has a constant thickness, only TK(I) need be input. If the thickness is not constant, all four thicknesses must be input.

If you set ESOP = 0 and define the integration points using S(i), and WF(i), and possibly BETA(i) and MAT(i), note the following:

- If KEYOPT(1) = 1, 6, 7, or 11, then the thicknesses you define will remain defined through the results determination.
- If KEYOPT(1) = 2, 3, 4, 5, 8, 9, 10, or 12, then the ANSYS program overrides any thickness values you specify and averages the thicknesses for the results determination.

S(i) is the relative coordinate of the integration point and must be within the range -1 to 1. WF(i) is the weighting factor for the i-th integration point. It is calculated by dividing the thickness associated with the integration point by the actual shell thickness (that is, $\Delta t_i/t$); see Figure 163.2: Arbitrary Ordering of Integration Points for User Defined Shell Integration Rule (p. 594). In the user defined shell integration rule, the ordering of the integration points is arbitrary. If using these real constants to define integration points, then S(i) and WF(i) must both be specified for each integration point (maximum of 100). BETA(i) is the material angle (in degrees) at the i-th integration point and must be specified for each integration point. The material model (BKIN, MKIN, MISO, etc.) is not allowed to change within an element, although the material properties (EX, NUXY, etc.), as defined per MAT(i), can change. However, the density may not vary through the thickness of the shell element. If more than one material is used, and the densities vary between materials, the density of the material of the first layer will be used for the entire element.

If KEYOPT(4) = 0, the integration rule is defined by KEYOPT(2). The Gauss rule (KEYOPT(2) = 0) is valid for up to five layers (integration points). The trapezoidal rule (KEYOPT(2) = 1) allows up to 100 layers, but is not recommended for less than 20 layers, especially if bending is involved.

Figure 163.2: Arbitrary Ordering of Integration Points for User Defined Shell Integration Rule

Use the **EDLOAD** command to apply nodal loads and other load types described below. For detailed information on how to apply loads in an explicit dynamic analysis, see Loading in the *ANSYS LS-DYNA User's Guide*.

Pressures can be input as surface loads on the element midsurfaces. Positive normal pressures act into the element (that is, positive pressure acts in the negative z direction). Note, however, that pressure *is*

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

actually applied to the midsurface. See Figure 163.3: Nodal Numbering for Pressure Loads (Positive Pressure Acts in Negative Z Direction) (p. 595).

Figure 163.3: Nodal Numbering for Pressure Loads (Positive Pressure Acts in Negative Z Direction)

Base accelerations and angular velocities in the x, y, and z directions can be applied at the nodes using the **EDLOAD** command. To apply these loads, you need to first select the nodes and create a component. The load is then applied to that component. Each node in the component will have the specified load.

You can also use the **EDLOAD** command to apply loads (displacements, forces, etc.) on rigid bodies.

Several types of temperature loading are also available for this element. See Temperature Loading in the *ANSYS LS-DYNA User's Guide*.

For this element, you can choose from the following materials:

- Isotropic Elastic
- Orthotropic Elastic
- Bilinear Kinematic
- Plastic Kinematic
- Blatz-Ko Rubber
- Bilinear Isotropic
- Temperature Dependent Bilinear Isotropic
- Power Law Plasticity
- Strain Rate Dependent Plasticity
- Composite Damage
- Piecewise Linear Plasticity

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Modified Piecewise Linear Plasticity
- Mooney-Rivlin Rubber
- Barlat Anisotropic Plasticity
- 3-Parameter Barlat Plasticity
- Transversely Anisotropic Elastic Plastic
- Rate Sensitive Power Law Plasticity
- Transversely Anisotropic FLD
- Elastic Viscoplastic Thermal
- Johnson-Cook Plasticity
- Bamman

The orthotropic elastic material model does not accept integration point angles (BETA(i)). Therefore, to model a composite material, you need to use the composite damage material model. If you do not wish to use the damage features of this material model, just set the required strength values to zero.

KEYOPT(1) allows you to specify 1 of 12 element formulations for SHELL163 (see "SHELL163 Input Summary" (p. 597)). A brief description about each element formulation follows:

The Hughes-Liu element formulation (KEYOPT(1) = 1) is based on a degenerated continuum formulation. This formulation results in substantially large computational costs, but it is effective when very large deformations are expected. This formulation treats warped configurations accurately but does not pass the patch test. It uses one-point quadrature with the same hourglass control as the Belytschko-Tsay.

The Belytschko-Tsay (default) element formulation (KEYOPT(1) = 0 or 2) is the fastest of the explicit dynamics shells. It is based on the Mindlin-Reissner assumption, so transverse shear is included. It does not treat warped configurations accurately, so it should not be used in coarse mesh models. One-point quadrature is used with hourglass control. A default value is set for the hourglass parameter. When hourglassing appears, you should increase this parameter to avoid hourglassing. It does not pass the patch test.

The BCIZ Triangular Shell element formulation (KEYOPT(1) = 3) is based on a Kirchhoff plate theory and uses cubic velocity fields. Three sets of quadrature points are used in each element, so it is relatively slow. It passes the patch test only when the mesh is generated from three sets of parallel lines.

The C^0 Triangular Shell element formulation (KEYOPT(1) = 4) is based on a Mindlin-Reissner plate theory and uses linear velocity fields. One quadrature point is used in the element formulation. This formulation is rather stiff, so it should not be used for constructing an entire mesh, only to transition between meshes.

The Belytschko-Tsay membrane element formulation (KEYOPT(1) = 5) is the same as the Belytschko-Tsay but with no bending stiffness.

The S/R Hughes-Liu element formulation (KEYOPT(1) = 6) is the same as the Hughes-Liu, but instead of using one-point quadrature with hourglass control, this formulation uses selective reduced integration. This increases the cost by a factor of 3 to 4, but avoids certain hourglass modes; certain bending hourglass modes are still possible.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The S/R corotational Hughes-Liu element formulation (KEYOPT(1) = 7) is the same as the S/R Hughes-Liu except it uses the corotational system.

The Belytschko-Leviathan shell formulation (KEYOPT(1) = 8) is similar to the Belytschko-Wong-Chiang with one-point quadrature but it uses physical hourglass control, thus no user-set hourglass control parameters need to be set.

The fully-integrated Belytschko-Tsay membrane element formulation (KEYOPT(1) = 9) is the same as the Belytschko-Tsay membrane except is uses a 2 x 2 quadrature instead of a one-point quadrature. This formulation is more robust for warped configurations.

The Belytschko-Wong-Chiang formulation (KEYOPT(1) = 10) is the same as the Belytschko-Tsay except the shortcomings in warped configuration are avoided. Costs about 10% more.

The fast (corotational) Hughes-Liu formulation (KEYOPT(1) = 11) is the same as the Hughes-Liu except this formulation uses the corotational system.

The fully-integrated Belytschko-Tsay shell element formulation (KEYOPT(1) = 12) uses a 2 x 2 quadrature in the shell plane and is about 2.5 times slower than KEYOPT(1) = 2. It is useful in overcoming hourglass modes. The shear locking is remedied by introducing an assumed strain for the transverse shear.

Of the twelve shell element formulations, only KEYOPT(1) = 1, 2, 6, 7, 8, 9, 10, 11, and 12 are valid for an explicit-to-implicit sequential solution. For metal forming analyses, KEYOPT(1) = 10 and 12 are recommended in order to properly account for warping.

When the Mooney-Rivlin Rubber material model is used with SHELL163 elements, the LS-DYNA code will automatically use a total Lagrangian modification of the Belytschko-Tsay formulation instead of using the formulation you specify via KEYOPT(1). This program-chosen formulation is required to address the special needs of the hyperelastic material.

A summary of the element input is given in "SHELL163 Input Summary" (p. 597). A general description of element input is given in Element Input (p. 43).

SHELL163 Input Summary

Nodes

I, J, K, L

Degrees of Freedom

UX, UY, UZ, VX, VY, VZ, AX, AY, AZ, ROTX, ROTY, ROTZ

Note

For explicit dynamic analyses, V(X, Y, Z) refers to nodal velocity, and A(X, Y, Z) refers to nodal acceleration. Although V(X, Y, Z) and A(X, Y, Z) appear as DOFs, they are not actually physical DOFs. However, these quantities are computed as DOF solutions and stored for postprocessing.

Real Constants

SHRF, NIP, T1, T2, T3, T4,
NLOC, ESOP, BETA(i), S(i), WF(i), MAT(i)
(BETA(i), S(i), WF(i), MAT(i) may repeat for each integration point, depending on the keyoption settings.)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specify NLOC only if KEYOPT(1) = 1, 6, 7, or 11.
See Table 163.1: SHELL163 Real Constants (p. 599) for descriptions of the real constants.

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, EY, EZ, NUXY, NUYZ, NUXZ,
PRXY, PRXZ, PRYZ,
ALPX (or CTEX *or* THSX), GXY, GYZ, GXZ,
DENS, ALPD, BETD
EDMP command: RIGID, HGLS (except KEYOPT(1) = 3, 4, 6, 7, 9, and 12), ORTHO

Surface Loads
Pressure (applied on midsurface)

Body Loads
Temperatures (see Temperature Loading in the *ANSYS LS-DYNA User's Guide*.

Special Features
All nonlinear features allowed for an explicit dynamic analysis.

KEYOPT(1)
Element formulation:

1 --
Hughes-Liu

0, 2 --
Belytschko-Tsay (default)

3 --
BCIZ triangular shell

4 --
C^0 triangular shell

5 --
Belytschko-Tsay membrane

6 --
S/R Hughes-Liu

7 --
S/R corotational Hughes-Liu

8 --
Belytschko-Levithan shell

9 --
Fully integrated Belytschko-Tsay membrane

10 --
Belytschko-Wong-Chiang

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11 --
Fast (corotational) Hughes-Liu

12 --
Fully integrated Belytschko-Tsay shell

KEYOPT(2)
Quadrature rule (used for standard integration rules, KEYOPT(4) = 0):

0 --
Gauss rule (up to five integration points are permitted)

1 --
Trapezoidal rule (up to 100 integration points are permitted)

KEYOPT(3)
Flag for layered composite material mode:

0 --
Non-composite material mode

1 --
Composite material mode; a material angle is defined for each through thickness integration point

KEYOPT(4)
Integration rule ID:

0 --
Standard integration option

n --
User-defined integration rule ID (valid range is 1 to 9999; if selected, it overrides the integration rule set by KEYOPT(2))

Table 163.1: SHELL163 Real Constants

No.	Name	Description
1	SHRF	Shear factor Suggested value: 5/6; if left blank, defaults to 1
2	NIP	Number of integration points If input as 0 or blank, defaults to 2.
3	T1	Shell thickness at node I
4	T2	Shell thickness at node J
5	T3	Shell thickness at node K
6	T4	Shell thickness at node L
7	NLOC	Location of reference surface = 1, top surface = 0, middle surface

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

No.	Name	Description
		= -1, bottom surface Used only if KEYOPT(1) = 1, 6, or 7.
8	ESOP	Option for the spacing of integration points: 0 - Integration points are defined using real constants S(i) and WF(i). 1 - Integration points are equally spaced through the thickness such that the shell is subdivided into NIP layers of equal thickness.
9, 13, 17,... 405	BETA(i)	Material angle at the i-th integration point.[1]
10, 14, 18, ... 406	S(i)	Coordinate of integration point in the range -1 to 1. i = 1, NIP (NIP = 100 max)[1]
11,15, 19, ... 407	WF(i)	Weighting factor; that is, the thickness associated with the integration point divided by the actual shell thickness. i = 1, NIP (NIP = 100 max)[1]
12, 16, 20, ... 408	MAT(i)	Material ID for each layer. [1]

1. If KEYOPT(3) = 1, then BETA(i), S(i), WF(i), and MAT(i) should be specified for each integration point. For example, for 20 integration points, you would specify BETA(1), S(1), WF(1), MAT(1), BETA(2), S(2), WF(2), MAT(2), ..., BETA(20), S(20), WF(20), MAT(20). If KEYOPT(3) = 0, then only S(i) and WF(i) need to be specified. The material used will be that specified by the **MAT** command.

SHELL163 Output Data

To store output data for this element, you must specify the number of output locations for which you want data using the **EDINT**,SHELLIP command during solution. To review the stored data for a specified layer, use the **LAYER**,NUM command. However, be aware that the output location for this data is always at the integration point. "Top" and "bottom" refer to the top or bottom integration point, which is not necessarily the top or bottom surface.

Stress data is always output from the bottom of the shell to the top. See Figure 163.2: Arbitrary Ordering of Integration Points for User Defined Shell Integration Rule (p. 594).

In all cases (default and otherwise), strain is always output for two layers only: Layer 1 = bottom and layer 2 = top.

The number of integration points specified by real constant NIP controls the output locations through the thickness of the shell. If NIP = SHELLIP, then each layer corresponds to an integration point, and those are the locations where you will get output data. If NIP>SHELLIP, then data is output only at the

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SHELLIP number of locations (first bottom layer, then layers 2 through *n* moving up from the bottom). If NIP<SHELLIP (but NIP>2), then results are output only for NIP number of layers.

By default, the number of integration points (NIP) is 2, and the number of output locations/layers (SHELLIP) is 3. In this case, stress data is output in the following order: Layer 1 = bottom, layer 2 = middle, and layer 3 = top. When SHELLIP = 3, the middle layer will be an interpolated value if NIP is an even number or an actual value at an integration point if NIP is an odd number.

If NIP = 1, the integration point is at the element midplane, and only one stress and one strain value are output.

For elements with 2 x 2 integration points in the shell plane (KEYOPT(1) = 6, 7, 9, 12), LS-DYNA performs an averaging of any data output at those points in every layer so that the output is the same for all shell formulations.

For the default **RSYS** setting, strains (EPTO) and generalized stresses (M, T, N) are output in the element coordinate system, and stresses (S) are output in the global Cartesian system for all formulations associated with SHELL163, except the Hughes-Liu formulation. Strain output (EPTO) for the Hughes-Liu formulation (KEYOPT(1) = 1) is output in the global Cartesian system.

You can rotate stress results for this element into another coordinate system using the **RSYS** command. However, **RSYS** has no effect on the stress results for composite SHELL163 elements (KEYOPT(3) = 1). In addition, **RSYS** cannot be used to rotate strain results for any of the SHELL163 element formulations.

The following items are available in the results file.

Table 163.2: SHELL163 Element Output Definitions

Name	Definition
S(X, Y, Z, XY, YZ, XZ)	Stresses
S(1, 2, 3)	Principal stresses
SINT	Stress intensity
SEQV	Equivalent stress
EPTO(X, Y, Z, XY, YZ, XZ)	Total strain
EPTO(1, 2, 3)	Total principal strains
EPTO(INT)	Total strain intensity
EPTO(EQV)	Total equivalent strain
EPEL(X, Y, Z, XY, YZ, XZ)	Elastic strains
EPEL(1, 2, 3)	Principal elastic strains
EPEL(INT)	Elastic strain intensity
EPEL(EQV)	Equivalent elastic strain
EPPL(EQV)	Equivalent plastic strain
M(X, Y, XY)	Element X, Y, and XY moments
N(X. Y)	Out-of-plane X, Y shear
T(X, Y, XY)	In-plane element X, Y, and XY forces

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition
Thick	Element thickness

Note

Stress and total strain are always available. The availability of elastic strain and equivalent plastic strain depends on the material model used for the element (see Element Output Data in the *ANSYS LS-DYNA User's Guide* for details).

Table 163.3: SHELL163 Item and Sequence Numbers (p. 602) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 163.3: SHELL163 Item and Sequence Numbers (p. 602):

Name
output quantity as defined in the Table 163.2: SHELL163 Element Output Definitions (p. 601)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 163.3: SHELL163 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
MX	SMISC	1
MY	SMISC	2
MXY	SMISC	3
NX	SMISC	4
NY	SMISC	5
TX	SMISC	6
TY	SMISC	7
TXY	SMISC	8
EPEQ (top)[1]	NMISC	1
EPEQ (middle)[1], [2]	NMISC	2
EPEQ (bottom)[1]	NMISC	3
Thick[1]	NMISC	4

1. The sequence numbers for NMISC items in this table are based on the assumption that the number of integration points for output (SHELLIP on the **EDINT** command) is set to the default value of 3.

2. If the number of integration points (NIP) is even, the middle EPEQ value (NMISC,2) will be an interpolated value.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The SMISC quantities in the above table are independent of layers (that is, you will get one set of SMISC quantities output per element). However, the NMISC items are layer-dependent, and the order of the NMISC items is dependent on the SHELLIP and NIP values. The order shown in the table corresponds to the default SHELLIP value (SHELLIP = 3). If NIP > 3, it is strongly recommended that you set SHELLIP = NIP. In this case, the **ETABLE** output will go from top (NMISC,1) to bottom (NMISC,*n* where *n* is the total number of layers). If SHELLIP is not equal to NIP, the order of NMISC items will vary. Therefore, you should not use **ETABLE** to access the NMISC items when NIP > 3 and SHELLIP is not equal to NIP.

SHELL163 Assumptions and Restrictions

- Zero area elements are not allowed. This occurs most often whenever the elements are not numbered properly.
- Zero thickness elements or elements tapering down to a zero thickness at any corner are not allowed.
- A triangular element may be formed by defining duplicate K and L node numbers as described in Degenerated Shape Elements (p. 39). In this event, the C^0 triangular shell element (KEYOPT(1) = 4) will be used.
- An assemblage of flat shell elements can produce a good approximation to a curved shell surface provided that each flat element does not extend over more than a 15° arc.

SHELL163 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID164

Explicit 3-D Structural Solid

<> <> <> <> <> <> <>DY <> <> <>
Product Restrictions

SOLID164 Element Description

SOLID164 is used for the 3-D modeling of solid structures. The element is defined by eight nodes having the following degrees of freedom at each node: translations, velocities, and accelerations in the nodal x, y, and z directions.

This element is used in explicit dynamic analyses only. Refer to the *LS-DYNA Theoretical Manual* for more information.

Figure 164.1: SOLID164 Geometry

SOLID164 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 164.1: SOLID164 Geometry (p. 605). The element is defined by eight nodes. Orthotropic material properties may be defined. Use the **EDMP** command to specify an orthotropic material and the **EDLCS** command to define the orthotropic material directions.

By default, SOLID164 uses reduced (one point) integration plus viscous hourglass control for faster element formulation. A fully integrated solid formulation (KEYOPT(1) = 2) is also available.

KEYOPT(5) defines the element continuum treatment. Two different formulations are available: Lagrangian (default) and Arbitrary Lagrangian-Eulerian (ALE). In addition to setting KEYOPT(5) = 1, you must also

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

set appropriate parameters on the **EDALE** and **EDGCALE** commands in order for the ALE formulation to take affect. See Arbitrary Lagrangian-Eulerian Formulation in the *ANSYS LS-DYNA User's Guide* for more information. Some material models that are normally available for this element type are not supported when the ALE formulation is used. See the material list below for details.

Use the **EDLOAD** command to apply nodal loads and other load types described below. For detailed information on how to apply loads in an explicit dynamic analysis, see the *ANSYS LS-DYNA User's Guide*.

Pressures can be input as surface loads on the element faces as shown by the circled numbers in Figure 164.1: SOLID164 Geometry (p. 605). Positive normal pressures act into the element.

Base accelerations and angular velocities in the x, y, and z directions can be applied at the nodes using the **EDLOAD** command. To apply these loads, you need to first select the nodes and create a component. The load is then applied to that component.

You can also use the **EDLOAD** command to apply loads (displacements, forces, etc.) on rigid bodies.

Several types of temperature loading are also available for this element. See Temperature Loading in the *ANSYS LS-DYNA User's Guide*.

For this element, you can choose from the materials listed below. The material models marked by an asterisk (*) are not supported by the ALE formulation (KEYOPT(5) = 1).

- Isotropic Elastic
- Orthotropic Elastic*
- Anisotropic Elastic*
- Bilinear Kinematic
- Plastic Kinematic
- Viscoelastic*
- Blatz-Ko Rubber*
- Bilinear Isotropic
- Temperature Dependent Bilinear Isotropic
- Power Law Plasticity
- Strain Rate Dependent Plasticity
- Composite Damage*
- Concrete Damage*
- Geological Cap
- Piecewise Linear Plasticity*
- Honeycomb*
- Mooney-Rivlin Rubber*

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Barlat Anisotropic Plasticity
- Elastic-Plastic Hydrodynamic
- Rate Sensitive Power Law Plasticity
- Elastic Viscoplastic Thermal
- Closed Cell Foam*
- Low Density Foam
- Viscous Foam*
- Crushable Foam
- Johnson-Cook Plasticity
- Null
- Zerilli-Armstrong
- Bamman*
- Steinberg
- Elastic Fluid

SOLID164 Input Summary

Nodes

I, J, K, L, M, N, O, P

Degrees of Freedom

UX, UY, UZ, VX, VY, VZ, AX, AY, AZ

For explicit dynamic analyses, V(X, Y, Z) refers to nodal velocity, and A(X, Y, Z) refers to nodal acceleration. AlthoughV (X, Y, Z) and A(X, Y, Z) appear as DOFs, they are not actually physical DOFs. However, these quantities are computed as DOF solutions and stored for postprocessing.

Real Constants

None

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, EY, EZ, NUXY, NUYZ, NUXZ,
PRXY, PRXZ, PRYZ,
ALPX (or CTEX *or* THSX),
GXY, GYZ, GXZ,
DENS, ALPD, BETD
EDMP command: RIGID, HGLS, ORTHO, FLUID

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface Loads

Pressures --
face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N), face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads
Temperatures (see Temperature Loading in the *ANSYS LS-DYNA User's Guide*.

Special Features
All nonlinear features allowed for an explicit dynamic analysis.

KEYOPT(1)
Element formulation:

0, 1 --
Constant stress solid element (default)

2 --
Fully integrated selectively-reduced solid

KEYOPT(5)
Element continuum treatment:

0 --
Lagrangian (default)

1 --
ALE (Arbitrary Lagrangian-Eulerian)

SOLID164 Output Data

Output for SOLID164 is listed in Table 164.1: SOLID164 Element Output Definitions (p. 608). If you issue **PRNSOL**, a single set of stress and a single set of strain values is output at all eight nodes; that is, you will get the same sets of values at each node. If you issue **PRESOL**, you will get only a single set of values at the centroid.

You can rotate stress results for SOLID164 into a defined coordinate system using the **RSYS** command. However, **RSYS** cannot be used to rotate strain results for this element type.

The following items are available on the results file.

Table 164.1: SOLID164 Element Output Definitions

Name	Definition
S(X, Y, Z, XY, YZ, XZ)	Stresses
S(1, 2, 3)	Principal stresses
SINT	Stress intensity
SEQV	Equivalent stress
EPTO(X, Y, Z, XY, YZ, XZ)	Total strains
EPTO(1, 2, 3)	Total principal strains
EPTO(INT)	Total strain intensity
EPTO(EQV)	Total equivalent strain

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition
EPEL(X, Y, Z, XY, YZ, XZ)	Elastic strains
EPEL(1, 2, 3)	Principal elastic strains
EPEL(INT)	Elastic strain intensity
EPEL(EQV)	Equivalent elastic strain
EPPL(EQV)	Equivalent plastic strain

Note

Stress and total strain are always available. The availability of elastic strain and equivalent plastic strain depends on the material model used for the element (see Element Output Data in the *ANSYS LS-DYNA User's Guide* for details).

SOLID164 Assumptions and Restrictions

- Zero volume elements are not allowed.
- The element may not be twisted such that it has two separate volumes. This occurs most frequently when the element is not numbered properly.
- The element must have eight nodes.
- A prism-shaped element may be formed by defining duplicate K and L and duplicate O and P node numbers (see Degenerated Shape Elements (p. 39)). A tetrahedron shape is also available. When the degenerated elements are used, be careful in choosing the element formulations. For the tetrahedron shape, SOLID168 should be considered instead.

SOLID164 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

COMBI165

Explicit Spring-Damper

<> <> <> <> <> <> <>DY <> <> <>
Product Restrictions

COMBI165 Element Description

COMBI165 allows you to model simple spring or damper systems, as well as the response of more complicated mechanisms such as the energy absorbers used in passenger vehicle bumpers. These mechanisms are often experimentally characterized in terms of force-displacement curves. This element provides a variety of discrete element formulations that can be used individually or in combination to model complex force-displacement relations.

COMBI165 is a two-node, 1-D element. You cannot define both spring and damper properties for the same element. Separate spring and damper elements are required, but they may use the same nodes (that is, you can overlay two different COMBI165 elements). A COMBI165 element can be attached to any of the other explicit elements.

This element is used in explicit dynamic analyses only. Refer to the *LS-DYNA Theoretical Manual* for more information.

Figure 165.1: COMBI165 Geometry

Both figures above show two COMBI165 elements (a spring and a damper) attached to the same two nodes.

COMBI165 Input Data

The real constants K_d to TDL are optional and do not need to be defined.

For example, if K_d, the dynamic magnification factor, is nonzero, the forces computed from the spring elements are assumed to be the static values and are scaled by an amplification factor to obtain the dynamic value:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$F_{dynamic} = \left(1. + K_d \frac{V}{V_0}\right) F_{static}$$

For example, if it is known that a component shows a dynamic crush force at 15m/s equal to 2.5 times the static crush force, use K_d = 1.5 and V_0 = 15, where V_0 is the test velocity.

Here, clearance (CL) defines a compressive displacement which the spring sustains before beginning the force-displacement relation given by the load curve. If a nonzero clearance is defined, the spring is compressive only.

The deflection limit in compression (CDL) and tension (TDL) is restricted in its application to no more than one spring per node subject to this limit, and to deformable bodies only. For example, in the former case, if three springs are in series, either the center spring or the two end springs may be subject to a limit, but not all three. When the limiting deflection (FD) is reached, momentum conservation calculations are performed and a common acceleration is computed in the appropriate direction. An error termination will occur if a rigid body node is used in a spring definition where compression is limited.

For this element, you can choose from the following materials:

- Linear Elastic Spring
- Linear Viscous Damper
- Elastoplastic Spring
- Nonlinear Elastic Spring
- Nonlinear Viscous Damper
- General Nonlinear Spring
- Maxwell Viscoelastic Spring
- Inelastic Tension or Compression-Only Spring

A summary of the element input is given in "COMBI165 Input Summary" (p. 612). A general description of element input is given in Element Input (p. 43).

COMBI165 Input Summary

Nodes

I, J

Degrees of Freedom

UX, UY, UZ, VX, VY, VZ, AX, AY, AZ (KEYOPT(1) = 0)
ROTX, ROTY, ROTZ (KEYOPT(1) = 1)

Note

For explicit dynamic analyses, V(X, Y, Z) refers to nodal velocity, and A(X, Y, Z) refers to nodal acceleration. Although V(X, Y, Z) and A(X, Y, Z) appear as DOFs, they are not actually

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

physical DOFs. However, these quantities are computed as DOF solutions and stored for postprocessing.

Real Constants

Kd - Dynamic magnification factor,
Vo - Test velocity,
CL - Clearance,
FD - Failure deflection,
CDL - Deflection limit (compression),
TDL - Deflection limit (tension)

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.

MP command: BETD

Surface Loads

None

Body Loads

None

Special Features

All nonlinear features allowed for an explicit dynamic analysis.

KEYOPT(1)

Spring/damper type (translational or torsional):

0 --
The material describes a translational spring/damper

1 --
The material describes a torsional spring/damper

COMBI165 Output Data

Output data for COMBI165 consists of the following:

Table 165.1: COMBI165 Item and Sequence Numbers

Output Quantity Name[1]	ETABLE and ESOL Command Input		
	Description	Item	Seq. No.
MFORX/MMOMX	X-component of member force/moment	SMISC	1
MFORY/MMOMY	Y-component of member force/moment	SMISC	2
MFORZ/MMOMZ	Z-component of member force/moment	SMISC	3
MFORSUM/MMOMSUM	Vector sum of X, Y, and Z components of member force/moment	SMISC	4

1. You must specify *either* force or moment via KEYOPT(1). Note that you cannot specify *both* force and moment.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MFOR: KEYOPT(1) = 0

MMOM: KEYOPT(1) = 1

To output the element data in POST1, you must use the **ETABLE** command. Then, you can use the **PRETAB** command to print the output data. The **RSYS** command has no effect when postprocessing output for this element.

In POST26, you can postprocess the element data using the **ESOL** command only when postprocessing the `Jobname.RST` file. The element results are not available on the `Jobname.HIS` file.

COMBI165 Assumptions and Restrictions

- The time step size calculation is approximated by using the instantaneous stiffness and one-half the nodal mass of the nodes joined by the spring. If the global time step size is controlled by an explicit spring-damper element, instabilities can develop with the default time step size due to the approximations in the step size calculation.
- When used to interconnect under-integrated elements, the explicit spring-damper can sometimes excite the zero-energy hourglass modes.
- To ensure that parts are uniquely defined when using COMBI165, specify a unique set of real constants (**R**), the element type (**ET**), and the material properties (**TB**) for each part. Defining a unique material number (**MAT**) alone is insufficient.

COMBI165 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MASS166

Explicit 3-D Structural Mass

<> <> <> <> <> <> <>DY <> <> <>
Product Restrictions

MASS166 Element Description

MASS166 is a point element having up to nine degrees of freedom: translations, velocities, and accelerations in the nodal x, y, and z directions.

Figure 166.1: MASS166 Geometry

MASS166 Input Data

The mass element is defined by a single node with concentrated mass components (Force*Time^2/Length) in the element coordinate directions about the element coordinate axes. The element also has an option for rotary inertia (without mass) which allows the definition of lumped rotary inertia at a defined nodal point. For the inertia option (KEYOPT(1) = 1), six polar moment of inertia values must be input instead of mass. To include both mass and rotary inertia, you must define two MASS166 elements at the same node.

A summary of the element input is given in "MASS166 Input Summary" (p. 615). A general description of element input is given in Element Input (p. 43).

MASS166 Input Summary

Nodes

I

Degrees of Freedom

UX, UY, UZ, VX, VY, VZ, AX, AY, AZ

Note

For explicit dynamic analyses, V(X, Y, Z) refers to nodal velocity, and A(X, Y, Z) refers to nodal acceleration. Although V(X, Y, Z) and A(X, Y, Z) appear as DOFs, they are not actually physical DOFs. However, these quantities are computed as DOF solutions and stored for postprocessing.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Real Constants

If KEYOPT(1) = 0:

MASS - Concentrated mass (Force*Time^2/Length) (must be > 0)
If KEYOPT(1) = 1:
IXX - Moment of inertia (must be > 0),
IXY - Moment of inertia,
IXZ - Moment of inertia,
IYY - Moment of inertia (must be > 0),
IYZ - Moment of inertia,
IZZ - Moment of inertia (must be > 0)

Material Properties

None, but you must define realistic dummy material properties to make this element behave correctly. The dummy properties will not be used in any solution.

Surface Loads

None

Body Loads

None

Special Features

All nonlinear features allowed for an explicit dynamic analysis.

KEYOPT(1)

Rotary inertia option:

0 --
3-D mass without rotary inertia (default)

1 --
3-D rotary inertia (no mass)

MASS166 Output Data

Nodal displacements are included in the overall displacement solution. There is no printed or post element data output for the mass element.

MASS166 Assumptions and Restrictions

None.

MASS166 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LINK167

Explicit Tension-Only Spar

<> <> <> <> <> <> <>DY <> <> <>
Product Restrictions

LINK167 Element Description

LINK167 allows elastic cables to be realistically modeled; thus, no force will develop in compression.

This element is used in explicit dynamic analyses only.

Figure 167.1: LINK167 Geometry

LINK167 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 167.1: LINK167 Geometry (p. 617). Node K determines the initial orientation of the cross section.

The element is defined by nodes I and J in the global coordinate system. Node K defines a plane (with I and J) containing the element s-axis. The element r-axis runs parallel to the length of the element and through nodes I and J. Node K is always required to define the element axis system and it must not be colinear with nodes I and J. The location of node K is used only to *initially* orient the element.

Real constants for this element are link area (AREA) and offset for cable (OFFSET). For a slack element, the offset should be input as a negative value. For an initial tensile force, the offset should be positive.

The force, *F*, generated by the link is nonzero if and only if the link is in tension. The force is given by:

$F = K \cdot max\ (\Delta\ L, 0.)$

where ΔL is the change in length

$\Delta\ L = current\ length - (initial\ length - offset)$

and the stiffness is defined as:

$$K = \frac{E \times area}{(initial\ length - offset)}$$

You can use only the material type cable for this element. For this material, you need to define the density (DENS) and Young's modulus (EX) or load curve ID. If you specify a load curve ID (**EDMP**,CABLE,VAL1, where VAL1 is the load curve ID), the Young's modulus will be ignored and the load curve will be used instead. The points on the load curve are defined as engineering stress versus engin-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

eering strain (that is, the change in length over the initial length). Use the **EDCURVE** command to define the load curve ID. The unloading behavior follows the loading.

Base accelerations and angular velocities in the x, y, and z directions can be applied at the nodes using the **EDLOAD** command. To apply these loads, you need to first select the nodes and create a component. The load is then applied to that component.

A summary of the element input is given in "LINK167 Input Summary" (p. 618). Additional information about real constants for this element is provided in Table 161.1: BEAM161 Real Constants (p. 578). For more information about this element, see the *LS-DYNA Theoretical Manual*.

LINK167 Input Summary

Nodes
I, J, K (K is the orientation node)

Degrees of Freedom
UX, UY, UZ, VX, VY, VZ, AX, AY, AZ

Note

For explicit dynamic analyses, V(X, Y, Z) refers to nodal velocity, and A(X, Y, Z) refers to nodal acceleration. Although V(X, Y, Z) and A(X, Y, Z) appear as DOFs, they are not actually physical DOFs. However, these quantities are computed as DOF solutions and stored for postprocessing.

Real Constants

AREA - Cross-sectional area
OFFSET - Offset value for cable

Material Properties

MP command: EX, DENS, ALPD, BETD,
EDMP command: CABLE (See Material Models in the *ANSYS LS-DYNA User's Guide*)

Surface Loads
None

Body Loads
None

Special Features
All nonlinear features allowed for an explicit dynamic analysis.

KEYOPTs
None

LINK167 Output Data

Output for LINK167 consists of the following: Axial force

To output the data, you must use the **ETABLE** command. For the ITEM label, specify SMISC. For the COMP label, specify 1 for axial force. Then, you can use the **PRETAB** command to print the output data.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LINK167 Assumptions and Restrictions

- The sum of the element length plus the offset must be greater than zero.
- The cross-sectional area must be greater than zero.

LINK167 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID168

Explicit 3-D 10-Node Tetrahedral Structural Solid

<> <> <> <> <> <> <>DY <> <> <>
Product Restrictions

SOLID168 Element Description

SOLID168 is a higher order 3-D, 10-node explicit dynamic element. It is well suited to modeling irregular meshes such as those produced from various CAD/CAM systems. The element is defined by ten nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions.

By default, SOLID168 uses a quadratic displacement behavior with five point integration (KEYOPT(1) = 0 or 1). A composite formulation which is an assemblage of linear sub-tetrahedral shapes (KEYOPT(1) = 2) is also available. This second formulation effectively overcomes the difficulty of lumped mass calculations and volume locking inherent to the quadratic elements.

Figure 168.1: SOLID168 Geometry

SOLID168 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 168.1: SOLID168 Geometry (p. 621). The element is defined by ten nodes. Orthotropic material properties may be defined. Use the **EDMP** command to specify an orthotropic material and the **EDLCS** command to define the orthotropic material directions.

Use the **EDLOAD** command to apply nodal loads and other load types described below. For detailed information on how to apply loads in an explicit dynamic analysis, see the *ANSYS LS-DYNA User's Guide*.

Pressures can be input as surface loads on the element faces as shown by the circled numbers in Figure 168.1: SOLID168 Geometry (p. 621). Positive normal pressures act into the element.

Base accelerations and angular velocities in the x, y, and z directions can be applied at the nodes using the **EDLOAD** command. To apply these loads, you need to first select the nodes and create a component. The load is then applied to that component. You can also use the **EDLOAD** command to apply loads (displacements, forces, etc.) on rigid bodies.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Several types of temperature loading are also available for this element. See Temperature Loading in the *ANSYS LS-DYNA User's Guide*. For this element, you can choose from the materials listed below.

- Isotropic Elastic
- Orthotropic Elastic
- Anisotropic Elastic
- Bilinear Kinematic
- Plastic Kinematic
- Viscoelastic
- Blatz-Ko Rubber
- Bilinear Isotropic
- Temperature Dependent Bilinear Isotropic
- Power Law Plasticity
- Strain Rate Dependent Plasticity
- Composite Damage
- Concrete Damage
- Geological Cap
- Piecewise Linear Plasticity
- Honeycomb
- Mooney-Rivlin Rubber
- Barlat Anisotropic Plasticity
- Elastic-Plastic Hydrodynamic
- Rate Sensitive Power Law Plasticity
- Elastic Viscoplastic Thermal
- Closed Cell Foam
- Low Density Foam
- Viscous Foam
- Crushable Foam
- Johnson-Cook Plasticity
- Null

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Zerilli-Armstrong
- Bamman
- Steinberg
- Elastic Fluid

SOLID168 Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R

Degrees of Freedom
UX, UY, UZ, VX, VY, VZ, AX, AY, AZ

For explicit dynamic analyses, V(X, Y, Z) refers to nodal velocity, and A(X, Y, Z) refers to nodal acceleration. Although V(X, Y, Z) and A(X, Y, Z) appear as DOFs, they are not actually physical DOFs. However, these quantities are computed as DOF solutions and stored for postprocessing.

Real Constants
None

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, EY, EZ, NUXY, NUYZ, NUXZ,
PRXY, PRXZ, PRYZ,
ALPX (or CTEX *or* THSX),
DENS, ALPD, BETD
EDMP command: RIGID, HGLS, ORTHO, FLUID

Surface Loads

Pressures --
face 1 (J-I-K), face 2 (I-J-L), face 3 (J-K-L), face 4 (K-I-L)

Body Loads

Temperatures --
See Temperature Loading in the *ANSYS LS-DYNA User's Guide*

Special Features
All nonlinear features allowed for an explicit dynamic analysis.

KEYOPT(1)
Element formulation:

0, 1 --
Quadratic interpolation

2 --
Composite (assemblages of linear tetrahedral shapes)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID168 Output Data

Output for SOLID168 is listed in Table 168.1: SOLID168 Element Output Definitions (p. 624). If you issue **PRNSOL**, a single set of stress and a single set of strain values is output at all ten nodes; that is, you will get the same sets of values at each node. If you issue **PRESOL**, you will get only a single set of values at the centroid.

You can rotate stress results for SOLID168 into a defined coordinate system using the **RSYS** command. However, **RSYS** cannot be used to rotate strain results for this element type.

The following items are available on the results file.

Table 168.1: SOLID168 Element Output Definitions

Name	Definition
S:X, Y, Z, XY, YZ, XZ	Stresses
S:1, 2, 3	Principal stresses
S:INT	Stress intensity
S:EQV	Equivalent stress
EPTO:X, Y, Z, XY, YZ, XZ	Total strains
EPTO:1, 2, 3	Total principal strains
EPTO:INT	Total strain intensity
EPTO:EQV	Total equivalent strain
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains
EPEL:1, 2, 3	Principal elastic strains
EPEL:INT	Elastic strain intensity
EPEL:EQV	Equivalent elastic strains
EPPL:EQV	Equivalent plastic strains

Note

Stress and total strain are always available. The availability of elastic strain and equivalent plastic strain depends on the material model used for the element (see Element Output Data in the *ANSYS LS-DYNA User's Guide* for details).

SOLID168 Assumptions and Restrictions

- Zero volume elements are not allowed.
- The element may not be twisted such that it has two separate volumes. This occurs most frequently when the element is not numbered properly.
- The element must have ten nodes.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID168 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

TARGE169

2-D Target Segment

MP ME ST PR PRN DS EM <> PP EME MFS
Product Restrictions

TARGE169 Element Description

TARGE169 is used to represent various 2-D "target" surfaces for the associated contact elements (CONTA171, CONTA172, and CONTA175). The contact elements themselves overlay the solid elements describing the boundary of a deformable body and are potentially in contact with the target surface, defined by TARGE169. This target surface is discretized by a set of target segment elements (TARGE169) and is paired with its associated contact surface via a shared real constant set. You can impose any translational or rotational displacement, temperature, voltage, and magnetic potential on the target segment element. You can also impose forces and moments on target elements. See TARGE169 in the *Mechanical APDL Theory Reference* for more details about this element. To represent 3-D target surfaces, use TARGE170, a 3-D target segment element. For rigid targets, these elements can easily model complex target shapes. For flexible targets, these elements will overlay the solid elements describing the boundary of the deformable target body.

Figure 169.1: TARGE169 Geometry

TARGE169 Input Data

The target surface is modeled through a set of *target segments,* typically, several target segments comprise one target surface.

The target surface can either be rigid or deformable. For modeling rigid-flexible contact, the rigid surface must be represented by a target surface. For flexible-flexible contact, one of the deformable surfaces must be overlayed by a target surface. See the *Contact Technology Guide* for more information about designating contact and target surfaces.

The target and associated contact surfaces are identified by a shared real constant set. This real constant set includes all real constants for both the target and contact elements.

Each target surface can be associated with only one contact surface, and vice-versa. However, several contact elements could make up the contact surface and thus come in contact with the same target

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

surface. Likewise, several target elements could make up the target surface and thus come in contact with the same contact surface. For either the target or contact surfaces, you can put many elements in a single target or contact surface, or you can localize the contact and target surfaces by splitting the large surfaces into smaller target and contact surfaces, each of which contain fewer elements.

If one contact surface may contact more than one target surface, you must define duplicate contact surfaces that share the same geometry but relate to separate targets, that is, have separate real constant set numbers.

For any target surface definition, the node ordering of the target segment element is critical for proper detection of contact. The nodes must be ordered so that, for a 2-D surface, the associated contact elements (CONTA171, CONTA172, or CONTA175) must lie to the right of the target surface when moving from target node I to target node J. For a rigid 2-D complete circle, contact must occur on the outside of the circle; internal contacting is not allowed.

Considerations for Rigid Targets

Each target segment is a single element with a specific shape, or *segment type*. The segment types are defined by one, two, or three nodes and a target shape code, **TSHAP**, and are described in Table 169.1: TARGE169 2-D Segment Types, Target Shape Codes, and Nodes (p. 628). The **TSHAP** command indicates the geometry (shape) of the element. The segment dimensions are defined by a real constant (R1), and the segment location is determined by the nodes. ANSYS supports seven 2-D segment types; see Table 169.1: TARGE169 2-D Segment Types, Target Shape Codes, and Nodes (p. 628).

Table 169.1: TARGE169 2-D Segment Types, Target Shape Codes, and Nodes

TSHAP	Segment Type	Node1 (DOF)	Node 2 (DOF)[1]	Node 3 (DOF)	R1[2]	R2
LINE	Straight line	1st corner pt (UX, UY) (TEMP) (VOLT) (AZ)	2nd corner pt (UX, UY) (TEMP) (VOLT) (AZ)	None	None	None
ARC	Arc, clockwise	1st corner pt (UX, UY) (TEMP) (VOLT) (AZ)	2nd corner pt (UX, UY) (TEMP) (VOLT) (AZ)	Circle center pt (UX, UY) (TEMP) (VOLT) (AZ)	None	None
CARC	Arc, counter- clockwise	1st corner pt (UX, UY) (TEMP) (VOLT) (AZ)	2nd corner pt (UX, UY) (TEMP) (VOLT) (AZ)	Circle center pt (UX, UY) (TEMP) (VOLT) (AZ)	None	None
PARA	Parabola	1st corner pt (UX, UY) (TEMP) (VOLT) (AZ)	2nd corner pt (UX, UY) (TEMP) (VOLT) (AZ)	Midside pt (UX, UY) (TEMP) (VOLT) (AZ)	None	None
CIRC	Circle	Circle center pt (UX, UY) (TEMP) (VOLT) (AZ)	None	None	Radius	None
PILO	Pilot node	1st point (UX, UY, ROTZ)	None	None	None	None

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TSHAP	Segment Type	Node1 (DOF)	Node 2 (DOF)[1]	Node 3 (DOF)	R1[2]	R2
		(TEMP) (VOLT) (AZ)				
POINT	Point[3]	1st point (UX, UY)	None	None	None	None

1. The DOF available depends on the setting of KEYOPT(1) for the associated contact element. For more information, see the element documentation for CONTA171, CONTA172, or CONTA175.

2. When creating a circle via direct generation, define the real constant R1 *before* creating the element.

3. Rigid surface node. This segment type is only used to apply boundary conditions to rigid target surfaces.

Figure 169.2: TARGE169 2-D Segment Types

For simple rigid target surfaces, you can define the target segment elements individually by direct generation. You must first specify the *SHAPE* argument for the **TSHAP** command. When creating circles through direct generation, you must also define the real constant R1 before creating the element. Real constant R1 (see Table 169.1: TARGE169 2-D Segment Types, Target Shape Codes, and Nodes (p. 628)) defines the radius of the target circle.

For general 2-D rigid surfaces, target segment elements can be defined by line meshing (**LMESH**). You can also use keypoint meshing (**KMESH**) to generate the pilot node.

If the TARGE169 elements will be created via automatic meshing (**LMESH** or **KMESH**), then the **TSHAP** command is ignored and ANSYS chooses the correct shape automatically.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For rigid-to-flexible contact, by default, ANSYS automatically fixes the structural degree of freedom for rigid target nodes if they aren't explicitly constrained (KEYOPT(2) = 0). If you wish, you can override the automatic boundary condition settings by setting KEYOPT(2) = 1 for the target elements. For flexible-to-flexible contact, no special boundary conditions treatment is performed, and the KEYOPT(2) = 0 setting should be used.

For each rigid-flexible contact pair, you can assign only one pilot node to an entire rigid target surface (or none if it is not needed). The pilot node, unlike the other segment types, is used to define the degrees of freedom for the entire target surface. This node can be any of the target surface nodes, but it does not have to be. All possible rigid motions of the target surface will be a combination of a translation and a rotation around the pilot node. The pilot node provides a convenient and powerful way to assign boundary conditions such as rotations, translations, moments, temperature, voltage, and magnetic potential on an entire rigid target surface. By default (KEYOPT(2) = 0), you can assign the boundary conditions only to the pilot node, eliminating the need to assign boundary conditions to individual target nodes, thus reducing the chance of errors. ANSYS will also automatically fix the structural degrees of freedom on the pilot node if they aren't explicitly constrained.

By setting KEYOPT(2) = 1 for the target elements, you can apply boundary conditions on any rigid target nodes rather than only on the pilot node. It is your responsibility to make sure the rigid target surface is not under-constrained or over-constrained. It is still recommended that you apply all boundary conditions on the pilot node, even when KEYOPT(2) = 1.

Considerations for Deformable Target Surfaces

For general deformable surfaces, you will normally use the **ESURF** command to overlay the target elements on the boundary of the existing mesh. Note that the segment types (**TSHAP** command) should not be used for this case.

Considerations for Geometry Correction

In general, curved contact and target surfaces can be well approximated by linear or quadratic contact and target elements when the mesh is sufficiently refined. However, in certain circumstances (for example, when linear elements are used or when the midside nodes of quadratic elements do not lie exactly on the initial curved geometry because a third party mesh generator was used), using a straight line in place of the true curved geometry can significantly affect the accuracy of contact stresses. An optional geometric correction can be used for a circular (or nearly circular) arc via **SECTYPE** and **SECDATA** section commands. The defined geometry correction can be applied to specific contact elements via a section ID (**SECNUM** command). For details, see Geometry Correction for Contact and Target Surfaces in the *Contact Technology Guide*.

Considerations for Fluid Penetration Loading

To model fluid penetration loads, use the **SFE** command to specify the fluid pressure and fluid penetration starting points. For more information, see Applying Fluid Pressure-Penetration Loads in the *Contact Technology Guide*.

Considerations for Thermal Contact Analysis

By default, the temperature is set to the value of TUNIF, and if this has no explicit value the temperature is set to zero. For thermal contact analysis, such as convection and radiation modeling, the behavior of a thermal contact surface (whether a “near-field” or “free” surface) is usually based on the contact status. Contact status affects the behavior of the contact surface as follows:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If the contact surface is outside the pinball region, its behavior is as a far-field of free surface. In this instance, convection/radiation occurs with the ambient temperature.
- If the contact surface is inside the pinball region, the behavior is as a near-field surface.

However, the thermal contact surface status is ignored if KEYOPT(3) = 1 is set, and the surface is always treated as a free surface (see CONTA171, CONTA172, or CONTA175 for details).

A summary of the element input is given in "TARGE169 Input Summary" (p. 631). A general description of element input is given in Element Input (p. 43).

TARGE169 Input Summary

Nodes
I, J, K (J and K are not required for all segment types)

Degrees of Freedom
UX, UY, ROTZ, TEMP, VOLT, AZ (ROTZ is used for the pilot node only)

Real Constants
R1, R2, [the others are defined through the associated CONTA171, CONTA172, or CONTA175 element]

Material Properties
None

Surface Loads
Pressure, Face 1 (I-J) (opposite to target normal direction)

Body Loads
None

Special Features

Birth and death
Fluid pressure penetration
Linear perturbation
Nonlinearity
Section definition for geometry correction of spherical and revolute surfaces

KEYOPT(2)
Boundary conditions for rigid target nodes:

0 --
Automatically constrained by ANSYS

1 --
Specified by user

KEYOPT(3)
Behavior of thermal contact surface:

0 --
Based on contact status

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
Treated as free-surface

KEYOPT(4)
DOF set to be constrained on the dependent or independent DOF for internally-generated multipoint constraints (MPCs). This option is used for these situations: solid-solid and shell-shell assemblies; surface-based constraints that use a single pilot node for the target element; and rigid target surfaces that use the KEYOPT(2) = 1 setting.

***n* --**
Enter a three digit value that represents the DOF set to be constrained. The first to third digits represent ROTZ, UY, UX, respectively. The number 1 (one) indicates the DOF is active, and the number 0 (zero) indicates the DOF is not active. For example, 011 means that UX and UY will be used in the multipoint constraint. Leading zeros may be omitted; for example, you can enter 1 to indicate that UX is the only active DOF. If KEYOPT(4) = 0 (which is the default) or 111, all DOF are constrained.

KEYOPT(6)
Symmetry condition of a constrained surface. This option is only used for a force-distributed constraint that uses a single pilot node for the target element:

0 --
No symmetry on the constrained surface (default).

1 --
The constrained surface has a symmetry condition with respect to the y axis of the nodal coordinate system of the pilot node.

10 --
The constrained surface has a symmetry condition with respect to the x axis of the nodal coordinate system of the pilot node.

Note

Keep the following points in mind when using this symmetry condition:

- When a symmetry condition is used, the pilot node must be defined on the symmetry axis.
- KEYOPT(6) = 11 is not valid input.
- When the constrained surface is defined on a 2-D axisymmetric model and the pilot node is located on the y axis of the nodal coordinate system, KEYOPT(6) = 1 will be set internally. In this case, do not set KEYOPT(6) = 10; otherwise, the initial constraint equations will be redundant and the solution will be invalid.

TARGE169 Output Data

The solution output associated with the element is shown in Table 169.2: TARGE169 Element Output Definitions (p. 633). The following notation is used:

The Element Output Definitions table uses the following notation:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 169.2: TARGE169 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes I, J, and K	Y	Y
ITRGET	Target surface number (assigned by ANSYS)	Y	Y
TSHAP	Segment shape type	Y	Y
ISEG	Segment numbering	1	1
FPRS	Actual applied fluid penetration pressure	Y	Y

1. An internal segment number determined by ANSYS. ISEG for the target element has a different meaning than ISEG reported for the contact element.

You can display or list the actual fluid pressure applied to the target element through several POST1 postprocessing commands, as shown below:

```
PLESOL,CONT,FPRS
PLNSOL,CONT,FPRS
PRESOL,CONT
PRNSOL,CONT
```

Note that only the FPRS (fluid penetration pressure) output item is meaningful when the **PRESOL** and **PRNSOL** commands are used for target elements.

Table 169.3: TARGE169 Item and Sequence Numbers (p. 634) lists output available through the **ETABLE** command using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 169.3: TARGE169 Item and Sequence Numbers (p. 634):

Name
output quantity as defined in the Table 169.2: TARGE169 Element Output Definitions (p. 633)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

I,J

sequence number for data at nodes I, J

Table 169.3: TARGE169 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
FPRS	SMISC	3	1	2

TARGE169 Assumptions and Restrictions

- The 2-D segment element must be defined in an X-Y plane.
- For each pilot node, ANSYS automatically defines an internal node and an internal constraint equation. The rotational DOF of the pilot node is connected to the translational DOF of the internal node by the internal constraint equation. ANSYS, Inc. recommends against using external constraint equations or coupling on pilot nodes; if you do, conflicts may occur, yielding incorrect results.
- For circular arcs, the third node defines the actual center of the circle and must be defined accurately when the element is generated and must be moved consistently with the other nodes during the deformation process. If the third node is not moved consistently with the other nodes, the arc shape will change with that node's movement. To ensure the correct behavior, apply all boundary conditions to a pilot node.
- For parabolic segments, the third point must lie at the middle of the parabola.
- Generally speaking, you should not change the R1 real constant between load steps or during restart stages; otherwise ANSYS assumes the radius of the circle varies between the load steps. When using direct generation, the real constant R1 for circles may be defined before the input of the element nodes. If multiple rigid circles are defined, each having a different radius, they must be defined by different target surfaces.
- For rotation of a rigid body constrained only by a bonded, rigid-flexible contact pair with a pilot node, use the MPC algorithm or a surface-based constraint as described in Multipoint Constraints and Assemblies in the *Contact Technology Guide*. Penalty-based algorithms can create undesirable rotational energies in this situation.

TARGE169 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TARGE170

3-D Target Segment

MP ME ST PR PRN DS EM <> PP EME MFS
Product Restrictions

TARGE170 Element Description

TARGE170 is used to represent various 3-D "target" surfaces for the associated contact elements (CONTA173, CONTA174, CONTA175, CONTA176, and CONTA177). The contact elements themselves overlay the solid, shell, or line elements describing the boundary of a deformable body and are potentially in contact with the target surface, defined by TARGE170. This target surface is discretized by a set of target segment elements (TARGE170) and is paired with its associated contact surface via a shared real constant set. You can impose any translational or rotational displacement, temperature, voltage, and magnetic potential on the target segment element. You can also impose forces and moments on target elements. See TARGE170 in the *Mechanical APDL Theory Reference* for more details about this element. To represent 2-D target surfaces, use TARGE169, a 2-D target segment element.

For rigid target surfaces, these elements can easily model complex target shapes. For flexible targets, these elements will overlay the solid, shell, or line elements describing the boundary of the deformable target body.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 170.1: TARGE170 Geometry

TARGE170 Input Data

The target surface is modeled through a set of *target segments,* typically, several target segments comprise one target surface.

The target surface can either be rigid or deformable. For modeling rigid-flexible contact, the rigid surface must be represented by a target surface. For flexible-flexible contact, one of the deformable surfaces must be overlayed by a target surface. See the *Contact Technology Guide* for more information about designating contact and target surfaces.

The target and associated contact surfaces are identified via a shared real constant set. This real constant set includes all real constants for both the target and contact elements.

Each target surface can be associated with only one contact surface, and vice-versa. However, several contact elements could make up the contact surface and thus come in contact with the same target surface. Likewise, several target elements could make up the target surface and thus come in contact with the same contact surface. For either the target or contact surfaces, you can put many elements in

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a single target or contact surface, or you can localize the contact and target surfaces by splitting the large surfaces into smaller target and contact surfaces, each of which contain fewer elements.

If a contact surface may contact more than one target surface, you must define duplicate contact surfaces that share the same geometry but relate to separate targets, that is, that have separate real constant set numbers.

Figure 170.2: TARGE170 Segment Types (p. 639) shows the available segment types for TARGE170. The general 3-D surface segments (3-node and 6-node triangles, and 4-node and 8-node quadrilaterals) and the primitive segments (cylinder, cone, and sphere) can be paired with 3-D surface-to-surface contact elements, CONTA173 and CONTA174, the 3-D node-to-surface contact element, CONTA175, and the 3-D line-to-surface contact element, CONTA177. The line segments (2-node line and 3-node parabola) can only be paired with the 3-D line-to-line contact element, CONTA176, to model 3-D beam-to-beam contact.

For any target surface definition, the node ordering of the target segment element is critical for proper detection of contact. For the general 3-D surface segments (triangle and quadrilateral segment types), the nodes must be ordered so that the outward normal to the target surface is defined by the right hand rule (see Figure 170.2: TARGE170 Segment Types (p. 639)). Therefore, for the surface target segments, the outward normal by the right hand rule is consistent to the external normal. For 3-D line segments (straight line and parabolic line), the nodes must be entered in a sequence that defines a continuous line. For a rigid cylinder, cone, or sphere, contact must occur on the outside of the elements; internal contacting of these segments is not allowed.

Considerations for Rigid Target Surfaces

Each target segment of a rigid surface is a single element with a specific shape, or *segment type*.The segment types are defined by several nodes and a target shape code, **TSHAP**, and are described in Table 170.1: TARGE170 3-D Segment Types, Target Shape Codes, and Nodes (p. 637). The **TSHAP** command indicates the geometry (shape) of the element. The segment radii are defined by real constants (R1 and R2), and the segment location is determined by the nodes. ANSYS supports eleven 3-D segment types; see Table 170.1: TARGE170 3-D Segment Types, Target Shape Codes, and Nodes (p. 637).

Table 170.1: TARGE170 3-D Segment Types, Target Shape Codes, and Nodes

TSHAP	Segment Type	Nodes (DOF)[1]	R1	R2
TRIA	3-node triangle	1st - 3rd nodes are corner points (UX, UY, UZ) (TEMP) (VOLT) (MAG)	None	None
QUAD	4-node quadrilateral	1st - 4th nodes are corner points (UX, UY, UZ) (TEMP) (VOLT) (MAG)	None	None
TRI6	6-node triangle	1st - 3rd nodes are corner points, 4th - 6th are midside nodes (UX, UY, UZ) (TEMP) (VOLT) (MAG)	None	None
QUA8	8-node quadrilateral	1st - 4th nodes are corner points, 5th - 8th are midside nodes (UX, UY, UZ) (TEMP) (VOLT) (MAG)	None	None
LINE	2-node straight line	1st - 2nd nodes are line end points (UX, UY, UZ)	Target Radius[4]	Contact Radius[5]
PARA	3-node parabola	1st - 2nd nodes are line end points, 3rd is a midside node (UX, UY, UZ)	Target Radius[4]	Contact Radius[5]

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TSHAP	Segment Type	Nodes (DOF)[1]	R1	R2
CYLI	Cylinder[2]	1st - 2nd nodes are axial end points (UX, UY, UZ) (TEMP) (VOLT) (MAG)	Radius	None
CONE	Cone[2]	1st - 2nd nodes are axial end points (UX, UY, UZ) (TEMP) (VOLT) (MAG)	Radius at node 1	Radius at node 2
SPHE	Sphere[2]	Sphere center point (UX, UY, UZ) (TEMP) (VOLT) (MAG)	Radius	None
PILO	Pilot node[3]	1st point: (UX, UY, UZ, ROTX, ROTY, ROTZ) (TEMP) (VOLT) (MAG)	None	None
POINT	Point[6]	1st point: (UX, UY, UZ)	None	None

1. The DOF available depends on the setting of KEYOPT(1) of the associated contact element. Refer to the element documentation for either CONTA173, CONTA174, or CONTA175 for more details.

2. When creating a cylinder, cone, or sphere via direct generation, define the real constant set *before* creating the element.

3. Only pilot nodes have rotational degrees of freedom (ROTX, ROTY, ROTZ).

4. Input the target radius as a negative value when modeling internal pipe-to-pipe contact (a pipe contacting/sliding inside another pipe). Input a positive value to model external 3-D beam-to-beam contact.

5. Input a positive contact radius when modeling internal pipe-to-pipe contact or external 3-D beam-to-beam contact.

6. Rigid surface node. This segment type is only used to apply boundary conditions to rigid target surfaces.

Figure 170.2: TARGE170 Segment Types (p. 639) shows the 3-D segment shapes.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 170.2: TARGE170 Segment Types

For simple rigid target surfaces (including line segments), you can define the target segment elements individually by direct generation. You must first specify the *SHAPE* argument on the **TSHAP** command. When creating cylinders, cones, or spheres through direct generation, you must also define the real constant R1 (and R2 for cones) before creating the element. Real constants R1 and R2 (see Table 170.1: TARGE170 3-D Segment Types, Target Shape Codes, and Nodes (p. 637)) define the dimensions of the target shape.

For general 3-D rigid surfaces, target segment elements can be defined by area meshing (**AMESH**). Set KEYOPT(1) = 0 (the default) to generate low order target elements (3-node triangles and/or 4-node quadrilaterals) for rigid surfaces. Set KEYOPT(1) = 1 to generate target elements with midside nodes (6-node triangles and/or 8-node quadrilaterals).

For 3-D rigid lines, target segment elements can be defined by line meshing (**LMESH**). Set KEYOPT(1) = 0 (the default) to generate low order target elements (2-node straight lines). Set KEYOPT(1) = 1 to generate target elements with midside nodes (3-node parabolas).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also use keypoint meshing (**KMESH**) to generate the pilot node.

If the TARGE170 elements will be created via program meshing (**AMESH**, **LMESH**, or **KMESH** commands), then the **TSHAP** command is ignored and ANSYS chooses the correct shape automatically.

For rigid-to-flexible contact, by default, ANSYS automatically fixes the structural degree of freedom for rigid target nodes if they aren't explicitly constrained (KEYOPT(2) = 0). If you wish, you can override the automatic boundary condition settings by setting KEYOPT(2) = 1 for the target elements. For flexible-to-flexible contact, no special boundary conditions treatment is performed, and the KEYOPT(2) = 0 setting should be used.

For each rigid-flexible contact pair, you can assign only one pilot node to an entire rigid target surface (or none if it is not needed). The pilot node, unlike the other segment types, is used to define the degrees of freedom for the entire target surface. This node can be any of the target surface nodes, but it does not have to be. All possible rigid motions of the target surface will be a combination of a translation and a rotation around the pilot node. The pilot node provides a convenient and powerful way to assign boundary conditions such as rotations, translations, moments, temperature, voltage, and magnetic potential on an entire rigid target surface. By default (KEYOPT(2) = 0), you can assign the boundary conditions only to the pilot node, eliminating the need to assign boundary conditions to individual target nodes, thus reducing the chance of errors. ANSYS will also automatically fix the structural degrees of freedom on the pilot node if they aren't explicitly constrained.

By setting KEYOPT(2) = 1 for the target elements, you can apply boundary conditions on any rigid target nodes rather than only on the pilot node. It is your responsibility to make sure the rigid target surface is not under-constrained or over-constrained. It is still recommended that you apply all boundary conditions on the pilot node, even when KEYOPT(2) = 1.

Considerations for Deformable Target Surfaces

For general deformable surfaces, use the **ESURF** command to overlay the target elements on the boundary of the existing mesh. By default, the command generates a target element with an external surface that has the same shape as the underlying element. You can issue **ESURF**,,,LINE to generate 3-D line or parabola segments on an exterior of selected 3-D elements (e.g., shell edges). Segment types (**TSHAP** command) should not be used prior to **ESURF** when generating target elements on deformable target surfaces.

The cylinder, cone, sphere, point, and pilot node target segments should not be used for deformable target surfaces. However, you can use geometry correction (see below) for deformable target surfaces that represent (or approximately represent) a sphere, cylinder, or cone.

Considerations for Geometry Correction

In general, curved contact and target surfaces can be well approximated by linear or quadratic contact and target elements when the mesh is sufficiently refined. However, in certain circumstances (for example, when linear elements are used or when the midside nodes of quadratic elements do not lie exactly on the initial curved geometry because a third party mesh generator was used), using a faceted surface in place of the true curved geometry can significantly affect the accuracy of contact stresses. An optional geometric correction can be used for two types of curved surfaces (spherical and revolute) via **SECTYPE** and **SECDATA** section commands. The defined geometry correction can be applied to specific contact elements via a section ID (**SECNUM** command). For details, see Geometry Correction for Contact and Target Surfaces in the *Contact Technology Guide*.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Considerations for Fluid Penetration Loading

To model fluid penetration loads, use the **SFE** command to specify the fluid pressure and fluid penetration starting points. For more information, see Applying Fluid Pressure-Penetration Loads in the *Contact Technology Guide*.

Considerations for Thermal Contact Analysis

By default, the temperature is set to the value of TUNIF, and if this has no explicit value the temperature is set to zero. For thermal contact analysis, such as convection and radiation modeling, the behavior of a thermal contact surface (whether a "near-field" or "free" surface) is usually based on the contact status. Contact status affects the behavior of the contact surface as follows:

- If the contact surface is outside the pinball region, its behavior is as a far-field of free surface. In this instance, convection/radiation occurs with the ambient temperature.
- If the contact surface is inside the pinball region, the behavior is as a near-field surface.

However, the thermal contact surface status is ignored if KEYOPT(3) = 1 is set, and the surface is always treated as a free surface (see CONTA173, CONTA174, or CONTA175 for details).

A summary of the element input is given in "TARGE170 Input Summary" (p. 641). A general description of element input is given in Element Input (p. 43).

TARGE170 Input Summary

Nodes
I, J, K, L, M, N, O, P (J - P are not required for all segment types)

Degrees of Freedom
UX, UY, UZ, TEMP, VOLT, MAG (ROTX, ROTY, ROTZ for pilot nodes only)

Real Constants
R1, R2, [the others are defined through the associated CONTA173, CONTA174, CONTA175, CONTA176, or CONTA177 elements]

Material Properties
None

Surface Loads
Pressure, Face 1 (I-J-K-L) (opposite to target normal direction)

Body Loads
None

Special Features

Birth and death
Fluid pressure penetration
Linear perturbation
Nonlinearity
Section definition for geometry correction of spherical and revolute surfaces

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(1)

Element order (used by **AMESH** and **LMESH** commands only):

0 --

Low order elements

1 --

High order elements

KEYOPT(2)

Boundary conditions for rigid target nodes:

0 --

Automatically constrained by ANSYS

1 --

Specified by user

KEYOPT(3)

Behavior of thermal contact surface:

0 --

Based on contact status

1 --

Treated as free-surface

KEYOPT(4)

DOF set to be constrained on the dependent or independent DOF for internally-generated multipoint constraints (MPCs). This option is used for these situations: solid-solid and shell-shell assemblies; surface-based constraints that use a single pilot node for the target element; and rigid target surfaces that use the KEYOPT(2) = 1 setting.

***n* --**

Enter a six digit value that represents the DOF set to be constrained. The first to sixth digits represent ROTZ, ROTY, ROTX, UZ, UY, UX, respectively. The number 1 (one) indicates the DOF is active, and the number 0 (zero) indicates the DOF is not active. For example, 100011 means that UX, UY, and ROTZ will be used in the multipoint constraint. Leading zeros may be omitted; for example, you can enter 11 to indicate that UX and UY are the only active DOF. If KEYOPT(4) = 0 (which is the default) or 111111, all DOF are constrained.

KEYOPT(5)

DOF set to be used in internally-generated multipoint constraints (MPCs), with the MPC algorithm and no separation or bonded behavior (KEYOPT(2) = 2 and KEYOPT(12) = 4, 5, or 6 on the contact element). Note that this key option is not used for surface-based constraints. (See Multipoint Constraints and Assemblies in the *Contact Technology Guide* for more information):

0 --

Automatic constraint type detection (default)

1 --

Solid-solid constraint (no rotational DOFs are constrained)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2 --
Shell-shell constraint (both translational and rotational DOFs are constrained independently). Also used with penalty based shell-shell assembly (KEYOPT(2) = 0 or 1 and KEYOPT(12) = 5 or 6 on the contact element); see Bonded Contact for Shell-Shell Assemblies in the *Contact Technology Guide* for more information.

3 --
Shell-solid constraint - contact normal direction (both translational and rotational DOFs from the contact surface are included in the constraint set; only translational DOFs from the target surface are included in the constraint set).

4 --
Shell-solid constraint - all directions. This option acts the same as KEYOPT(5) = 3 if an intersection is found from the contact normal to the target surface. Otherwise, constraint equations are still built as long as contact node(s) and target segments are inside the pinball region.

5 --
Shell-solid constraint - anywhere inside pinball region. Constraint equations are always built as long as contact node(s) and target segments are inside the pinball region, regardless of whether an intersection exists between the contact normal and the target surface.

Note

When the no separation option (KEYOPT(12) = 4 on the contact element) is used with the MPC approach, only the KEYOPT(5) = 0 and 1 options (auto detection or solid-solid constraint) described above are valid. If the auto detection option is set and the program finds a shell-shell or shell-solid constraint in this situation, the solution will terminate.

KEYOPT(6)
Symmetry condition of a constrained surface. This option is only used for a force-distributed constraint that uses a single pilot node for the target element:

***n* --**
Enter a three digit value that represents the symmetry conditions on the constrained surface. Symmetry is defined with respect to the nodal coordinate system of the pilot node. The first, second, and third digits represent a symmetry condition with respect to the xy, xz, and yz planes, respectively. The number 1 (one) indicates a symmetry condition, and the number 0 (zero) indicates no symmetry condition. For example, KEYOPT(6) = 110 means the force distributed constraint is built on a surface or edge that has symmetry about the xy and xz planes. Leading zeros may be omitted (e.g., KEYOPT(6) = 10 indicates symmetry about the xz plane only).

Note

Keep the following points in mind when using this symmetry condition:

- When a symmetry condition is used, the pilot node must be defined on the symmetry plane/edge.
- KEYOPT(6) = 111 is not valid input.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TARGE170 Output Data

The solution output associated with the element is shown in Table 170.2: TARGE170 Element Output Definitions (p. 644).

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 170.2: TARGE170 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes I, J, and K	Y	Y
ITRGET	Target surface number (assigned by ANSYS)	Y	Y
TSHAP	Segment shape type	Y	Y
ISEG	Segment numbering	1	1
FPRS	Actual applied fluid penetration pressure	Y	Y

1. An internal segment number determined by ANSYS. ISEG for the target element has a different meaning than ISEG reported for the contact element.

You can display or list the actual fluid pressure applied to the target element through several POST1 postprocessing commands, as shown below:

```
PLESOL,CONT,FPRS
PLNSOL,CONT,FPRS
PRESOL,CONT
PRNSOL,CONT
```

Note that only the FPRS (fluid penetration pressure) output item is meaningful when the **PRESOL** and **PRNSOL** commands are used for target elements.

Table 170.3: TARGE170 Item and Sequence Numbers (p. 645) lists output available through the **ETABLE** command using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 170.3: TARGE170 Item and Sequence Numbers (p. 645):

Name
output quantity as defined in the Table 170.2: TARGE170 Element Output Definitions (p. 644)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

I,J,K,L
sequence number for data at nodes I, J, K, L

Table 170.3: TARGE170 Item and Sequence Numbers

Output Quant-ity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
FPRS	SMISC	5	1	2	3	4

TARGE170 Assumptions and Restrictions

- Generally speaking, you should not change real constants R1 or R2, either between load steps or during restart stages; otherwise ANSYS assumes the radii of the primitive segments varies between the load steps. When using direct generation, the real constants for cylinders, cones, and spheres may be defined before the input of the element nodes. If multiple rigid primitives are defined, each having different radii, they must be defined by different target surfaces.
- For each pilot node, ANSYS automatically defines an internal node and an internal constraint equation. The rotational DOF of the pilot node is connected to the translational DOF of the internal node by the internal constraint equation. ANSYS, Inc. recommends against using external constraint equations or coupling on pilot nodes; if you do, conflicts may occur, yielding incorrect results.
- For rotation of a rigid body constrained only by a bonded, rigid-flexible contact pair with a pilot node, use the MPC algorithm or a surface-based constraint as described in Multipoint Constraints and Assemblies in the *Contact Technology Guide*. Penalty-based algorithms can create undesirable rotational energies in this situation.
- When using deformable TARGE170 elements overlaid on SOLID285 elements, make sure that KEYOPT(1) = 0 (the default) if you plan to use **CDWRITE**/**CDREAD** commands to write out model data and later read it back in. Otherwise, the TARGE170 elements will not be generated correctly after the **CDREAD** operation.

TARGE170 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

CONTA171

2-D 2-Node Surface-to-Surface Contact

MP ME ST PR PRN DS EM <> PP EME MFS
Product Restrictions

CONTA171 Element Description

CONTA171 is used to represent contact and sliding between 2-D "target" surfaces (TARGE169) and a deformable surface, defined by this element. The element is applicable to 2-D structural and coupled field contact analyses. This element is located on the surfaces of 2-D solid, shell, or beam elements without midside nodes (such as PLANE13, PLANE55, PLANE182, MATRIX50, and SHELL208). It has the same geometric characteristics as the solid, shell, or beam element face with which it is connected (see Figure 171.1: CONTA171 Geometry (p. 647)). Contact occurs when the element surface penetrates one of the target segment elements (TARGE169) on a specified target surface. Coulomb friction, shear stress friction, user-defined friction with the `USERFRIC` subroutine, and user-defined contact interaction with the `USERINTER` subroutine are allowed. This element also allows separation of bonded contact to simulate interface delamination. See CONTA171 in the *Mechanical APDL Theory Reference* for more details about this element. Other surface-to-surface contact elements (CONTA172, CONTA173, CONTA174) are also available.

Figure 171.1: CONTA171 Geometry

CONTA171 Input Data

The geometry and node locations are shown in Figure 171.1: CONTA171 Geometry (p. 647). The element is defined by two nodes (the underlying solid, shell, or beam element has no midside nodes). If the underlying solid, shell, or beam elements do have midside nodes, use CONTA172. The element x-axis is along the I-J line of the element. The correct node ordering of the contact element is critical for proper detection of contact. The nodes must be ordered such that the target must lie to the right side of the contact element when moving from the first contact element node to the second contact element node as in Figure 171.1: CONTA171 Geometry (p. 647). See Generating Contact Elements in the *Contact Technology Guide* for more information on generating elements automatically using the **ESURF** command.

The 2-D contact surface elements are associated with the 2-D target segment elements (TARGE169) via a shared real constant set. ANSYS looks for contact only between surfaces with the same real constant set. For modeling either rigid-flexible or flexible-flexible contact, one of the deformable surfaces must be represented by a contact surface. See Designating Contact and Target Surfaces in the *Contact Technology Guide* for more information.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If more than one target surface will make contact with the same boundary of solid elements, you must define several contact elements that share the same geometry but relate to separate targets (targets which have different real constant numbers), or you must combine the two target surfaces into one (targets that share the same real constant numbers).

To model proper momentum transfer and energy balance between contact and target surfaces, impact constraints should be used in transient dynamic analysis. See the description of KEYOPT(7) below and the contact element discussion in the *Mechanical APDL Theory Reference* for details.

To model isotropic friction, use the **TB**,FRIC,,,,ISO command. You can define a coefficient of friction that is dependent on temperature, time, normal pressure, sliding distance, or sliding relative velocity by using the **TBFIELD** command along with **TB**,FRIC,,,,ISO. See Contact Friction in the *Material Reference* for more information.

To implement a user-defined friction model, use the **TB**,FRIC command with *TBOPT* = USER to specify friction properties and write a `USERFRIC` subroutine to compute friction forces. See Writing Your Own Friction Law (USERFRIC) in the *Mechanical APDL Contact Technology Guide* for more information on how to use this feature. See also the *Guide to User-Programmable Features* for a detailed description of the `USERFRIC` subroutine.

In addition to the user-defined friction subroutine, the contact interaction subroutine `USERINTER` is available for user-defined interface interactions, including interactions in the normal and tangential directions as well coupled-field interactions. See Defining Your Own Contact Interaction (USERINTER) in the *Mechanical APDL Contact Technology Guide* for more information on how to use this feature. See also the *Guide to User-Programmable Features* for a detailed description of the `USERINTER` subroutine.

To model separation of bonded contact with KEYOPT(12) = 2, 3, 4, 5, or 6, use the **TB** command with the CZM label. See Debonding in the *Contact Technology Guide* for more information.

To model wear at the contact surface, use the **TB** command with the WEAR label. See Contact Surface Wear in the *Contact Technology Guide* for more information.

To model fluid penetration loads, use the **SFE** command to specify the fluid pressure and fluid penetration starting points. For more information, see Applying Fluid Pressure-Penetration Loads in the *Contact Technology Guide*.

This element supports various 2-D stress states, including plane stress, plane strain, and axisymmetric states. The stress state is automatically detected according to the stress state of the underlying element. However, if the underlying element is a superelement, you must use KEYOPT(3) to specify the stress state.

Two types of geometry correction are available for this element: surface smoothing and bolt thread modeling. Surface smoothing is a geometry correction technique that eliminates inaccuracies introduced by linear elements on a curved (circular or nearly circular) contact surface. Bolt thread modeling provides a method for simulating contact between a threaded bolt and bolt hole without having to model the detailed thread geometry. Both of these geometry correction techniques are implemented through section definitions (**SECTYPE**, **SECDATA**, and **SECNUM** commands). For more information, see Geometry Correction for Contact and Target Surfaces in the *Contact Technology Guide*.

A summary of the element input is given in "CONTA171 Input Summary" (p. 649). A general description of element input is given in Element Input (p. 43). For axisymmetric applications see Harmonic Axisymmetric Elements (p. 83).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CONTA171 Input Summary

Nodes

I, J

Degrees of Freedom

UX, UY (if KEYOPT(1) = 0)
UX, UY, TEMP (if KEYOPT(1) = 1)
TEMP (if KEYOPT(1) = 2)
UX, UY, TEMP, VOLT (if KEYOPT(1) = 3)
TEMP, VOLT (if KEYOPT(1) = 4)
UX, UY, VOLT (if KEYOPT(1) = 5)
VOLT (if KEYOPT(1) = 6)
AZ (if KEYOPT(1) = 7)

Real Constants

R1, R2, FKN, FTOLN, ICONT, PINB,
PMAX, PMIN, TAUMAX, CNOF, FKOP, FKT,
COHE, TCC, FHTG, SBCT, RDVF, FWGT,
ECC, FHEG, FACT, DC, SLTO, TNOP,
TOLS, , PPCN, FPAT, COR, STRM
FDMN, FDMT, , , TBND
See Table 171.1: CONTA171 Real Constants (p. 655) for descriptions of the real constants.

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: MU, EMIS

Surface Loads

Pressure, Face 1 (I-J) (opposite to contact normal direction); used for fluid pressure penetration loading. On the **SFE** command use *LKEY* = 1 to specify the pressure values, and use *LKEY* = 2 to specify starting points and penetrating points.
Convection, Face 1 (I-J)
Heat Flux, Face 1 (I-J)

Special Features

Birth and death
Debonding
Fluid pressure penetration
Isotropic friction
Large deflection
Linear perturbation
Nonlinearity
User-defined contact interaction
User-defined friction

KEYOPTs

Presented below is a list of KEYOPTS available for this element. Included are links to sections in the *Contact Technology Guide* where more information is available on a particular topic.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(1)

Selects degrees of freedom:

0 --
UX, UY

1 --
UX, UY, TEMP

2 --
TEMP

3 --
UX, UY, TEMP, VOLT

4 --
TEMP, VOLT

5 --
UX, UY, VOLT

6 --
VOLT

7 --
AZ

KEYOPT(2)

Contact algorithm:

0 --
Augmented Lagrangian (default)

1 --
Penalty function

2 --
Multipoint constraint (MPC); see Multipoint Constraints and Assemblies in the *Contact Technology Guide* for more information

3 --
Lagrange multiplier on contact normal and penalty on tangent

4 --
Pure Lagrange multiplier on contact normal and tangent

KEYOPT(3)

Units of normal contact stiffness:

0 --
FORCE/LENGTH^3 (default)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
FORCE/LENGTH

Note

KEYOPT(3) = 1 is valid only when a penalty-based algorithm is used (KEYOPT(2) = 0 or 1) and the absolute normal contact stiffness value is explicitly specified (that is, a negative value input for real constant FKN).

If superelements are present in a 2-D model, KEYOPT(3) does not control units of normal contact stiffness. Instead, KEYOPT(3) specifies the stress state as follows: KEYOPT(3) = 1 for axisymmetric; KEYOPT(3) = 2 for plane stress/plane strain with unit thickness; KEYOPT(3) = 3 for plane stress with thickness input. (KEYOPT(3) = 0 indicates no superelements.)

KEYOPT(4)

Location of contact detection point:

0 --
On Gauss point (for general cases)

1 --
On nodal point - normal from contact surface

2 --
On nodal point - normal to target surface

3 --
On nodal point - normal from contact surface (projection-based method)

Note

When using the multipoint constraint (MPC) approach to define surface-based constraints, use KEYOPT(4) in the following way: set KEYOPT(4) = 1 for a force-distributed constraint; set KEYOPT(4) = 2 for a rigid surface constraint; set KEYOPT(4) = 3 for a coupling constraint. See Surface-based Constraints for more information.

Note

Certain restrictions apply when the surface projection-based method (KEYOPT(4) = 3) is defined. See Using the Surface Projection Based Contact Method (KEYOPT(4) = 3) for more information.

KEYOPT(5)

CNOF/ICONT Automated adjustment:

0 --
No automated adjustment

1 --
Close gap with auto CNOF

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2 --
Reduce penetration with auto CNOF

3 --
Close gap/reduce penetration with auto CNOF

4 --
Auto ICONT

KEYOPT(6)
Contact stiffness variation (used to enhance stiffness updating when KEYOPT(10) > 0):

0 --
Use default range for stiffness updating

1 --
Make a nominal refinement to the allowable stiffness range

2 --
Make an aggressive refinement to the allowable stiffness range

KEYOPT(7)
Element level time incrementation control / impact constraints:

0 --
No control

1 --
Automatic bisection of increment

2 --
Change in contact predictions made to maintain a reasonable time/load increment

3 --
Change in contact predictions made to achieve the minimum time/load increment whenever a change in contact status occurs

4 --
Use impact constraints for standard or rough contact (KEYOPT(12) = 0 or 1) in a transient dynamic analysis with automatic adjustment of time increment

---

Note

KEYOPT(7) = 2, 3, and 4 include an automatic adjustment of the time increment. This is activated only if the command **SOLCONTROL**,ON,ON was issued prior to the solution.

KEYOPT(8)
Asymmetric contact selection:

0 --
No action

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2 --
ANSYS internally selects which asymmetric contact pair is used at the solution stage (used only when symmetry contact is defined).

KEYOPT(9)
Effect of initial penetration or gap:

0 --

Include both initial geometrical penetration or gap and offset

1 --
Exclude both initial geometrical penetration or gap and offset

2 --
Include both initial geometrical penetration or gap and offset, but with ramped effects

3 --
Include offset only (exclude initial geometrical penetration or gap)

4 --
Include offset only (exclude initial geometrical penetration or gap), but with ramped effects

5 --
Include offset only (exclude initial geometrical penetration or gap) regardless of the initial contact status (near-field or closed)

6 --
Include offset only (exclude initial geometrical penetration or gap), but with ramped effects regardless of the initial contact status (near-field or closed)

Note

The effects of KEYOPT(9) are dependent on settings for other KEYOPTs. The indicated initial gap effect is considered only if KEYOPT(12) = 4 or 5. See the discussion on using KEYOPT(9) in the *Contact Technology Guide* for more information.

KEYOPT(10)
Contact stiffness update:

0 --
Each load step if FKN is redefined during load step (pair based).

2 --
Each iteration based on current mean stress of underlying elements (pair based).

KEYOPT(11)
Beam/Shell thickness effect:

0 --
Exclude

1 --
Include

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(12)

Behavior of contact surface:

0 --
Standard

1 --
Rough

2 --
No separation (sliding permitted)

3 --
Bonded

4 --
No separation (always)

5 --
Bonded (always)

6 --
Bonded (initial contact)

Note

When KEYOPT(12) = 5 or 6 is used with the MPC algorithm to model surface-based constraints, the KEYOPT(12) setting will have an impact on the local coordinate system of the contact element nodes. See Specifying a Local Coordinate System in the *Contact Technology Guide* for more information.

KEYOPT(14)

Behavior of fluid pressure penetration load. KEYOPT(14) is valid only if a fluid pressure penetration load (**SFE**,,,PRES) is applied to the contact element:

0 --
Fluid pressure penetration load varies during iterations (default)

1 --
Fluid pressure penetration load remains constant over the substep

KEYOPT(15)

Effect of contact stabilization damping:

0 --
Damping is activated only in the first load step (default).

1 --
Deactivate automatic damping.

2 --
Damping is activated for all load steps.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3 --
Damping is activated at all times regardless of the contact status of previous substeps.

Note

Stabilization damping is only applied to contact pairs in near-field contact. When KEYOPT(15) = 0, 1, or 2, stabilization damping will not be applied in the current substep if any contact detection point had a closed status in the previous substep. However, when KEYOPT(15) = 3, stabilization damping is always applied as long as the current contact status is near-field. See Applying Contact Stabilization Damping in the *Contact Technology Guide* for more information.

Table 171.1: CONTA171 Real Constants

No.	Name	Description	For more information, see this section in the ***Contact Technology Guide*** . . .
1	R1	Target circle radius	Defining the Target Surface
2	R2	Superelement thickness	Defining the Target Surface
3	FKN	Normal penalty stiffness factor [1] [2]	Determining Contact Stiffness and Penetration
4	FTOLN	Penetration tolerance factor	Determining Contact Stiffness and Penetration
5	ICONT	Initial contact closure	Adjusting Initial Contact Conditions
6	PINB	Pinball region	Determining Contact Status and the Pinball Region or Defining Influence Range (PINB)
7	PMAX	Upper limit of initial allowable penetration	Adjusting Initial Contact Conditions
8	PMIN	Lower limit of initial allowable penetration	Adjusting Initial Contact Conditions
9	TAUMAX	Maximum friction stress [1] [2]	Choosing a Friction Model
10	CNOF	Contact surface offset [1]	Adjusting Initial Contact Conditions
11	FKOP	Contact opening stiffness [1] [2]	Selecting Surface Interaction Models
12	FKT	Tangent penalty stiffness factor [1] [2]	Determining Contact Stiffness

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

No.	Name	Description	For more information, see this section in the ***Contact Technology Guide*** . . .
13	COHE	Contact cohesion	Choosing a Friction Model
14	TCC	Thermal contact conductance [1] [2]	Modeling Conduction
15	FHTG	Frictional heating factor	Modeling Heat Generation Due to Friction
16	SBCT	Stefan-Boltzmann constant	Modeling Radiation
17	RDVF	Radiation view factor [1] [2]	Modeling Radiation
18	FWGT	Heat distribution weighing factor	Modeling Heat Generation Due to Friction (thermal) or Heat Generation Due to Electric Current (electric)
19	ECC	Electric contact conductance [1] [2]	Modeling Surface Interaction
20	FHEG	Joule dissipation weight factor	Heat Generation Due to Electric Current
21	FACT	Static/dynamic ratio	Static and Dynamic Friction Coefficients
22	DC	Exponential decay coefficient	Static and Dynamic Friction Coefficients
23	SLTO	Allowable elastic slip	Using FKT and SLTO
24	TNOP	Maximum allowable tensile contact pressure	Chattering Control Parameters
25	TOLS	Target edge extension factor	Selecting Location of Contact Detection
27	PPCN	Pressure penetration criterion [1] [2]	Specifying a Pressure Penetration Criterion
28	FPAT	Fluid penetration acting time	Specifying a Fluid Penetration Acting Time
29	COR	Coefficient of restitution	Impact Between Rigid Bodies
30	STRM	Load step number for ramping penetration	Adjusting Initial Contact Conditions
31	FDMN	Normal stabilization damping factor [1] [2]	Applying Contact Stabilization Damping
32	FDMT	Tangential stabilization damping factor [1] [2]	Applying Contact Stabilization Damping
35	TBND	Critical bonding temperature [1] [2]	Using TBND

1. This real constant can be defined as a function of certain primary variables.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. This real constant can be defined by the user subroutine USERCNPROP.F.

CONTA171 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 171.2: CONTA171 Element Output Definitions (p. 657)

A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 171.2: CONTA171 Element Output Definitions (p. 657) gives element output. In the results file, the nodal results are obtained from its closest integration point.

Table 171.2: CONTA171 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes I, J	Y	Y
XC, YC	Location where results are reported	Y	5
TEMP	Temperatures T(I), T(J)	Y	Y
LENGTH	Element length	Y	-
VOLU	AREA	Y	Y
NPI	Number of integration points	Y	-
ITRGET	Target surface number (assigned by ANSYS)	Y	-
ISOLID	Underlying solid, shell, or beam element number	Y	-
CONT:STAT	Current contact statuses	1	1
OLDST	Old contact statuses	1	1
NX, NY	Surface normal vector components	Y	-
ISEG	Current contacting target element number	Y	Y
OLDSEG	Underlying old target number	Y	-
CONT:PENE	Current penetration (gap = 0; penetration = positive value)	Y	Y
CONT:GAP	Current gap (gap = negative value; penetration = 0)	Y	Y
NGAP	New or current gap at current converged substep (gap = negative value; penetration = positive value)	Y	-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
OGAP	Old gap at previously converged substep (gap = negative value; penetration = positive value)	Y	-
IGAP	Initial gap at start of current substep (gap = negative value; penetration = positive value)	Y	Y
GGAP	Geometric gap at current converged substep (gap = negative value; penetration = positive value)	-	Y
CONT:PRES	Normal contact pressure	Y	Y
CONT:SFRIC	Tangential contact stress	Y	Y
KN	Current normal contact stiffness (Force/Length^3)	Y	Y
KT	Current tangent contact stiffness (Force/Length^3)	Y	Y
MU	Friction coefficient	Y	Y
CONT:SLIDE	Total accumulated sliding (algebraic sum)	3	3
ASLIDE	Total accumulated sliding (absolute sum)	3	3
TOLN	Penetration tolerance	Y	Y
CONT:STOTAL	Total stress, SQRT (PRES**2+SFRIC**2)	Y	Y
FDDIS	Frictional energy dissipation	6	6
ELSI	Total elastic slip distance	-	Y
PLSI	Total accumulated plastic slip due to frictional sliding	-	Y
GSLID	Algebraic sum sliding	-	7
VREL	Sliding velocity (slip rate)	-	Y
DBA	Penetration variation	Y	Y
PINB	Pinball Region	-	Y
CONT:CNOS	Total number of contact status changes during substep	Y	Y
TNOP	Maximum allowable tensile contact pressure	Y	Y
SLTO	Allowable elastic slip	Y	Y
CAREA	Contacting area	-	Y
CONT:FPRS	Actual applied fluid penetration pressure	-	Y
FSTART	Fluid penetration starting time	-	Y
DTSTART	Load step time during debonding	Y	Y
DPARAM	Debonding parameter	Y	Y
DENERI	Energy released due to separation in normal direction - mode I debonding	Y	Y
DENERII	Energy released due to separation in tangential direction - mode II debonding	Y	Y
CNFX	Contact element force-X component	-	4
CNFY	Contact element force-Y component	-	Y
SDAMP	Stabilization damping coefficient	-	Y
CONV	Convection coefficient	Y	Y
RAC	Radiation coefficient	Y	Y
TCC	Conductance coefficient	Y	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
TEMPS	Temperature at contact point	Y	Y
TEMPT	Temperature at target surface	Y	Y
FXCV	Heat flux due to convection	Y	Y
FXRD	Heat flux due to radiation	Y	Y
FXCD	Heat flux due to conductance	Y	Y
CONT:FLUX	Total heat flux at contact surface	Y	Y
FXNP	Flux input	-	Y
CNFH	Contact element heat flow	-	Y
JCONT	Contact current density (Current/Unit Area)	Y	Y
CCONT	Contact charge density (Charge/Unit Area)	Y	Y
HJOU	Contact power/area	Y	Y
ECURT	Current per contact element	-	Y
ECHAR	Charge per contact element	-	Y
ECC	Electric contact conductance (for electric current DOF), or electric contact capacitance per unit area (for piezoelectric or electrostatic DOFs)	Y	Y
VOLTS	Voltage on contact nodes	Y	Y
VOLTT	Voltage on associated target	Y	Y
WEARX, WEARY	Wear correction - X and Y components	-	Y

1. The possible values of *STAT* and *OLDST* are:

   0 = Open and not near contact
   1 = Open but near contact
   2 = Closed and sliding
   3 = Closed and sticking

2. ANSYS will evaluate model to detect initial conditions.

3. Only accumulates the sliding for sliding and closed contact (STAT = 2,3).

4. Contact element forces are defined in the global Cartesian system.

5. Available only at centroid as a ***GET** item.

6. FDDIS = (contact friction stress)*(sliding distance of substep)/(time increment of substep)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Accumulated sliding distance for near-field, sliding, and closed contact (STAT = 1,2,3).

Note

If **ETABLE** is used for the CONT items, the reported data is averaged across the element.

Note

Contact results (including all element results) are generally not reported for elements that have a status of "open and not near contact" (far-field).

Table 171.3: CONTA171 Item and Sequence Numbers (p. 660) lists output available through the **ETABLE** command using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 171.3: CONTA171 Item and Sequence Numbers (p. 660):

Name
output quantity as defined in the Table 171.2: CONTA171 Element Output Definitions (p. 657)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J
sequence number for data at nodes I, J

Table 171.3: CONTA171 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
PRES	SMISC	5	1	2
SFRIC	SMISC	-	3	4
FLUX	SMISC	-	6	7
FDDIS	SMISC	-	8	9
FXCV	SMISC	-	10	11
FXRD	SMISC	-	12	13
FXCD	SMISC	-	14	15
FXNP	SMISC	-	16	17
JCONT	SMISC	-	18	19
CCONT	SMISC	-	18	19
HJOU	SMISC	-	20	21
STAT[1]	NMISC	19	1	2
OLDST	NMISC	-	3	4
PENE[2]	NMISC	-	5	6

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
DBA	NMISC	-	7	8
SLIDE	NMISC	-	9	10
KN	NMISC	-	11	12
KT	NMISC	-	13	14
TOLN	NMISC	-	15	16
IGAP	NMISC	-	17	18
PINB	NMISC	20	-	-
CNFX	NMISC	21	-	-
CNFY	NMISC	22	-	-
ISEG	NMISC	-	23	24
ASLIDE	NMISC	-	25	26
CAREA	NMISC	27	-	-
MU	NMISC	-	29	30
DTSTART	NMISC	-	31	32
DPARAM	NMISC	-	33	34
FPRS	NMISC	-	35	36
TEMPS	NMISC	-	37	38
TEMPT	NMISC	-	39	40
CONV	NMISC	-	41	42
RAC	NMISC	-	43	44
TCC	NMISC	-	45	46
CNFH	NMISC	47	-	-
ECURT	NMISC	48	-	-
ECHAR	NMISC	48	-	-
ECC	NMISC	-	49	50
VOLTS	NMISC	-	51	52
VOLTT	NMISC	-	53	54
CNOS	NMISC	-	55	56
TNOP	NMISC	-	57	58
SLTO	NMISC	-	59	60
ELSI	NMISC	-	67	68
DENERI	NMISC	-	69	70
DENERII	NMISC	-	71	72
FSTART	NMISC	-	73	74
GGAP	NMISC	-	75	76
VREL	NMISC	-	77	78

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
SDAMP	NMISC	-	79	80
PLSI	NMISC	-	81	82
GSLID	NMISC	-	83	84
WEARX	NMISC	-	85	86
WEARY	NMISC	-	87	88

1. Element Status = highest value of status of integration points within the element

2. Penetration = positive value, gap = negative value

You can display or list contact results through several POST1 postprocessor commands. The contact specific items for the **PLNSOL**, **PLESOL**, **PRNSOL**, and **PRESOL** commands are listed below:

STAT	Contact status
PENE	Contact penetration
PRES	Contact pressure
SFRIC	Contact friction stress
STOT	Contact total stress (pressure plus friction)
SLIDE	Contact sliding distance
GAP	Contact gap distance
FLUX	Total heat flux at contact surface
CNOS	Total number of contact status changes during substep
FPRS	Actual applied fluid penetration pressure

CONTA171 Assumptions and Restrictions

- The 2-D contact element must be defined in an X-Y plane and the Y-axis must be the axis of symmetry for axisymmetric analyses.
- An axisymmetric structure should be modeled in the +X quadrants.
- This 2-D contact element works with any 3-D elements in your model.
- Do not use this element in any model that contains axisymmetric harmonic elements.
- Node numbering must coincide with the external surface of the underlying solid, shell, or beam element, or with the original elements comprising the superelement.
- This element is nonlinear and requires a full Newton iterative solution, regardless of whether large or small deflections are specified. An exception to this is when MPC bonded contact is specified (KEYOPT(2) = 2 and KEYOPT(12) = 5 or 6).
- The normal contact stiffness factor (FKN) must not be so large as to cause numerical instability.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- FTOLN, PINB, and FKOP can be changed between load steps or during restart stages.
- You can use this element in nonlinear static or nonlinear full transient analyses. In addition, you can use it in modal analyses, eigenvalue buckling analyses, and harmonic analyses. For these analysis types, the program assumes that the initial status of the element (that is, the status at the completion of the static prestress analysis, if any) does not change.
- When nodal detection is used and the contact node is on the axis of symmetry in an axisymmetric analysis, the contact pressure on that node is not accurate since the area of the node is zero. The contact force is accurate in this situation.

CONTA171 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The MU material property is not allowed in the Nonlinear Thermal product.
- The birth and death special feature is not allowed in the Nonlinear Structural and Nonlinear Thermal products.

ANSYS Structural

- The VOLT DOF (KEYOPT(1) = 3 through 6) is not allowed.
- The AZ DOF (KEYOPT(1) = 7) is not allowed.

ANSYS Mechanical

- The AZ DOF (KEYOPT(1) = 7) is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

CONTA172

2-D 3-Node Surface-to-Surface Contact

MP ME ST PR PRN DS EM <> PP EME MFS
Product Restrictions

CONTA172 Element Description

CONTA172 represents contact and sliding between 2-D "target" surfaces (TARGE169) and a deformable surface, defined by this element. The element is applicable to 2-D structural and coupled field contact analyses. This element is located on the surfaces of 2-D solid elements with midside nodes (PLANE35, PLANE77, PLANE53, PLANE121, PLANE183, SHELL209, PLANE223, PLANE230, or MATRIX50). It has the same geometric characteristics as the solid element face with which it is connected (see Figure 172.1: CONTA172 Geometry (p. 665)). Contact occurs when the element surface penetrates one of the target segment elements (TARGE169) on a specified target surface. Coulomb friction, shear stress friction, user-defined friction with the `USERFRIC` subroutine, and user-defined contact interaction with the `USERINTER` subroutine are allowed. This element also allows separation of bonded contact to simulate interface delamination. See CONTA172 in the *Mechanical APDL Theory Reference* for more details about this element. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for a discussion concerning midside nodes. Other surface-to-surface contact elements (CONTA171, CONTA173, CONTA174) are also available.

Figure 172.1: CONTA172 Geometry

CONTA172 Input Data

The geometry and node locations are shown in Figure 172.1: CONTA172 Geometry (p. 665). The element is defined by three nodes (the underlying solid element has midside nodes). If the underlying solid elements do not have midside nodes, use CONTA171 (you may still use CONTA172 but you must drop the midside nodes). The element x-axis is along the I-J line of the element. The correct node ordering of the contact element is critical for proper detection of contact. The nodes must be ordered such that the target must lie to the right side of the contact element when moving from the first contact element node to the second contact element node as in Figure 172.1: CONTA172 Geometry (p. 665). See Generating Contact Elements in the *Contact Technology Guide* for more information on generating elements automatically using the **ESURF** command.

The 2-D contact surface elements are associated with the 2-D target segment elements (TARGE169) via a shared real constant set. ANSYS looks for contact only between surfaces with the same real constant set. For either rigid-flexible or flexible-flexible contact, one of the deformable surfaces must be represented by a contact surface. See Designating Contact and Target Surfaces in the *Contact Technology Guide* for more information.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If more than one target surface will make contact with the same boundary of solid elements, you must define several contact elements that share the same geometry but relate to separate targets (targets with different real constant numbers), or you must combine the two target surfaces into one (both having the same real constant number).

To model proper momentum transfer and energy balance between contact and target surfaces, impact constraints should be used in transient dynamic analysis. See the description of KEYOPT(7) below and the contact element discussion in the *Mechanical APDL Theory Reference* for details.

To model isotropic friction, use the **TB**,FRIC,,,,ISO command. You can define a coefficient of friction that is dependent on temperature, time, normal pressure, sliding distance, or sliding relative velocity by using the **TBFIELD** command along with **TB**,FRIC,,,,ISO. See Contact Friction in the *Material Reference* for more information.

To implement a user-defined friction model, use the **TB**,FRIC command with *TBOPT* = USER to specify friction properties and write a `USERFRIC` subroutine to compute friction forces. See Writing Your Own Friction Law (USERFRIC) in the *Mechanical APDL Contact Technology Guide* for more information on how to use this feature. See also the *Guide to User-Programmable Features* for a detailed description of the `USERFRIC` subroutine.

In addition to the user-defined friction subroutine, the contact interaction subroutine `USERINTER` is available for user-defined interface interactions, including interactions in the normal and tangential directions as well coupled-field interactions. See Defining Your Own Contact Interaction (USERINTER) in the *Mechanical APDL Contact Technology Guide* for more information on how to use this feature. See also the *Guide to User-Programmable Features* for a detailed description of the `USERINTER` subroutine.

To model separation of bonded contact with KEYOPT(12) = 2, 3, 4, 5, or 6, use the **TB** command with the CZM label. See Debonding in the *Contact Technology Guide* for more information.

To model wear at the contact surface, use the **TB** command with the WEAR label. See Contact Surface Wear in the *Contact Technology Guide* for more information.

To model fluid penetration loads, use the **SFE** command to specify the fluid pressure and fluid penetration starting points. For more information, see Applying Fluid Pressure-Penetration Loads in the *Contact Technology Guide*.

This element supports various 2-D stress states, including plane stress, plane strain, and axisymmetric states. The stress state is automatically detected according to the stress state of the underlying element. However, if the underlying element is a superelement, you must use KEYOPT(3) to specify the stress state.

Two types of geometry correction are available for this element: surface smoothing and bolt thread modeling. Surface smoothing is a geometry correction technique that eliminates inaccuracies introduced by linear elements on a curved (circular or nearly circular) contact surface. Bolt thread modeling provides a method for simulating contact between a threaded bolt and bolt hole without having to model the detailed thread geometry. Both of these geometry correction techniques are implemented through section definitions (**SECTYPE**, **SECDATA**, and **SECNUM** commands). For more information, see Geometry Correction for Contact and Target Surfaces in the *Contact Technology Guide*.

A summary of the element input is given in "CONTA172 Input Summary" (p. 667). A general description of element input is given in Element Input (p. 43).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CONTA172 Input Summary

Nodes

I, J, K

Degrees of Freedom

UX, UY (if KEYOPT(1) = 0)
UX, UY, TEMP (if KEYOPT(1) = 1)
TEMP (if KEYOPT(1) = 2)
UX, UY, TEMP, VOLT (if KEYOPT(1) = 3)
TEMP, VOLT (if KEYOPT(1) = 4)
UX, UY, VOLT (if KEYOPT(1) = 5)
VOLT (if KEYOPT(1) = 6)
AZ (if KEYOPT(1) = 7)

Real Constants

R1, R2, FKN, FTOLN, ICONT, PINB,
PMAX, PMIN, TAUMAX, CNOF, FKOP, FKT,
COHE, TCC, FHTG, SBCT, RDVF, FWGT,
ECC, FHEG, FACT, DC, SLTO, TNOP,
TOLS, , PPCN, FPAT, COR, STRM
FDMN, FDMT, , , TBND
See Table 172.1: CONTA172 Real Constants (p. 673) for descriptions of the real constants.

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: MU, EMIS

Surface Loads

Pressure, Face 1 (I-J) (opposite to contact normal direction); used for fluid pressure penetration loading. On the **SFE** command use *LKEY* = 1 to specify the pressure values, and use *LKEY* = 2 to specify starting points and penetrating points.
Convection, Face 1 (I-J-K)
Heat Flux, Face 1 (I-J-K)

Special Features

Birth and death
Debonding
Fluid pressure penetration
Isotropic friction
Large deflection
Linear perturbation
Nonlinearity
Section definition for geometry correction of spherical and revolute surfaces
User-defined contact interaction
User-defined friction

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPTs

Presented below is a list of KEYOPTS available for this element. Included are links to sections in the *Contact Technology Guide* where more information is available on a particular topic.

KEYOPT(1)

Selects degrees of freedom:

0 --
UX, UY

1 --
UX, UY, TEMP

2 --
TEMP

3 --
UX, UY, TEMP, VOLT

4 --
TEMP, VOLT

5 --
UX, UY, VOLT

6 --
VOLT

7 --
AZ

KEYOPT(2)

Contact algorithm:

0 --
Augmented Lagrangian (default)

1 --
Penalty function

2 --
Multipoint constraint (MPC); see Multipoint Constraints and Assemblies in the *Contact Technology Guide* for more information

3 --
Lagrange multiplier on contact normal and penalty on tangent

4 --
Pure Lagrange multiplier on contact normal and tangent

KEYOPT(3)

Units of normal contact stiffness:

0 --
FORCE/LENGTH^3 (default)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
FORCE/LENGTH

Note

KEYOPT(3) = 1 is valid only when a penalty-based algorithm is used (KEYOPT(2) = 0 or 1) and the absolute normal contact stiffness value is explicitly specified (that is, a negative value input for real constant FKN).

If superelements are present in a 2-D model, KEYOPT(3) does not control units of normal contact stiffness. Instead, KEYOPT(3) specifies the stress state as follows: KEYOPT(3) = 1 for axisymmetric; KEYOPT(3) = 2 for plane stress/plane strain with unit thickness; KEYOPT(3) = 3 for plane stress with thickness input. (KEYOPT(3) = 0 indicates no superelements.)

KEYOPT(4)

Location of contact detection point:

0 --
On Gauss point (for general cases)

1 --
On nodal point - normal from contact surface

2 --
On nodal point - normal to target surface

3 --
On nodal point - normal from contact surface (projection-based method)

Note

When using the multipoint constraint (MPC) approach to define surface-based constraints, use KEYOPT(4) in the following way: set KEYOPT(4) = 1 for a force-distributed constraint; set KEYOPT(4) = 2 for a rigid surface constraint; set KEYOPT(4) = 3 for a coupling constraint. See Surface-based Constraints for more information.

Note

Certain restrictions apply when the surface projection-based method (KEYOPT(4) = 3) is defined. See Using the Surface Projection Based Contact Method (KEYOPT(4) = 3) for more information.

KEYOPT(5)

CNOF/ICONT Automated adjustment:

0 --
No automated adjustment

1 --
Close gap with auto CNOF

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2 --
Reduce penetration with auto CNOF

3 --
Close gap/reduce penetration with auto CNOF

4 --
Auto ICONT

KEYOPT(6)
Contact stiffness variation (used to enhance stiffness updating when KEYOPT(10) > 0):

0 --
Use default range for stiffness updating

1 --
Make a nominal refinement to the allowable stiffness range

2 --
Make an aggressive refinement to the allowable stiffness range

KEYOPT(7)
Element level time incrementation control / impact constraints:

0 --
No control

1 --
Automatic bisection of increment

2 --
Change in contact predictions made to maintain a reasonable time/load increment

3 --
Change in contact predictions made to achieve the minimum time/load increment whenever a change in contact status occurs

4 --
Use impact constraints for standard or rough contact (KEYOPT(12) = 0 or 1) in a transient dynamic analysis with automatic adjustment of time increment

---

Note

KEYOPT(7) = 2, 3, and 4 include an automatic adjustment of the time increment. This is activated only if the command **SOLCONTROL**,ON,ON was issued prior to the solution.

KEYOPT(8)
Asymmetric contact selection:

0 --
No action

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2 --
ANSYS internally selects which asymmetric contact pair is used at the solution stage (used only when symmetry contact is defined).

KEYOPT(9)
Effect of initial penetration or gap:

0 --

Include both initial geometrical penetration or gap and offset

1 --
Exclude both initial geometrical penetration or gap and offset

2 --
Include both initial geometrical penetration or gap and offset, but with ramped effects

3 --
Include offset only (exclude initial geometrical penetration or gap)

4 --
Include offset only (exclude initial geometrical penetration or gap), but with ramped effects

5 --
Include offset only (exclude initial geometrical penetration or gap) regardless of the initial contact status (near-field or closed)

6 --
Include offset only (exclude initial geometrical penetration or gap), but with ramped effects regardless of the initial contact status (near-field or closed)

Note

The effects of KEYOPT(9) are dependent on settings for other KEYOPTs. The indicated initial gap effect is considered only if KEYOPT(12) = 4 or 5. See the discussion on using KEYOPT(9) in the *Contact Technology Guide* for more information.

KEYOPT(10)
Contact stiffness update:

0 --
Each load step if FKN is redefined during load step (pair based).

2 --
Each iteration based on current mean stress of underlying elements (pair based).

KEYOPT(11)
Beam/Shell thickness effect:

0 --
Exclude

1 --
Include

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(12)

Behavior of contact surface:

0 --
Standard

1 --
Rough

2 --
No separation (sliding permitted)

3 --
Bonded

4 --
No separation (always)

5 --
Bonded (always)

6 --
Bonded (initial contact)

Note

When KEYOPT(12) = 5 or 6 is used with the MPC algorithm to model surface-based constraints, the KEYOPT(12) setting will have an impact on the local coordinate system of the contact element nodes. See Specifying a Local Coordinate System in the *Contact Technology Guide* for more information.

KEYOPT(14)

Behavior of fluid pressure penetration load. KEYOPT(14) is valid only if a fluid pressure penetration load (**SFE**,,,PRES) is applied to the contact element:

0 --
Fluid pressure penetration load varies during iterations (default)

1 --
Fluid pressure penetration load remains constant over the substep

KEYOPT(15)

Effect of contact stabilization damping:

0 --
Damping is activated only in the first load step (default).

1 --
Deactivate automatic damping.

2 --
Damping is activated for all load steps.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3 --
Damping is activated at all times regardless of the contact status of previous substeps.

Note

Stabilization damping is only applied to contact pairs in near-field contact. When KEYOPT(15) = 0, 1, or 2, stabilization damping will not be applied in the current substep if any contact detection point had a closed status in the previous substep. However, when KEYOPT(15) = 3, stabilization damping is always applied as long as the current contact status is near-field. See Applying Contact Stabilization Damping in the *Contact Technology Guide* for more information.

Table 172.1: CONTA172 Real Constants

No.	Name	Description	For more information, see this section in the ***Contact Technology Guide*** . . .
1	R1	Target circle radius	Defining the Target Surface
2	R2	Superelement thickness	Defining the Target Surface
3	FKN	Normal penalty stiffness factor [1] [2]	Determining Contact Stiffness and Penetration
4	FTOLN	Penetration tolerance factor	Determining Contact Stiffness and Penetration
5	ICONT	Initial contact closure	Adjusting Initial Contact Conditions
6	PINB	Pinball region	Determining Contact Status and the Pinball Region or Defining Influence Range (PINB)
7	PMAX	Upper limit of initial allowable penetration	Adjusting Initial Contact Conditions
8	PMIN	Lower limit of initial allowable penetration	Adjusting Initial Contact Conditions
9	TAUMAX	Maximum friction stress [1] [2]	Choosing a Friction Model
10	CNOF	Contact surface offset [1]	Adjusting Initial Contact Conditions
11	FKOP	Contact opening stiffness [1] [2]	Selecting Surface Interaction Models
12	FKT	Tangent penalty stiffness factor [1] [2]	Determining Contact Stiffness

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

No.	Name	Description	For more information, see this section in the *Contact Technology Guide* . . .
13	COHE	Contact cohesion	Choosing a Friction Model
14	TCC	Thermal contact conductance [1] [2]	Modeling Conduction
15	FHTG	Frictional heating factor	Modeling Heat Generation Due to Friction
16	SBCT	Stefan-Boltzmann constant	Modeling Radiation
17	RDVF	Radiation view factor [1] [2]	Modeling Radiation
18	FWGT	Heat distribution weighing factor	Modeling Heat Generation Due to Friction (thermal) or Heat Generation Due to Electric Current (electric)
19	ECC	Electric contact conductance [1] [2]	Modeling Surface Interaction
20	FHEG	Joule dissipation weight factor	Heat Generation Due to Electric Current
21	FACT	Static/dynamic ratio	Static and Dynamic Friction Coefficients
22	DC	Exponential decay coefficient	Static and Dynamic Friction Coefficients
23	SLTO	Allowable elastic slip	Using FKT and SLTO
24	TNOP	Maximum allowable tensile contact pressure	Chattering Control Parameters
25	TOLS	Target edge extension factor	Selecting Location of Contact Detection
27	PPCN	Pressure penetration criterion [1] [2]	Specifying a Pressure Penetration Criterion
28	FPAT	Fluid penetration acting time	Specifying a Fluid Penetration Acting Time
29	COR	Coefficient of restitution	Impact Between Rigid Bodies
30	STRM	Load step number for ramping penetration	Adjusting Initial Contact Conditions
31	FDMN	Normal stabilization damping factor [1] [2]	Applying Contact Stabilization Damping
32	FDMT	Tangential stabilization damping factor [1] [2]	Applying Contact Stabilization Damping
35	TBND	Critical bonding temperature [1] [2]	Using TBND

1. This real constant can be defined as a function of certain primary variables.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. This real constant can be defined by the user subroutine USERCNPROP.F.

CONTA172 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 172.2: CONTA172 Element Output Definitions (p. 675)

A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 172.2: CONTA172 Element Output Definitions (p. 675) gives element output. In the results file, the nodal results are obtained from its closest integration point.

Table 172.2: CONTA172 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes I, J	Y	Y
XC, YC	Location where results are reported	Y	5
TEMP	Temperatures T(I), T(J)	Y	Y
LENGTH	Element length	Y	-
VOLU	AREA	Y	Y
NPI	Number of integration points	Y	-
ITRGET	Target surface number (assigned by ANSYS)	Y	-
ISOLID	Underlying solid, shell, or beam element number	Y	-
CONT:STAT	Current contact statuses	1	1
OLDST	Old contact statuses	1	1
NX, NY	Surface normal vector components	Y	-
ISEG	Current contacting target element number	Y	Y
OLDSEG	Underlying old target number	Y	-
CONT:PENE	Current penetration (gap = 0; penetration = positive value)	Y	Y
CONT:GAP	Current gap (gap = negative value; penetration = 0)	Y	Y
NGAP	New or current gap at current converged substep (gap = negative value; penetration = positive value)	Y	-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
OGAP	Old gap at previously converged substep (gap = negative value; penetration = positive value)	Y	-
IGAP	Initial gap at start of current substep (gap = negative value; penetration = positive value)	Y	Y
GGAP	Geometric gap at current converged substep (gap = negative value; penetration = positive value)	-	Y
CONT:PRES	Normal contact pressure	Y	Y
CONT:SFRIC	Tangential contact stress	Y	Y
KN	Current normal contact stiffness (Force/Length^3)	Y	Y
KT	Current tangent contact stiffness (Force/Length^3)	Y	Y
MU	Friction coefficient	Y	Y
CONT:SLIDE	Total accumulated sliding (algebraic sum)	3	3
ASLIDE	Total accumulated sliding (absolute sum)	3	3
TOLN	Penetration tolerance	Y	Y
CONT:STOTAL	Total stress, SQRT (PRES**2+SFRIC**2)	Y	Y
FDDIS	Frictional energy dissipation	6	6
ELSI	Total elastic slip distance	-	Y
PLSI	Total accumulated plastic slip due to frictional sliding	-	Y
GSLID	Algebraic sum sliding	-	7
VREL	Sliding velocity (slip rate)	-	Y
DBA	Penetration variation	Y	Y
PINB	Pinball Region	-	Y
CONT:CNOS	Total number of contact status changes during substep	Y	Y
TNOP	Maximum allowable tensile contact pressure	Y	Y
SLTO	Allowable elastic slip	Y	Y
CAREA	Contacting area	-	Y
CONT:FPRS	Actual applied fluid penetration pressure	-	Y
FSTART	Fluid penetration starting time	-	Y
DTSTART	Load step time during debonding	Y	Y
DPARAM	Debonding parameter	Y	Y
DENERI	Energy released due to separation in normal direction - mode I debonding	Y	Y
DENERII	Energy released due to separation in tangential direction - mode II debonding	Y	Y
CNFX	Contact element force - X component	-	4
CNFY	Contact element force - Y component	-	Y
SDAMP	Stabilization damping coefficient	-	Y
CONV	Convection coefficient	Y	Y
RAC	Radiation coefficient	Y	Y
TCC	Conductance coefficient	Y	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
TEMPS	Temperature at contact point	Y	Y
TEMPT	Temperature at target surface	Y	Y
FXCV	Heat flux due to convection	Y	Y
FXRD	Heat flux due to radiation	Y	Y
FXCD	Heat flux due to conductance	Y	Y
CONT:FLUX	Total heat flux at contact surface	Y	Y
FXNP	Flux input	-	Y
CNFH	Contact element heat flow	-	Y
JCONT	Contact current density (Current/Unit Area)	Y	Y
CCONT	Contact charge density (Charge/Unit Area)	Y	Y
HJOU	Contact power/area	Y	Y
ECURT	Current per contact element	-	Y
ECHAR	Charge per contact element	-	Y
ECC	Electric contact conductance (for electric current DOF), or electric contact capacitance per unit area (for piezoelectric or electrostatic DOFs)	Y	Y
VOLTS	Voltage on contact nodes	Y	Y
VOLTT	Voltage on associated target	Y	Y
WEARX, WEARY	Wear correction - X and Y components	-	Y

1. The possible values of STAT and OLDST are:

   0 = Open and not near contact
   1 = Open but near contact
   2 = Closed and sliding
   3 = Closed and sticking

2. ANSYS will evaluate model to detect initial conditions.

3. Only accumulates the sliding for sliding and closed contact (STAT = 2,3).

4. Contact element forces are defined in the global Cartesian system.

5. Available only at centroid as a ***GET** item.

6. FDDIS = (contact friction stress)*(sliding distance of substep)/(time increment of substep)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Accumulated sliding distance for near-field, sliding, and closed contact (STAT = 1,2,3).

Note

If **ETABLE** is used for the CONT items, the reported data is averaged across the element.

Note

Contact results (including all element results) are generally not reported for elements that have a status of "open and not near contact" (far-field).

Table 172.3: CONTA172 Item and Sequence Numbers (p. 678) lists output available through the **ETABLE** command using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 172.3: CONTA172 Item and Sequence Numbers (p. 678):

Name
output quantity as defined in the Table 172.2: CONTA172 Element Output Definitions (p. 675)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J
sequence number for data at nodes I, J

Table 172.3: CONTA172 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
PRES	SMISC	5	1	2
SFRIC	SMISC	-	3	4
FLUX	SMISC	-	6	7
FDDIS	SMISC	-	8	9
FXCV	SMISC	-	10	11
FXRD	SMISC	-	12	13
FXCD	SMISC	-	14	15
FXNP	SMISC	-	16	17
JCONT	SMISC	-	18	19
CCONT	SMISC	-	18	19
HJOU	SMISC	-	20	21
STAT[1]	NMISC	19	1	2
OLDST	NMISC	-	3	4
PENE[2]	NMISC	-	5	6

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
DBA	NMISC	-	7	8
SLIDE	NMISC	-	9	10
KN	NMISC	-	11	12
KT	NMISC	-	13	14
TOLN	NMISC	-	15	16
IGAP	NMISC	-	17	18
PINB	NMISC	20	-	-
CNFX	NMISC	21	-	-
CNFY	NMISC	22	-	-
ISEG	NMISC	-	23	24
ASLIDE	NMISC	-	25	26
CAREA	NMISC	27	-	-
MU	NMISC	-	29	30
DTSTART	NMISC	-	31	32
DPARAM	NMISC	-	33	34
FPRS	NMISC	-	35	36
TEMPS	NMISC	-	37	38
TEMPT	NMISC	-	39	40
CONV	NMISC	-	41	42
RAC	NMISC	-	43	44
TCC	NMISC	-	45	46
CNFH	NMISC	47	-	-
ECURT	NMISC	48	-	-
ECHAR	NMISC	48	-	-
ECC	NMISC	-	49	50
VOLTS	NMISC	-	51	52
VOLTT	NMISC	-	53	54
CNOS	NMISC	-	55	56
TNOP	NMISC	-	57	58
SLTO	NMISC	-	59	60
ELSI	NMISC	-	67	68
DENERI	NMISC	-	69	70
DENERII	NMISC	-	71	72
FSTART	NMISC	-	73	74
GGAP	NMISC	-	75	76
VREL	NMISC	-	77	78

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
SDAMP	NMISC	-	79	80
PLSI	NMISC	-	81	82
GSLID	NMISC	-	83	84
WEARX	NMISC	-	85	86
WEARY	NMISC	-	87	88

1. Element Status = highest value of status of integration points within the element
2. Penetration = positive value, gap = negative value
3. Contact element forces are defined in the global Cartesian system

You can display or list contact results through several POST1 postprocessor commands. The contact specific items for the **PLNSOL**, **PLESOL**, **PRNSOL**, and **PRESOL** commands are listed below:

STAT	Contact status
PENE	Contact penetration
PRES	Contact pressure
SFRIC	Contact friction stress
STOT	Contact total stress (pressure plus friction)
SLIDE	Contact sliding distance
GAP	Contact gap distance
FLUX	Total heat flux at contact surface
CNOS	Total number of contact status changes during substep
FPRS	Actual applied fluid penetration pressure

CONTA172 Assumptions and Restrictions

- The 2-D contact element must be defined in an X-Y plane and the Y-axis must be the axis of symmetry for axisymmetric analyses.
- An axisymmetric structure should be modeled in the +X quadrants.
- This 2-D contact element works with any 3-D elements in your model.
- Do not use this element in any model that contains axisymmetric harmonic elements.
- Node numbering must coincide with the external surface of the underlying solid element or with the original elements comprising the superelement.
- This element is nonlinear and requires a full Newton iterative solution, regardless of whether large or small deflections are specified. An exception to this is when MPC bonded contact is specified (KEYOPT(2) = 2 and KEYOPT(12) = 5 or 6).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The normal contact stiffness factor (FKN) must not be so large as to cause numerical instability.
- FTOLN, PINB, and FKOP can be changed between load steps or during restart stages.
- You can use this element in nonlinear static or nonlinear full transient analyses. In addition, you can use it in modal analyses, eigenvalue buckling analyses, and harmonic analyses. For these analysis types, the program assumes that the initial status of the element (that is, the status at the completion of the static prestress analysis, if any) does not change.
- When nodal detection is used and the contact node is on the axis of symmetry in an axisymmetric analysis, the contact pressure on that node is not accurate since the area of the node is zero. The contact force is accurate in this situation.

CONTA172 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The MU material property is not allowed in the Nonlinear Thermal product.
- The birth and death special feature is not allowed in the Nonlinear Structural and Nonlinear Thermal products.

ANSYS Structural

- The VOLT DOF (KEYOPT(1) = 3 through 6) is not allowed.
- The AZ DOF (KEYOPT(1) = 7) is not allowed.

ANSYS Mechanical

- The AZ DOF (KEYOPT(1) = 7) is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

CONTA173

3-D 4-Node Surface-to-Surface Contact

MP ME ST PR PRN DS EM <> PP EME MFS
Product Restrictions

CONTA173 Element Description

CONTA173 is used to represent contact and sliding between 3-D "target" surfaces (TARGE170) and a deformable surface, defined by this element. The element is applicable to 3-D structural and coupled field contact analyses. This element is located on the surfaces of 3-D solid or shell elements without midside nodes (SOLID65, SOLID70, SOLID96, SOLID185, SOLID285, SOLSH190, SHELL28, SHELL41, SHELL131, SHELL157, SHELL181, and MATRIX50). It has the same geometric characteristics as the solid or shell element face with which it is connected (see Figure 173.1: CONTA173 Geometry (p. 683)). Contact occurs when the element surface penetrates one of the target segment elements (TARGE170) on a specified target surface. Coulomb friction, shear stress friction, user-defined friction with the `USERFRIC` subroutine, and user-defined contact interaction with the `USERINTER` subroutine are allowed. This element also allows separation of bonded contact to simulate interface delamination. See CONTA173 in the *Mechanical APDL Theory Reference* for more details about this element. Other surface-to-surface contact elements (CONTA171, CONTA172, CONTA174) are also available.

Figure 173.1: CONTA173 Geometry

R = Element x-axis for isotropic friction

x_o = Element axis for orthotropic friction if **ESYS** is not supplied (parallel to global X-axis)

x = Element axis for orthotropic friction if **ESYS** is supplied

CONTA173 Input Data

The geometry and node locations are shown in Figure 173.1: CONTA173 Geometry (p. 683). The element is defined by four nodes (the underlying solid or shell element has no midside nodes). If the underlying solid or shell elements do have midside nodes, use CONTA174. The node ordering is consistent with

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the node ordering for the underlying solid or shell element. The positive normal is given by the right-hand rule going around the nodes of the element and is identical to the external normal direction of the underlying solid or shell element surface. For shell elements, the same nodal ordering between shell and contact elements defines upper surface contact; otherwise, it represents bottom surface contact. Remember the target surfaces must always be on its outward normal direction. See Generating Contact Elements in the *Contact Technology Guide* for more information on generating elements automatically using the **ESURF** command.

The 3-D contact surface elements are associated with the 3-D target segment elements (TARGE170) via a shared real constant set. ANSYS looks for contact only between surfaces with the same real constant set. For either rigid-flexible or flexible-flexible contact, one of the deformable surfaces must be represented by a contact surface. See Designating Contact and Target Surfaces in the *Contact Technology Guide* for more information.

If more than one target surface will make contact with the same boundary of solid elements, you must define several contact elements that share the same geometry but relate to separate targets (targets which have different real constant numbers), or you must combine the two target surfaces into one (targets that share the same real constant numbers).

CONTA173 supports isotropic and orthotropic Coulomb friction. For isotropic friction, specify a single coefficient of friction, MU, using either **TB** command input (recommended) or the **MP** command. For orthotropic friction, specify two coefficients of friction, MU1 and MU2, in two principal directions using **TB** command input. (See Contact Friction in the *Material Reference* for more information.)

For isotropic friction, the applicable coordinate system is the default element coordinate system (noted by the R and S axes in the above figure).

For orthotropic friction, the principal directions are determined as follows. The global coordinate system is used by default, or you may define a local element coordinate system with the **ESYS** command. (These are depicted by the $x_o$and x axes in the above figure.) The first principal direction is defined by projecting the first direction of the chosen coordinate system onto the contact surface. The second principal direction is defined by taking a cross product of the first principal direction and the contact normal. These directions also follow the rigid body rotation of the contact element to correctly model the directional dependence of friction. Be careful to choose the coordinate system (global or local) so that the first direction of that system is within 45° of the tangent to the contact surface.

If you want to set the coordinate directions for isotropic friction (to the global Cartesian system or another system via **ESYS**), you can define orthotropic friction and set MU1 = MU2.

To define a coefficient of friction for isotropic or orthotropic friction that is dependent on temperature, time, normal pressure, sliding distance, or sliding relative velocity, use the **TBFIELD** command along with **TB**,FRIC. See Contact Friction in the *Material Reference* for more information.

To implement a user-defined friction model, use the **TB**,FRIC command with *TBOPT* = USER to specify friction properties and write a `USERFRIC` subroutine to compute friction forces. See Writing Your Own Friction Law (USERFRIC) in the *Mechanical APDL Contact Technology Guide* for more information on how to use this feature. See also the *Guide to User-Programmable Features* for a detailed description of the `USERFRIC` subroutine.

In addition to the user-defined friction subroutine, the contact interaction subroutine `USERINTER` is available for user-defined interface interactions, including interactions in the normal and tangential directions as well coupled-field interactions. See Defining Your Own Contact Interaction (USERINTER) in the *Mechanical APDL Contact Technology Guide* for more information on how to use this feature. See also the *Guide to User-Programmable Features* for a detailed description of the `USERINTER` subroutine.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To model fluid penetration loads, use the **SFE** command to specify the fluid pressure and fluid penetration starting points. For more information, see Applying Fluid Pressure-Penetration Loads in the *Contact Technology Guide*.

To model proper momentum transfer and energy balance between contact and target surfaces, impact constraints should be used in transient dynamic analysis. See the description of KEYOPT(7) below and the contact element discussion in the *Mechanical APDL Theory Reference* for details.

To model separation of bonded contact with KEYOPT(12) = 2, 3, 4, 5, or 6, use the **TB** command with the CZM label. See Debonding in the *Contact Technology Guide* for more information.

To model wear at the contact surface, use the **TB** command with the WEAR label. See Contact Surface Wear in the *Contact Technology Guide* for more information.

Two types of geometry correction are available for this element: surface smoothing and bolt thread modeling. Surface smoothing is a geometry correction technique that eliminates inaccuracies introduced by faceted elements on a curved (spherical or revolute) contact surface. Bolt thread modeling provides a method for simulating contact between a threaded bolt and bolt hole without having to model the detailed thread geometry. Both of these geometry correction techniques are implemented through section definitions (**SECTYPE**, **SECDATA**, and **SECNUM** commands). For more information, see Geometry Correction for Contact and Target Surfaces in the *Contact Technology Guide*.

A summary of the element input is given in "CONTA173 Input Summary" (p. 685). A general description of element input is given in Element Input (p. 43).

CONTA173 Input Summary

Nodes

I, J, K, L

Degrees of Freedom

UX, UY, UZ (if KEYOPT(1) = 0)
UX, UY, UZ, TEMP (if KEYOPT(1) = 1)
TEMP (if KEYOPT(1) = 2)
UX, UY, UZ, TEMP, VOLT (if KEYOPT(1) = 3)
TEMP, VOLT (if KEYOPT(1) = 4)
UX, UY, UZ, VOLT (if KEYOPT(1) = 5)
VOLT (if KEYOPT(1) = 6)
MAG (if KEYOPT(1) = 7)

Real Constants

R1, R2, FKN, FTOLN, ICONT, PINB,
PMAX, PMIN, TAUMAX, CNOF, FKOP, FKT,
COHE, TCC, FHTG, SBCT, RDVF, FWGT,
ECC, FHEG, FACT, DC, SLTO, TNOP,
TOLS, MCC, PPCN, FPAT, COR, STRM
FDMN, FDMT, FDMD, FDMS, TBND
See Table 173.1: CONTA173 Real Constants (p. 692) for descriptions of the real constants.

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MP command: MU, EMIS

Surface Loads

Pressure, Face 1 (I-J-K-L) (opposite to contact normal direction); used for fluid pressure penetration loading. On the **SFE** command use *LKEY* = 1 to specify the pressure values, and use *LKEY* = 2 to specify starting points and penetrating points.
Convection, Face 1 (I-J-K-L)
Heat Flux, Face 1 (I-J-K-L)

Special Features

Birth and death
Debonding
Fluid pressure penetration
Isotropic friction
Large deflection
Linear perturbation
Nonlinearity
Orthotropic friction
Section definition for geometry correction of spherical and revolute surfaces
User-defined contact interaction
User-defined friction

KEYOPTs

Presented below is a list of KEYOPTS available for this element. Included are links to sections in the *Contact Technology Guide* where more information is available on a particular topic.

KEYOPT(1)

Selects degrees of freedom:

0 --
UX, UY, UZ

1 --
UX, UY, UZ, TEMP

2 --
TEMP

3 --
UX, UY, UZ, TEMP, VOLT

4 --
TEMP, VOLT

5 --
UX, UY, UZ, VOLT

6 --
VOLT

7 --
MAG

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(2)

Contact algorithm:

0 --
Augmented Lagrangian (default)

1 --
Penalty function

2 --
Multipoint constraint (MPC); see Multipoint Constraints and Assemblies in the *Contact Technology Guide* for more information

3 --
Lagrange multiplier on contact normal and penalty on tangent

4 --
Pure Lagrange multiplier on contact normal and tangent

KEYOPT(3)

Units of normal contact stiffness:

0 --
FORCE/LENGTH^3 (default)

1 --
FORCE/LENGTH

Note

KEYOPT(3) = 1 is valid only when a penalty-based algorithm is used (KEYOPT(2) = 0 or 1) and the absolute normal contact stiffness value is explicitly specified (that is, a negative value input for real constant FKN).

KEYOPT(4)

Location of contact detection point:

0 --
On Gauss point (for general cases)

1 --
On nodal point - normal from contact surface

2 --
On nodal point - normal to target surface

3 --
On nodal point - normal from contact surface (projection-based method)

Note

When using the multipoint constraint (MPC) approach to define surface-based constraints, use KEYOPT(4) in the following way: set KEYOPT(4) = 1 for a force-distributed constraint;

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

set KEYOPT(4) = 2 for a rigid surface constraint; set KEYOPT(4) = 3 for a coupling constraint. See Surface-based Constraints for more information.

Note

Certain restrictions apply when the surface-projection-based method (KEYOPT(4) = 3) is defined. See Using the Surface Projection Based Contact Method (KEYOPT(4) = 3) for more information.

KEYOPT(5)

CNOF/ICONT automated adjustment:

0 --
No automated adjustment

1 --
Close gap with auto CNOF

2 --
Reduce penetration with auto CNOF

3 --
Close gap/reduce penetration with auto CNOF

4 --
Auto ICONT

KEYOPT(6)

Contact stiffness variation (used to enhance stiffness updating when KEYOPT(10) > 0):

0 --
Use default range for stiffness updating

1 --
Make a nominal refinement to the allowable stiffness range

2 --
Make an aggressive refinement to the allowable stiffness range

KEYOPT(7)

Element level time incrementation control / impact constraints:

0 --
No control

1 --
Automatic bisection of increment

2 --
Change in contact predictions made to maintain a reasonable time/load increment

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3 --
Change in contact predictions made to achieve the minimum time/load increment whenever a change in contact status occurs

4 --
Use impact constraints for standard or rough contact (KEYOPT(12) = 0 or 1) in a transient dynamic analysis with automatic adjustment of time increment

---

Note

KEYOPT(7) = 2, 3, and 4 include an automatic adjustment of the time increment. This is activated only if the command **SOLCONTROL**,ON,ON was issued prior to the solution.

KEYOPT(8)
Asymmetric contact selection:

0 --
No action

2 --
ANSYS internally selects which asymmetric contact pair is used at the solution stage (used only when symmetry contact is defined).

KEYOPT(9)
Effect of initial penetration or gap:

0 --

Include both initial geometrical penetration or gap and offset

1 --
Exclude both initial geometrical penetration or gap and offset

2 --
Include both initial geometrical penetration or gap and offset, but with ramped effects

3 --
Include offset only (exclude initial geometrical penetration or gap)

4 --
Include offset only (exclude initial geometrical penetration or gap), but with ramped effects

5 --
Include offset only (exclude initial geometrical penetration or gap) regardless of the initial contact status (near-field or closed)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6 --
Include offset only (exclude initial geometrical penetration or gap), but with ramped effects regardless of the initial contact status (near-field or closed)

Note

The effects of KEYOPT(9) are dependent on settings for other KEYOPTs. The indicated initial gap effect is considered only if KEYOPT(12) = 4 or 5. See the discussion on using KEYOPT(9) in the *Contact Technology Guide* for more information.

KEYOPT(10)
Contact stiffness update:

0 --
Each load step if FKN is redefined during load step (pair based).

2 --
Each iteration based on current mean stress of underlying elements (pair based).

KEYOPT(11)
Shell thickness effect:

0 --
Exclude

1 --
Include

KEYOPT(12)
Behavior of contact surface:

0 --
Standard

1 --
Rough

2 --
No separation (sliding permitted)

3 --
Bonded

4 --
No separation (always)

5 --
Bonded (always)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6 --
Bonded (initial contact)

Note

When KEYOPT(12) = 5 or 6 is used with the MPC algorithm to model surface-based constraints, the KEYOPT(12) setting will have an impact on the local coordinate system of the contact element nodes. See Specifying a Local Coordinate System in the *Contact Technology Guide* for more information.

KEYOPT(14)
Behavior of fluid pressure penetration load. KEYOPT(14) is valid only if a fluid pressure penetration load (**SFE**,,,PRES) is applied to the contact element:

0 --
Fluid pressure penetration load varies during iterations (default)

1 --
Fluid pressure penetration load remains constant over the substep

KEYOPT(15)
Effect of contact stabilization damping:

0 --
Damping is activated only in the first load step (default).

1 --
Deactivate automatic damping.

2 --
Damping is activated for all load steps.

3 --
Damping is activated at all times regardless of the contact status of previous substeps.

Note

Stabilization damping is only applied to contact pairs in near-field contact. When KEYOPT(15) = 0, 1, or 2, stabilization damping will not be applied in the current substep if any contact detection point had a closed status in the previous substep. However, when KEYOPT(15) = 3, stabilization damping is always applied as long as the current contact status is near-field. See Applying Contact Stabilization Damping in the *Contact Technology Guide* for more information.

KEYOPT(16)
Squeal damping controls for interpretation of real constants FDMD and FDMS:

0 --
FDMD and FDMS are scaling factors for destabilizing and stabilizing damping (default).

1 --
FDMD is a constant friction-sliding velocity gradient. FDMS is the stabilization damping coefficient.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2 --

FDMD and FDMS are the destabilizing and stabilization damping coefficients.

Table 173.1: CONTA173 Real Constants

No.	Name	Description	For more information, see this section in the *Contact Technology Guide* . . .
1	R1	Target radius for cylinder, cone, or sphere	Defining the Target Surface
2	R2	Target radius at second node of cone	Defining the Target Surface
3	FKN	Normal penalty stiffness factor [1] [2]	Determining Contact Stiffness and Penetration
4	FTOLN	Penetration tolerance factor	Determining Contact Stiffness and Penetration
5	ICONT	Initial contact closure	Adjusting Initial Contact Conditions
6	PINB	Pinball region	Determining Contact Status and the Pinball Region or Defining Influence Range (PINB)
7	PMAX	Upper limit of initial allowable penetration	Adjusting Initial Contact Conditions
8	PMIN	Lower limit of initial allowable penetration	Adjusting Initial Contact Conditions
9	TAUMAX	Maximum friction stress [1] [2]	Choosing a Friction Model
10	CNOF	Contact surface offset [1]	Adjusting Initial Contact Conditions
11	FKOP	Contact opening stiffness [1] [2]	Selecting Surface Interaction Models
12	FKT	Tangent penalty stiffness factor [1] [2]	Determining Contact Stiffness
13	COHE	Contact cohesion	Choosing a Friction Model
14	TCC	Thermal contact conductance [1] [2]	Modeling Conduction
15	FHTG	Frictional heating factor	Modeling Heat Generation Due to Friction
16	SBCT	Stefan-Boltzmann constant	Modeling Radiation
17	RDVF	Radiation view factor [1] [2]	Modeling Radiation

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

No.	Name	Description	For more information, see this section in the *Contact Technology Guide* . . .
18	FWGT	Heat distribution weighing factor	Modeling Heat Generation Due to Friction (thermal) or Heat Generation Due to Electric Current (electric)
19	ECC	Electric contact conductance [1] [2]	Modeling Surface Interaction
20	FHEG	Joule dissipation weight factor	Heat Generation Due to Electric Current
21	FACT	Static/dynamic ratio	Static and Dynamic Friction Coefficients
22	DC	Exponential decay coefficient	Static and Dynamic Friction Coefficients
23	SLTO	Allowable elastic slip	Using FKT and SLTO
24	TNOP	Maximum allowable tensile contact pressure	Chattering Control Parameters
25	TOLS	Target edge extension factor	Selecting Location of Contact Detection
26	MCC	Magnetic contact permeance [1] [2]	Modeling Magnetic Contact
27	PPCN	Pressure penetration criterion [1] [2]	Specifying a Pressure Penetration Criterion
28	FPAT	Fluid penetration acting time	Specifying a Fluid Penetration Acting Time
29	COR	Coefficient of restitution	Impact Between Rigid Bodies
30	STRM	Load step number for ramping penetration	Adjusting Initial Contact Conditions
31	FDMN	Normal stabilization damping factor [1] [2]	Applying Contact Stabilization Damping
32	FDMT	Tangential stabilization damping factor [1] [2]	Applying Contact Stabilization Damping
33	FDMD	Destabilizing squeal damping factor	Forced Frictional Sliding Using Velocity Input
34	FDMS	Stabilizing squeal damping factor	Forced Frictional Sliding Using Velocity Input
35	TBND	Critical bonding temperature [1] [2]	Using TBND

1. This real constant can be defined as a function of certain primary variables.

2. This real constant can be defined by the user subroutine USERCNPROP.F.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CONTA173 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 173.2: CONTA173 Element Output Definitions (p. 694)

A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 173.2: CONTA173 Element Output Definitions (p. 694) gives element output at the element level. In the results file, the nodal results are obtained from its closest integration point.

Table 173.2: CONTA173 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes I, J, K, L	Y	Y
XC, YC, ZC	Location where results are reported	Y	5
TEMP	Temperatures T(I), T(J), T(K), T(L)	Y	Y
VOLU	AREA	Y	Y
NPI	Number of integration points	Y	-
ITRGET	Target surface number (assigned by ANSYS)	Y	-
ISOLID	Underlying solid or shell element number	Y	-
CONT:STAT	Current contact statuses	1	1
OLDST	Old contact statuses	1	1
ISEG	Current contacting target element number	Y	Y
OLDSEG	Underlying old target number	Y	-
CONT:PENE	Current penetration (gap = 0; penetration = positive value)	Y	Y
CONT:GAP	Current gap (gap = negative value; penetration = 0)	Y	Y
NGAP	New or current gap at current converged substep (gap = negative value; penetration = positive value)	Y	-
OGAP	Old gap at previously converged substep (gap = negative value; penetration = positive value)	Y	-
IGAP	Initial gap at start of current substep (gap = negative value; penetration = positive value)	Y	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
GGAP	Geometric gap at current converged substep (gap = negative value; penetration = positive value)	-	Y
CONT:PRES	Normal contact pressure	Y	Y
TAUR/TAUS[7]	Tangential contact stresses	Y	Y
KN	Current normal contact stiffness (Force/Length^3)	Y	Y
KT	Current tangent contact stiffness (Force/Length^3)	Y	Y
MU[8]	Friction coefficient	Y	Y
TASS/TASR[7]	Total (algebraic sum) sliding in S and R directions	3	3
AASS/AASR[7]	Total (absolute sum) sliding in S and R directions	3	3
TOLN	Penetration tolerance	Y	Y
CONT:SFRIC	Frictional stress, SQRT (TAUR**2+TAUS**2)	Y	Y
CONT:STOTAL	Total stress, SQRT (PRES**2+TAUR**2+TAUS**2)	Y	Y
CONT:SLIDE	Amplitude of total accumulated sliding, SQRT (TASS**2+TASR**2)	3	3
FDDIS	Frictional energy dissipation	6	6
ELSI	Total equivalent elastic slip distance	-	Y
PLSI	Total (accumulated) equivalent plastic slip due to frictional sliding	-	Y
GSLID	Amplitude of total accumulated sliding (including near-field)	-	9
VREL	Equivalent sliding velocity (slip rate)	-	Y
DBA	Penetration variation	Y	Y
PINB	Pinball Region	-	Y
CONT:CNOS	Total number of contact status changes during substep	Y	Y
TNOP	Maximum allowable tensile contact pressure	Y	Y
SLTO	Allowable elastic slip	Y	Y
CAREA	Contacting area	-	Y
CONT:FPRS	Actual applied fluid penetration pressure	-	Y
FSTART	Fluid penetration starting time	-	Y
DTSTART	Load step time during debonding	Y	Y
DPARAM	Debonding parameter	Y	Y
DENERI	Energy released due to separation in normal direction - mode I debonding	Y	Y
DENERII	Energy released due to separation in tangential direction - mode II debonding	Y	Y
CNFX[4]	Contact element force-X component	-	Y
CNFY	Contact element force-Y component	-	Y
CNFZ	Contact element force-Z component	-	Y
SDAMP	Squeal damping coefficient / Stabilization damping coefficient	-	Y
CONV	Convection coefficient	Y	Y
RAC	Radiation coefficient	Y	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
TCC	Conductance coefficient	Y	Y
TEMPS	Temperature at contact point	Y	Y
TEMPT	Temperature at target surface	Y	Y
FXCV	Heat flux due to convection	Y	Y
FXRD	Heat flux due to radiation	Y	Y
FXCD	Heat flux due to conductance	Y	Y
CONT:FLUX	Total heat flux at contact surface	Y	Y
FXNP	Flux input	-	Y
CNFH	Contact element heat flow	-	Y
JCONT	Contact current density (Current/Unit Area)	Y	Y
CCONT	Contact charge density (Charge/Unit Area)	Y	Y
HJOU	Contact power/area	Y	Y
ECURT	Current per contact element	-	Y
ECHAR	Charge per contact element	-	Y
ECC	Electric contact conductance (for electric current DOF), or electric contact capacitance per unit area (for piezoelectric or electrostatic DOFs)	Y	Y
VOLTS	Voltage on contact nodes	Y	Y
VOLTT	Voltage on associated target	Y	Y
MCC	Magnetic contact permeance	Y	Y
MFLUX	Magnetic flux density	Y	Y
MAGS	Magnetic potential on contact node	Y	Y
MAGT	Magnetic potential on associated target	Y	Y
WEARX, WEARY, WEARZ	Wear correction - X, Y, and Z components	-	Y

1. The possible values of *STAT* and *OLDST* are:

   0 = Open and not near contact
   1 = Open but near contact
   2 = Closed and sliding
   3 = Closed and sticking

2. ANSYS will evaluate model to detect initial conditions.

3. Only accumulates the sliding for sliding and closed contact (STAT = 2,3).

4. Contact element forces are defined in the global Cartesian system

5. Available only at centroid as a ***GET** item.

6. FDDIS = (contact friction stress)*(sliding distance of substep)/(time increment of substep)

7. For the case of orthotropic friction, components are defined in the global Cartesian system (default) or in the local element coordinate system specified by **ESYS**.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. For orthotropic friction, an equivalent coefficient of friction is output.

9. Accumulated sliding distance for near-field, sliding, and closed contact (STAT = 1,2,3).

Note

If **ETABLE** is used for the CONT items, the reported data is averaged across the element.

Note

Contact results (including all element results) are generally not reported for elements that have a status of "open and not near contact" (far-field).

Table 173.3: CONTA173 Item and Sequence Numbers (p. 697) lists output available through the **ETABLE** command using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 173.3: CONTA173 Item and Sequence Numbers (p. 697):

Name
output quantity as defined in the Table 173.2: CONTA173 Element Output Definitions (p. 694)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J,K,L
sequence number for data at nodes I,J,K,L

Table 173.3: CONTA173 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
PRES	SMISC	13	1	2	3	4
TAUR	SMISC	-	5	6	7	8
TAUS	SMISC	-	9	10	11	12
FLUX	SMISC	-	14	15	16	17
FDDIS	SMISC	-	18	19	20	21
FXCV	SMISC		22	23	24	25
FXRD	SMISC	-	26	27	28	29
FXCD	SMISC	-	30	31	32	33
FXNP	SMISC	-	34	35	36	37
JCONT	SMISC	-	38	39	40	41
CCONT	SMISC	-	38	39	40	41
HJOU	SMISC	-	42	43	44	45

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
MFLUX	SMISC	-	46	47	48	49
STAT[1]	NMISC	41	1	2	3	4
OLDST	NMISC	-	5	6	7	8
PENE[2]	NMISC	-	9	10	11	12
DBA	NMISC	-	13	14	15	16
TASR	NMISC	-	17	18	19	20
TASS	NMISC	-	21	22	23	24
KN	NMISC	-	25	26	27	28
KT	NMISC	-	29	30	31	32
TOLN	NMISC	-	33	34	35	36
IGAP	NMISC	-	37	38	39	40
PINB	NMISC	42	-	-	-	-
CNFX	NMISC	43	-	-	-	-
CNFY	NMISC	44	-	-	-	-
CNFZ	NMISC	45	-	-	-	-
ISEG	NMISC	-	46	47	48	49
AASR	NMISC	-	50	51	52	53
AASS	NMISC	-	54	55	56	57
CAREA	NMISC	58	-	-	-	-
MU	NMISC	-	62	63	64	65
DTSTART	NMISC	-	66	67	68	69
DPARAM	NMISC	-	70	71	72	73
FPRS	NMISC	-	74	75	76	77
TEMPS	NMISC	-	78	79	80	81
TEMPT	NMISC	-	82	83	84	85
CONV	NMISC	-	86	87	88	89
RAC	NMISC	-	90	91	92	93
TCC	NMISC	-	94	95	96	97
CNFH	NMISC	98	-	-	-	-
ECURT	NMISC	99	-	-	-	-
ECHAR	NMISC	99	-	-	-	-
ECC	NMISC	-	100	101	102	103
VOLTS	NMISC	-	104	105	106	107
VOLTT	NMISC	-	108	109	110	111
CNOS	NMISC	-	112	113	114	115
TNOP	NMISC	-	116	117	118	119

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quant-ity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
SLTO	NMISC	-	120	121	122	123
MCC	NMISC	-	124	125	126	127
MAGS	NMISC	-	128	129	130	131
MAGT	NMISC	-	132	133	134	135
ELSI	NMISC	-	136	137	138	139
DENERI	NMISC	-	140	141	142	143
DENERII	NMISC	-	144	145	146	147
FSTART	NMISC	-	148	140	150	151
GGAP	NMISC	-	152	153	154	155
VREL	NMISC	-	156	157	158	159
SDAMP	NMISC	-	160	161	162	163
PLSI	NMISC	-	164	165	166	167
GSLID	NMISC	-	168	169	170	171
WEARX	NMISC	-	172	173	174	175
WEARY	NMISC	-	176	177	178	179
WEARZ	NMISC	-	180	181	182	183

1. Element Status = highest value of status of integration points within the element

2. Penetration = positive value, gap = negative value

You can display or list contact results through several POST1 postprocessor commands. The contact specific items for the **PLNSOL**, **PLESOL**, **PRNSOL**, and **PRESOL** commands are listed below:

STAT	Contact status
PENE	Contact penetration
PRES	Contact pressure
SFRIC	Contact friction stress
STOT	Contact total stress (pressure plus friction)
SLIDE	Contact sliding distance
GAP	Contact gap distance
FLUX	Total heat flux at contact surface
CNOS	Total number of contact status changes during substep
FPRS	Actual applied fluid penetration pressure

CONTA173 Assumptions and Restrictions

- The 3-D contact element must coincide with the external surface of the underlying solid or shell element or with the original elements comprising the superelement.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- This element is nonlinear and requires a full Newton iterative solution, regardless of whether large or small deflections are specified. An exception to this is when MPC bonded contact is specified (KEYOPT(2) = 2 and KEYOPT(12) = 5 or 6).
- The normal contact stiffness factor (FKN) must not be so large as to cause numerical instability.
- FTOLN, PINB, and FKOP can be changed between load steps or during restart stages.
- You can use this element in nonlinear static or nonlinear full transient analyses. In addition, you can use it in modal analyses, eigenvalue buckling analyses, and harmonic analyses. For these analysis types, the program assumes that the initial status of the element (that is, the status at the completion of the static prestress analysis, if any) does not change.

CONTA173 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The MU material property is not allowed in the Nonlinear Thermal product.
- The birth and death special feature is not allowed in the Nonlinear Structural and Nonlinear Thermal products.

ANSYS Structural

- The VOLT DOF (KEYOPT(1) = 3 through 6) is not allowed.
- The MAG DOF (KEYOPT(1) = 7) is not allowed.

ANSYS Mechanical

- The MAG DOF (KEYOPT(1) = 7) is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CONTA174

3-D 8-Node Surface-to-Surface Contact

MP ME ST PR PRN DS EM <> PP EME MFS
Product Restrictions

CONTA174 Element Description

CONTA174 is used to represent contact and sliding between 3-D "target" surfaces (TARGE170) and a deformable surface, defined by this element. The element is applicable to 3-D structural and coupled field contact analyses.

The element is located on the surfaces of 3-D solid or shell elements with midside nodes (SOLID87, SOLID90, SOLID98, SOLID122, SOLID123, SOLID186, SOLID187, SOLID226, SOLID227, SOLID231, SOLID232, SHELL132, SHELL281, and MATRIX50).

The element has the same geometric characteristics as the solid or shell element face with which it is connected (see Figure 174.1: CONTA174 Geometry (p. 701) below). Contact occurs when the element surface penetrates one of the target segment elements (TARGE170) on a specified target surface. Coulomb friction, shear stress friction, user-defined friction with the `USERFRIC` subroutine, and user-defined contact interaction with the `USERINTER` subroutine are allowed. The element also allows separation of bonded contact to simulate interface delamination.

See CONTA174 in the *Mechanical APDL Theory Reference* for more details about this element. Other surface-to-surface contact elements (CONTA171, CONTA172, CONTA173) are also available.

Figure 174.1: CONTA174 Geometry

R = Element x-axis for isotropic friction

x_o = Element axis for orthotropic friction if **ESYS** is not supplied (parallel to global X-axis)

x = Element axis for orthotropic friction if **ESYS** is supplied

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CONTA174 Input Data

The geometry and node locations are shown in Figure 174.1: CONTA174 Geometry (p. 701). The element is defined by eight nodes (the underlying solid or shell element has midside nodes). It can degenerate to a six node element depending on the shape of the underlying solid or shell elements. If the underlying solid or shell elements do not have midside nodes, use CONTA173 (you may still use CONTA174 but you must drop all midside nodes). See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information on the use of midside nodes. The node ordering is consistent with the node ordering for the underlying solid or shell element. The positive normal is given by the right-hand rule going around the nodes of the element and is identical to the external normal direction of the underlying solid or shell element surface. For shell elements, the same nodal ordering between shell and contact elements defines upper surface contact; otherwise, it represents bottom surface contact. Remember the target surfaces must always be on its outward normal direction. See Generating Contact Elements in the *Contact Technology Guide* for more information on generating elements automatically using the **ESURF** command.

The 3-D contact surface elements (CONTA173 and CONTA174) are associated with the 3-D target segment elements (TARGE170) via a shared real constant set. ANSYS looks for contact only between surfaces with the same real constant set. For either rigid-flexible or flexible-flexible contact, one of the deformable surfaces must be represented by a contact surface. See Designating Contact and Target Surfaces in the *Contact Technology Guide* for more information.

If more than one target surface will make contact with the same boundary of solid elements, you must define several contact elements that share the same geometry but relate to separate targets (targets which have different real constant numbers), or you must combine two target surfaces into one (targets that share the same real constant numbers).

CONTA174 supports isotropic and orthotropic Coulomb friction. For isotropic friction, specify a single coefficient of friction, MU, using either **TB** command input (recommended) or the **MP** command. For orthotropic friction, specify two coefficients of friction, MU1 and MU2, in two principal directions using **TB** command input. (See Contact Friction in the *Material Reference* for more information.)

For isotropic friction, the applicable coordinate system is the default element coordinate system (noted by the R and S axes in the above figure).

For orthotropic friction, the principal directions are determined as follows. The global coordinate system is used by default, or you may define a local element coordinate system with the **ESYS** command. (These are depicted by the x_o and x axes in the above figure.) The first principal direction is defined by projecting the first direction of the chosen coordinate system onto the contact surface. The second principal direction is defined by taking a cross product of the first principal direction and the contact normal. These directions also follow the rigid body rotation of the contact element to correctly model the directional dependence of friction. Be careful to choose the coordinate system (global or local) so that the first direction of that system is within 45° of the tangent to the contact surface.

If you want to set the coordinate directions for isotropic friction (to the global Cartesian system or another system via **ESYS**), you can define orthotropic friction and set MU1 = MU2.

To define a coefficient of friction for isotropic or orthotropic friction that is dependent on temperature, time, normal pressure, sliding distance, or sliding relative velocity, use the **TBFIELD** command along with **TB**,FRIC. See Contact Friction in the *Material Reference* for more information.

To implement a user-defined friction model, use the **TB**,FRIC command with *TBOPT* = USER to specify friction properties and write a `USERFRIC` subroutine to compute friction forces. See Writing Your Own

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Friction Law (USERFRIC) in the *Mechanical APDL Contact Technology Guide* for more information on how to use this feature. See also the *Guide to User-Programmable Features* for a detailed description of the USERFRIC subroutine.

In addition to the user-defined friction subroutine, the contact interaction subroutine USERINTER is available for user-defined interface interactions, including interactions in the normal and tangential directions as well coupled-field interactions. See Defining Your Own Contact Interaction (USERINTER) in the *Mechanical APDL Contact Technology Guide* for more information on how to use this feature. See also the *Guide to User-Programmable Features* for a detailed description of the USERINTER subroutine.

To model fluid penetration loads, use the **SFE** command to specify the fluid pressure and fluid penetration starting points. For more information, see Applying Fluid Pressure-Penetration Loads in the *Contact Technology Guide*.

To model proper momentum transfer and energy balance between contact and target surfaces, impact constraints should be used in transient dynamic analysis. See the description of KEYOPT(7) below and the contact element discussion in the *Mechanical APDL Theory Reference* for details.

To model separation of bonded contact with KEYOPT(12) = 2, 3, 4, 5, or 6, use the **TB** command with the CZM label. See Debonding in the *Contact Technology Guide* for more information.

To model wear at the contact surface, use the **TB** command with the WEAR label. See Contact Surface Wear in the *Contact Technology Guide* for more information.

Two types of geometry correction are available for this element: surface smoothing and bolt thread modeling. Surface smoothing is a geometry correction technique that eliminates inaccuracies introduced by faceted elements on a curved (spherical or revolute) contact surface. Bolt thread modeling provides a method for simulating contact between a threaded bolt and bolt hole without having to model the detailed thread geometry. Both of these geometry correction techniques are implemented through section definitions (**SECTYPE**, **SECDATA**, and **SECNUM** commands). For more information, see Geometry Correction for Contact and Target Surfaces in the *Contact Technology Guide*.

A summary of the element input is given in "CONTA174 Input Summary" (p. 703). A general description of element input is given in Element Input (p. 43). For axisymmetric applications see Harmonic Axisymmetric Elements (p. 83).

CONTA174 Input Summary

Nodes

I, J, K, L, M, N, O, P

Degrees of Freedom

UX, UY, UZ (if KEYOPT(1) = 0)
UX, UY, UZ, TEMP (if KEYOPT(1) = 1)
TEMP (if KEYOPT(1) = 2)
UX, UY, UZ, TEMP, VOLT (if KEYOPT(1) = 3)
TEMP, VOLT (if KEYOPT(1) = 4)
UX, UY, UZ, VOLT (if KEYOPT(1) = 5)
VOLT (if KEYOPT(1) = 6)
MAG (if KEYOPT(1) = 7)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Real Constants

R1, R2, FKN, FTOLN, ICONT, PINB,
PMAX, PMIN, TAUMAX, CNOF, FKOP, FKT,
COHE, TCC, FHTG, SBCT, RDVF, FWGT,
ECC, FHEG, FACT, DC, SLTO, TNOP,
TOLS, MCC, PPCN, FPAT, COR, STRM
FDMN, FDMT, FDMD, FDMS, TBND
See Table 174.1: CONTA174 Real Constants (p. 710) for descriptions of the real constants.

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: MU, EMIS

Surface Loads

Pressure, Face 1 (I-J-K-L) (opposite to contact normal direction); used for fluid pressure penetration loading. On the **SFE** command use *LKEY* = 1 to specify the pressure values, and use *LKEY* = 2 to specify starting points and penetrating points.
Convection, Face 1 (I-J-K-L)
Heat Flux, Face 1 (I-J-K-L)

Special Features

Birth and death
Debonding
Fluid pressure penetration
Isotropic friction
Large deflection
Linear perturbation
Nonlinearity
Orthotropic friction
Section definition for geometry correction of spherical and revolute surfaces
User-defined contact interaction
User-defined friction

KEYOPTs

Presented below is a list of KEYOPTS available for this element. Included are links to sections in the *Contact Technology Guide* where more information is available on a particular topic.

KEYOPT(1)

Selects degrees of freedom:

0 --
UX, UY, UZ

1 --
UX, UY, UZ, TEMP

2 --
TEMP

3 --
UX, UY, UZ, TEMP, VOLT

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4 --
TEMP, VOLT

5 --
UX, UY, UZ, VOLT

6 --
VOLT

7 --
MAG

KEYOPT(2)
Contact algorithm:

0 --
Augmented Lagrangian (default)

1 --
Penalty function

2 --
Multipoint constraint (MPC); see Multipoint Constraints and Assemblies in the *Contact Technology Guide* for more information

3 --
Lagrange multiplier on contact normal and penalty on tangent

4 --
Pure Lagrange multiplier on contact normal and tangent

KEYOPT(3)
Units of normal contact stiffness:

0 --
FORCE/LENGTH^3 (default)

1 --
FORCE/LENGTH

Note

KEYOPT(3) = 1 is valid only when a penalty-based algorithm is used (KEYOPT(2) = 0 or 1) and the absolute normal contact stiffness value is explicitly specified (that is, a negative value input for real constant FKN). It is not recommended in conjunction with using nodal detection options (KEYOPT(4) = 1 or 2) if any midside nodes exist for the contact element.

KEYOPT(4)
Location of contact detection point:

0 --
On Gauss point (for general cases)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
On nodal point - normal from contact surface

2 --
On nodal point - normal to target surface

3 --
On nodal point - normal from contact surface (projection-based method)

Note

When using the multipoint constraint (MPC) approach to define surface-based constraints, use KEYOPT(4) in the following way: set KEYOPT(4) = 1 for a force-distributed constraint; set KEYOPT(4) = 2 for a rigid surface constraint; set KEYOPT(4) = 3 for a coupling constraint. See Surface-based Constraints for more information.

Note

Certain restrictions apply when the surface-projection-based method (KEYOPT(4) = 3) is defined. See Using the Surface Projection Based Contact Method (KEYOPT(4) = 3) for more information.

KEYOPT(5)

CNOF/ICONT Automated adjustment:

0 --
No automated adjustment

1 --
Close gap with auto CNOF

2 --
Reduce penetration with auto CNOF

3 --
Close gap/reduce penetration with auto CNOF

4 --
Auto ICONT

KEYOPT(6)

Contact stiffness variation (used to enhance stiffness updating when KEYOPT(10) > 0):

0 --
Use default range for stiffness updating

1 --
Make a nominal refinement to the allowable stiffness range

2 --
Make an aggressive refinement to the allowable stiffness range

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(7)

Element level time incrementation control / impact constraints:

0 --

No control

1 --

Automatic bisection of increment

2 --

Change in contact predictions made to maintain a reasonable time/load increment

3 --

Change in contact predictions made to achieve the minimum time/load increment whenever a change in contact status occurs

4 --

Use impact constraints for standard or rough contact (KEYOPT(12) = 0 or 1) in a transient dynamic analysis with automatic adjustment of time increment

---

Note

KEYOPT(7) = 2, 3, and 4 include an automatic adjustment of the time increment. This is activated only if the command **SOLCONTROL**,ON,ON was issued prior to the solution.

KEYOPT(8)

Asymmetric contact selection:

0 --

No action

2 --

ANSYS internally selects which asymmetric contact pair is used at the solution stage (used only when symmetry contact is defined).

KEYOPT(9)

Effect of initial penetration or gap:

0 --

Include both initial geometrical penetration or gap and offset

1 --

Exclude both initial geometrical penetration or gap and offset

2 --

Include both initial geometrical penetration or gap and offset, but with ramped effects

3 --

Include offset only (exclude initial geometrical penetration or gap)

4 --

Include offset only (exclude initial geometrical penetration or gap), but with ramped effects

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5 --
Include offset only (exclude initial geometrical penetration or gap) regardless of the initial contact status (near-field or closed)

6 --
Include offset only (exclude initial geometrical penetration or gap), but with ramped effects regardless of the initial contact status (near-field or closed)

Note

The effects of KEYOPT(9) are dependent on settings for other KEYOPTs. The indicated initial gap effect is considered only if KEYOPT(12) = 4 or 5. See the discussion on using KEYOPT(9) in the *Contact Technology Guide* for more information.

KEYOPT(10)

Contact stiffness update:

0 --
Each load step if FKN is redefined during load step (pair based).

2 --
Each iteration based on current mean stress of underlying elements (pair based).

KEYOPT(11)

Shell thickness effect:

0 --
Exclude

1 --
Include

KEYOPT(12)

Behavior of contact surface:

0 --
Standard

1 --
Rough

2 --
No separation (sliding permitted)

3 --
Bonded

4 --
No separation (always)

5 --
Bonded (always)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6 --
Bonded (initial contact)

Note

When KEYOPT(12) = 5 or 6 is used with the MPC algorithm to model surface-based constraints, the KEYOPT(12) setting will have an impact on the local coordinate system of the contact element nodes. See Specifying a Local Coordinate System in the *Contact Technology Guide* for more information.

KEYOPT(14)
Behavior of fluid pressure penetration load. KEYOPT(14) is valid only if a fluid pressure penetration load (**SFE**,,,PRES) is applied to the contact element:

0 --
Fluid pressure penetration load varies during iterations (default)

1 --
Fluid pressure penetration load remains constant over the substep

KEYOPT(15)
Effect of contact stabilization damping:

0 --
Damping is activated only in the first load step (default).

1 --
Deactivate automatic damping.

2 --
Damping is activated for all load steps.

3 --
Damping is activated at all times regardless of the contact status of previous substeps.

Note

Stabilization damping is only applied to contact pairs in near-field contact. When KEYOPT(15) = 0, 1, or 2, stabilization damping will not be applied in the current substep if any contact detection point had a closed status in the previous substep. However, when KEYOPT(15) = 3, stabilization damping is always applied as long as the current contact status is near-field. See Applying Contact Stabilization Damping in the *Contact Technology Guide* for more information.

KEYOPT(16)
Squeal damping controls for interpretation of real constants FDMD and FDMS:

0 --
FDMD and FDMS are scaling factors for destabilizing and stabilizing damping (default).

1 --
FDMD is a constant friction-sliding velocity gradient. FDMS is the stabilization damping coefficient.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2 --

FDMD and FDMS are the destabilizing and stabilization damping coefficients.

Table 174.1: CONTA174 Real Constants

No.	Name	Description	For more information, see this section in the *Contact Technology Guide* . . .
1	R1	Target radius for cylinder, cone, or sphere	Defining the Target Surface
2	R2	Target radius at second node of cone	Defining the Target Surface
3	FKN	Normal penalty stiffness factor [1] [2]	Determining Contact Stiffness and Penetration
4	FTOLN	Penetration tolerance factor	Determining Contact Stiffness and Penetration
5	ICONT	Initial contact closure	Adjusting Initial Contact Conditions
6	PINB	Pinball region	Determining Contact Status and the Pinball Region or Defining Influence Range (PINB)
7	PMAX	Upper limit of initial allowable penetration	Adjusting Initial Contact Conditions
8	PMIN	Lower limit of initial allowable penetration	Adjusting Initial Contact Conditions
9	TAUMAX	Maximum friction stress [1] [2]	Choosing a Friction Model
10	CNOF	Contact surface offset [1]	Adjusting Initial Contact Conditions
11	FKOP	Contact opening stiffness [1] [2]	Selecting Surface Interaction Models
12	FKT	Tangent penalty stiffness factor [1] [2]	Determining Contact Stiffness
13	COHE	Contact cohesion	Choosing a Friction Model
14	TCC	Thermal contact conductance [1] [2]	Modeling Conduction
15	FHTG	Frictional heating factor	Modeling Heat Generation Due to Friction
16	SBCT	Stefan-Boltzmann constant	Modeling Radiation
17	RDVF	Radiation view factor [1] [2]	Modeling Radiation

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

No.	Name	Description	For more information, see this section in the *Contact Technology Guide* . . .
18	FWGT	Heat distribution weighing factor	Modeling Heat Generation Due to Friction (thermal) or Heat Generation Due to Electric Current (electric)
19	ECC	Electric contact conductance [1] [2]	Modeling Surface Interaction
20	FHEG	Joule dissipation weight factor	Heat Generation Due to Electric Current
21	FACT	Static/dynamic ratio	Static and Dynamic Friction Coefficients
22	DC	Exponential decay coefficient	Static and Dynamic Friction Coefficients
23	SLTO	Allowable elastic slip	Using FKT and SLTO
24	TNOP	Maximum allowable tensile contact pressure	Chattering Control Parameters
25	TOLS	Target edge extension factor	Selecting Location of Contact Detection
26	MCC	Magnetic contact permeance [1] [2]	Modeling Magnetic Contact
27	PPCN	Pressure penetration criterion [1] [2]	Specifying a Pressure Penetration Criterion
28	FPAT	Fluid penetration acting time	Specifying a Fluid Penetration Acting Time
29	COR	Coefficient of restitution	Impact Between Rigid Bodies
30	STRM	Load step number for ramping penetration	Adjusting Initial Contact Conditions
31	FDMN	Normal stabilization damping factor [1] [2]	Applying Contact Stabilization Damping
32	FDMT	Tangential stabilization damping factor [1] [2]	Applying Contact Stabilization Damping
33	FDMD	Destabilization squeal damping factor	Forced Frictional Sliding Using Velocity Input
34	FDMS	Stabilization squeal damping factor	Forced Frictional Sliding Using Velocity Input
35	TBND	Critical bonding temperature [1] [2]	Using TBND

1. This real constant can be defined as a function of certain primary variables.

2. This real constant can be defined by the user subroutine USERCNPROP.F.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CONTA174 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 174.2: CONTA174 Element Output Definitions (p. 712)

A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 174.2: CONTA174 Element Output Definitions (p. 712) gives element output. In the results file, the nodal results are obtained from its closest integration point.

Table 174.2: CONTA174 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes I, J, K, L, M, N, O, P	Y	Y
XC, YC, ZC	Location where results are reported	Y	5
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	Y	Y
VOLU	AREA	Y	Y
NPI	Number of integration points	Y	-
ITRGET	Target surface number (assigned by ANSYS)	Y	-
ISOLID	Underlying solid or shell element number	Y	-
CONT:STAT	Current contact statuses	1	1
OLDST	Old contact statuses	1	1
ISEG	Current contacting target element number	Y	Y
OLDSEG	Underlying old target number	Y	-
CONT:PENE	Current penetration (gap = 0; penetration = positive value)	Y	Y
CONT:GAP	Current gap (gap = negative value; penetration = 0)	Y	Y
NGAP	New or current gap at current converged substep (gap = negative value; penetration = positive value)	Y	-
OGAP	Old gap from previously converged substep (gap = negative value; penetration = positive value)	Y	-
IGAP	Initial gap at start of current substep (gap = negative value; penetration = positive value)	Y	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
GGAP	Geometric gap at current converged substep (gap = negative value; penetration = positive value)	-	Y
CONT:PRES	Normal contact pressure	Y	Y
TAUR/TAUS[7]	Tangential contact stresses	Y	Y
KN	Current normal contact stiffness (Force/Length^3)	Y	Y
KT	Current tangent contact stiffness (Force/Length^3)	Y	Y
MU[8]	Friction coefficient	Y	Y
TASS/TASR[7]	Total (algebraic sum) sliding in S and R directions	3	3
AASS/AASR[7]	Total (absolute sum) sliding in S and R directions	3	3
TOLN	Penetration tolerance	Y	Y
CONT:SFRIC	Frictional stress, SQRT (TAUR**2+TAUS**2)	Y	Y
CONT:STOTAL	Total stress, SQRT (PRES**2+TAUR**2+TAUS**2)	Y	Y
CONT:SLIDE	Amplitude of total accumulated sliding, SQRT (TASS**2 + TASR**2)	3	3
FDDIS	Frictional energy dissipation	6	6
ELSI	Total equivalent elastic slip distance	-	Y
PLSI	Total (accumulated) equivalent plastic slip due to frictional sliding	-	Y
GSLID	Amplitude of total accumulated sliding (including near-field)	-	9
VREL	Equivalent sliding velocity (slip rate)	-	Y
DBA	Penetration variation	Y	Y
PINB	Pinball Region	-	Y
CONT:CNOS	Total number of contact status changes during substep	Y	Y
TNOP	Maximum allowable tensile contact pressure	Y	Y
SLTO	Allowable elastic slip	Y	Y
CAREA	Contacting area	-	Y
CONT:FPRS	Actual applied fluid penetration pressure	-	Y
FSTART	Fluid penetration starting time	-	Y
DTSTART	Load step time during debonding	Y	Y
DPARAM	Debonding parameter	Y	Y
DENERI	Energy released due to separation in normal direction - mode I debonding	Y	Y
DENERII	Energy released due to separation in tangential direction - mode II debonding	Y	Y
CNFX	Contact element force-X component	-	4
CNFY	Contact element force-Y component	-	Y
CNFZ	Contact element force-Z component	-	Y
SDAMP	Squeal damping coefficient / Stabilization damping coefficient	-	Y
CONV	Convection coefficient	Y	Y
RAC	Radiation coefficient	Y	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
TCC	Conductance coefficient	Y	Y
TEMPS	Temperature at contact point	Y	Y
TEMPT	Temperature at target surface	Y	Y
FXCV	Heat flux due to convection	Y	Y
FXRD	Heat flux due to radiation	Y	Y
FXCD	Heat flux due to conductance	Y	Y
CONT:FLUX	Total heat flux at contact surface	Y	Y
FXNP	Flux input	-	Y
CNFH	Contact element heat flow	-	Y
JCONT	Contact current density (Current/Unit Area)	Y	Y
CCONT	Contact charge density (Charge/Unit Area)	Y	Y
HJOU	Contact power/area	Y	Y
ECURT	Current per contact element	-	Y
ECHAR	Charge per contact element	-	Y
ECC	Electric contact conductance (for electric current DOF), or electric contact capacitance per unit area (for piezoelectric or electrostatic DOFs)	Y	Y
VOLTS	Voltage on contact nodes	Y	Y
VOLTT	Voltage on associated target	Y	Y
MCC	Magnetic contact permeance	Y	Y
MFLUX	Magnetic flux density	Y	Y
MAGS	Magnetic potential on contact node	Y	Y
MAGT	Magnetic potential on associated target	Y	Y
WEARX, WEARY, WEARZ	Wear correction - X, Y, and Z components	-	Y

1. The possible values of *STAT* and *OLDST* are:

   0 = Open and not near contact
   1 = Open but near contact
   2 = Closed and sliding
   3 = Closed and sticking

2. ANSYS will evaluate model to detect initial conditions.

3. Only accumulates the sliding for sliding and closed contact (STAT = 2,3).

4. Contact element forces are defined in the global Cartesian system.

5. Available only at centroid as a ***GET** item.

6. FDDIS = (contact friction stress)*(sliding distance of substep)/(time increment of substep)

7. For the case of orthotropic friction, components are defined in the global Cartesian system (default) or in the local element coordinate system specified by **ESYS**.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. For orthotropic friction, an equivalent coefficient of friction is output.

9. Accumulated sliding distance for near-field, sliding, and closed contact (STAT = 1,2,3).

Note

If **ETABLE** is used for the CONT items, the reported data is averaged across the element.

Note

Contact results (including all element results) are generally not reported for elements that have a status of "open and not near contact" (far-field).

Table 174.3: CONTA174 Item and Sequence Numbers (p. 715) lists output available through the **ETABLE** command using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 174.3: CONTA174 Item and Sequence Numbers (p. 715):

Name
output quantity as defined in the Table 174.2: CONTA174 Element Output Definitions (p. 712)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J,K,L
sequence number for data at nodes I,J,K,L,

Table 174.3: CONTA174 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
PRES	SMISC	13	1	2	3	4
TAUR	SMISC	-	5	6	7	8
TAUS	SMISC	-	9	10	11	12
FLUX	SMISC	-	14	15	16	17
FDDIS	SMISC	-	18	19	20	21
FXCV	SMISC		22	23	24	25
FXRD	SMISC	-	26	27	28	29
FXCD	SMISC	-	30	31	32	33
FXNP	SMISC	-	34	35	36	37
JCONT	SMISC	-	38	39	40	41
CCONT	SMISC	-	38	39	40	41
HJOU	SMISC	-	42	43	44	45

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
MFLUX	SMISC	-	46	47	48	49
STAT[1]	NMISC	41	1	2	3	4
OLDST	NMISC	-	5	6	7	8
PENE[2]	NMISC	-	9	10	11	12
DBA	NMISC	-	13	14	15	16
TASR	NMISC	-	17	18	19	20
TASS	NMISC	-	21	22	23	24
KN	NMISC	-	25	26	27	28
KT	NMISC	-	29	30	31	32
TOLN	NMISC	-	33	34	35	36
IGAP	NMISC	-	37	38	39	40
PINB	NMISC	42	-	-	-	-
CNFX	NMISC	43	-	-	-	-
CNFY	NMISC	44	-	-	-	-
CNFZ	NMISC	45	-	-	-	-
ISEG	NMISC	-	46	47	48	49
AASR	NMISC	-	50	51	52	53
AASS	NMISC	-	54	55	56	57
CAREA	NMISC	58	-	-	-	-
MU	NMISC	-	62	63	64	65
DTSTART	NMISC	-	66	67	68	69
DPARAM	NMISC	-	70	71	72	73
FPRS	NMISC	-	74	75	76	77
TEMPS	NMISC	-	78	79	80	81
TEMPT	NMISC	-	82	83	84	85
CONV	NMISC	-	86	87	88	89
RAC	NMISC	-	90	91	92	93
TCC	NMISC	-	94	95	96	97
CNFH	NMISC	98	-	-	-	-
ECURT	NMISC	99	-	-	-	-
ECHAR	NMISC	99	-	-	-	-
ECC	NMISC	-	100	101	102	103
VOLTS	NMISC	-	104	105	106	107
VOLTT	NMISC	-	108	109	110	111
CNOS	NMISC	-	112	113	114	115
TNOP	NMISC	-	116	117	118	119

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quant-ity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
SLTO	NMISC	-	120	121	122	123
MCC	NMISC	-	124	125	126	127
MAGS	NMISC	-	128	129	130	131
MAGT	NMISC	-	132	133	134	135
ELSI	NMISC	-	136	137	138	139
DENERI	NMISC	-	140	141	142	143
DENERII	NMISC	-	144	145	146	147
FSTART	NMISC	-	148	149	150	151
GGAP	NMISC	-	152	153	154	155
VREL	NMISC	-	156	157	158	159
SDAMP	NMISC	-	160	161	162	163
PLSI	NMISC	-	164	165	166	167
GSLID	NMISC	-	168	169	170	171
WEARX	NMISC	-	172	173	174	175
WEARY	NMISC	-	176	177	178	179
WEARZ	NMISC	-	180	181	182	183

1. Element Status = highest value of status of integration points within the element

2. Penetration = positive value, gap = negative value

You can display or list contact results through several POST1 postprocessor commands. The contact specific items for the **PLNSOL**, **PLESOL**, **PRNSOL**, and **PRESOL** commands are listed below:

STAT	Contact status
PENE	Contact penetration
PRES	Contact pressure
SFRIC	Contact friction stress
STOT	Contact total stress (pressure plus friction)
SLIDE	Contact sliding distance
GAP	Contact gap distance
FLUX	Total heat flux at contact surface
CNOS	Total number of contact status changes during substep
FPRS	Actual applied fluid penetration pressure

CONTA174 Assumptions and Restrictions

- The 3-D contact element must coincide with the external surface of the underlying solid or shell element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- This element is nonlinear and requires a full Newton iterative solution, regardless of whether large or small deflections are specified. An exception to this is when MPC bonded contact is specified (KEYOPT(2) = 2 and KEYOPT(12) = 5 or 6).
- The normal contact stiffness factor (FKN) must not be so large as to cause numerical instability.
- FTOLN, PINB, and FKOP can be changed between load steps or during restart stages.
- You can use this element in nonlinear static or nonlinear full transient analyses.
- In addition, you can use it in modal analyses, eigenvalue buckling analyses, and harmonic analyses. For these analysis types, the program assumes that the initial status of the element (that is, the status at the completion of the static prestress analysis, if any) does not change.

CONTA174 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The MU material property is not allowed in the Nonlinear Thermal product.
- The birth and death special feature is not allowed in the Nonlinear Structural and Nonlinear Thermal products.

ANSYS Structural

- The VOLT DOF (KEYOPT(1) = 3 through 6) is not allowed.
- The MAG DOF (KEYOPT(1) = 7) is not allowed.

ANSYS Mechanical

- The MAG DOF (KEYOPT(1) = 7) is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CONTA175

2-D/3-D Node-to-Surface Contact

MP ME ST PR PRN DS EM <> PP EME MFS
Product Restrictions

CONTA175 Element Description

CONTA175 may be used to represent contact and sliding between two surfaces (or between a node and a surface, or between a line and a surface) in 2-D or 3-D. The element is applicable to 2-D or 3-D structural and coupled field contact analyses. This element is located on the surfaces of solid, beam, and shell elements. 3-D solid and shell elements with midside nodes are supported for bonded and no separation contact. For other contact types, lower order solid and shell elements are recommended.

Contact occurs when the element surface penetrates one of the target segment elements (TARGE169, TARGE170) on a specified target surface. Coulomb friction, shear stress friction, user-defined friction with the `USERFRIC` subroutine, and user-defined contact interaction with the `USERINTER` subroutine are allowed. This element also allows separation of bonded contact to simulate interface delamination. See CONTA175 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 175.1: CONTA175 Geometry

CONTA175 Input Data

The geometry is shown in Figure 175.1: CONTA175 Geometry (p. 719). The element is defined by one node. The underlying elements can be 2-D or 3-D solid, shell, or beam elements. The 3-D underlying solid or shell elements must have no midside nodes. CONTA175 represents 2-D or 3-D contact depending on whether the associated 2-D (TARGE169) or 3-D (TARGE170) segments are used. Remember, contact can occur only when the outward normal direction of the 2-D or 3-D target surface points to the contact surface. See Generating Contact Elements in the *Contact Technology Guide* for more information on generating elements automatically using the **ESURF** command.

The contact surface elements are associated with the target segment elements (TARGE169, TARGE170) via a shared real constant set. ANSYS looks for contact only between surfaces with the same real constant set. For either rigid-flexible or flexible-flexible contact, one of the deformable surfaces must be represented by a contact surface. See Designating Contact and Target Surfaces in the *Contact Technology Guide* for more information.

If more than one target surface will make contact with the same boundary of solid elements, you must define several contact elements that share the same geometry but relate to separate targets (targets

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

which have different real constant numbers), or you must combine the two target surfaces into one (targets that share the same real constant numbers).

CONTA175 supports isotropic and orthotropic Coulomb friction. For isotropic friction, specify a single coefficient of friction, MU, using either **TB** command input (recommended) or the **MP** command. For orthotropic friction, specify two coefficients of friction, MU1 and MU2, in two principal directions using **TB** command input. (See Contact Friction in the *Material Reference* for more information.)

For isotropic friction, the default element coordinate system (based on node connectivity of the underlying elements) is used. For orthotropic friction, the global coordinate system is used by default, or you may define a local element coordinate system with the **ESYS** command. The principal directions are computed on the target surface and then projected onto the contact element (node). The first principal direction is defined by projecting the first direction of the chosen coordinate system onto the target surface. The second principal direction is defined by taking a cross product of the first principal direction and the target normal. These directions also follow the rigid body rotation of the contact element to correctly model the directional dependence of friction. Be careful to choose the coordinate system (global or local) so that the first direction of that system is within 45° of the tangent to the contact surface.

If you want to set the coordinate directions for isotropic friction (to the global Cartesian system or another system via **ESYS**), you can define orthotropic friction and set MU1 = MU2.

To define a coefficient of friction for isotropic or orthotropic friction that is dependent on temperature, time, normal pressure, sliding distance, or sliding relative velocity, use the **TBFIELD** command along with **TB**,FRIC. See Contact Friction in the *Material Reference* for more information.

To implement a user-defined friction model, use the **TB**,FRIC command with *TBOPT* = USER to specify friction properties and write a `USERFRIC` subroutine to compute friction forces. See Writing Your Own Friction Law (USERFRIC) in the *Mechanical APDL Contact Technology Guide* for more information on how to use this feature. See also the *Guide to User-Programmable Features* for a detailed description of the `USERFRIC` subroutine.

In addition to the user-defined friction subroutine, the contact interaction subroutine `USERINTER` is available for user-defined interface interactions, including interactions in the normal and tangential directions as well coupled-field interactions. See Defining Your Own Contact Interaction (USERINTER) in the *Mechanical APDL Contact Technology Guide* for more information on how to use this feature. See also the *Guide to User-Programmable Features* for a detailed description of the `USERINTER` subroutine.

To model proper momentum transfer and energy balance between contact and target surfaces, impact constraints should be used in transient dynamic analysis. See the description of KEYOPT(7) below and the contact element discussion in the *Mechanical APDL Theory Reference* for details.

To model separation of bonded contact with KEYOPT(12) = 2, 3, 4, 5, or 6, use the **TB** command with the CZM label. See Debonding in the *Contact Technology Guide* for more information.

To model wear at the contact surface, use the **TB** command with the WEAR label. See Contact Surface Wear in the *Contact Technology Guide* for more information.

KEYOPT(3) allows you to choose between a contact force-based model (KEYOPT(3) = 0, default) and a contact traction-based model (KEYOPT(3) = 1). The units for certain real constants (FKN, FKT, TNOP, and so on) and postprocessing items (PRES, TAUR, TAUS, SFRIC, and so on) vary by a factor of AREA, depending on which model is specified. (For details, see the real constant table and output definitions table.) For more information on using KEYOPT(3) with CONTA175, see KEYOPT(3) in the *Contact Technology Guide*.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This element supports a bolt thread modeling technique that simulates contact between a threaded bolt and bolt hole without having to model the detailed thread geometry. Bolt thread modeling is available for 3-D models and 2-D axisymmetric models and is implemented through section definitions (**SECTYPE**, **SECDATA**, and **SECNUM** commands). For more information, see Simplified Bolt Thread Modeling in the *Contact Technology Guide*.

See the *Contact Technology Guide* for a detailed discussion on contact and using the contact elements. Node-to-Surface Contact discusses CONTA175 specifically, including the use of real constants and KEYOPTs.

A summary of the element input is given in "CONTAC175 Input Summary" (p. 721). A general description of element input is given in Element Input (p. 43).

CONTAC175 Input Summary

Nodes

I

Degrees of Freedom

UX, UY, (UZ) (if KEYOPT(1) = 0
UX, UY, (UZ), TEMP (if KEYOPT(1) = 1)
TEMP (if KEYOPT(1) = 2)
UX, UY, (UZ), TEMP, VOLT (if KEYOPT(1) = 3)
TEMP, VOLT (if KEYOPT(1) = 4)
UX, UY, (UZ), VOLT (if KEYOPT(1) = 5)
VOLT (if KEYOPT(1) = 6)
AZ (2-D), MAG (3-D) (if KEYOPT(1) = 7)

Real Constants

R1, R2, FKN, FTOLN, ICONT, PINB,
PMAX, PMIN, TAUMAX, CNOF, FKOP, FKT,
COHE, TCC, FHTG, SBCT, RDVF, FWGT,
ECC, FHEG, FACT, DC, SLTO, TNOP,
TOLS, MCC, , , COR, STRM
FDMN, FDMT, FDMD, FDMS, TBND
See Table 175.1: CONTA175 Real Constants (p. 727) for descriptions of the real constants.

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: MU, EMIS

Special Features

Birth and death
Debonding
Isotropic friction
Large deflection
Linear perturbation
Nonlinearity
Orthotropic friction
User-defined contact interaction

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

User-defined friction

KEYOPTs

Presented below is a list of KEYOPTS available for this element. Included are links to sections in the *Contact Technology Guide* where more information is available on a particular topic.

KEYOPT(1)

Selects degrees of freedom:

0 --
UX, UY, UZ

1 --
UX, UY, UZ, TEMP

2 --
TEMP

3 --
UX, UY, UZ, TEMP, VOLT

4 --
TEMP, VOLT

5 --
UX, UY, UZ, VOLT

6 --
VOLT

7 --
AZ (2-D) or MAG (3-D)

KEYOPT(2)

Contact algorithm:

0 --
Augmented Lagrangian (default)

1 --
Penalty function

2 --
Multipoint constraint (MPC); see Multipoint Constraints and Assemblies in the *Contact Technology Guide* for more information

3 --
Lagrange multiplier on contact normal and penalty on tangent

4 --
Pure Lagrange multiplier on contact normal and tangent

KEYOPT(3)

Contact model:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0 --
Contact force-based model (default)

1 --
Contact traction-based model

Note

The traction-based model (KEYOPT(3) = 1) should not be used if the underlying elements are 3-D beam or pipe elements.

KEYOPT(4)

Contact normal direction:

0 --
Normal to target surface (default)

1 --
Normal from contact nodes

2 --
Normal from contact nodes (used for shell/beam bottom surface contact when shell/beam thickness is accounted for; KEYOPT(11) = 1)

3 --
Normal to target surface (used for shell/beam bottom surface contact when shell/beam thickness is accounted for; KEYOPT(11) = 1)

Note

When using the multipoint constraint (MPC) approach to define surface-based constraints, use KEYOPT(4) in the following way: set KEYOPT(4) = 0 for a rigid surface constraint; set KEYOPT(4) = 1 for a force-distributed constraint; set KEYOPT(4) = 3 for a coupling constraint. See Surface-based Constraints for more information.

KEYOPT(5)

CNOF/ICONT Automated adjustment:

0 --
No automated adjustment

1 --
Close gap with auto CNOF

2 --
Reduce penetration with auto CNOF

3 --
Close gap/reduce penetration with auto CNOF

4 --
Auto ICONT

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(6)

Contact stiffness variation (used to enhance stiffness updating when KEYOPT(10) > 0):

0 --
Use default range for stiffness updating

1 --
Make a nominal refinement to the allowable stiffness range

2 --
Make an aggressive refinement to the allowable stiffness range

KEYOPT(7)

Element level time incrementation control / impact constraints:

0 --
No control

1 --
Automatic bisection of increment

2 --
Change in contact predictions are made to maintain a reasonable time/load increment

3 --
Change in contact predictions made to achieve the minimum time/load increment whenever a change in contact status occurs

4 --
Use impact constraints for standard or rough contact (KEYOPT(12) = 0 or 1) in a transient dynamic analysis with automatic adjustment of time increment

---

Note

KEYOPT(7) = 2, 3, and 4 include an automatic adjustment of the time increment. This is activated only if the command **SOLCONTROL**,ON,ON was issued prior to the solution.

KEYOPT(8)

Asymmetric contact selection:

0 --
No action

2 --
ANSYS internally selects which asymmetric contact pair is used at the solution stage (used only when symmetry contact is defined).

KEYOPT(9)

Effect of initial penetration or gap:

0 --
Include both initial geometrical penetration or gap and offset

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
Exclude both initial geometrical penetration or gap and offset

2 --
Include both initial geometrical penetration or gap and offset, but with ramped effects

3 --
Include offset only (exclude initial geometrical penetration or gap)

4 --
Include offset only (exclude initial geometrical penetration or gap), but with ramped effects

5 --
Include offset only (exclude initial geometrical penetration or gap) regardless of the initial contact status (near-field or closed)

6 --
Include offset only (exclude initial geometrical penetration or gap), but with ramped effects regardless of the initial contact status (near-field or closed)

Note

The effects of KEYOPT(9) are dependent on settings for other KEYOPTs. The indicated initial gap effect is considered only if KEYOPT(12) = 4 or 5. See the discussion on using KEYOPT(9) in the *Contact Technology Guide* for more information.

KEYOPT(10)
Contact Stiffness Update:

0 --
Each load step if FKN is redefined during load step (pair based).

2 --
Each iteration based on current mean stress of underlying elements (pair based).

KEYOPT(11)
Shell Thickness Effect:

0 --
Exclude

1 --
Include

Note

KEYOPT(11) is applicable to shell elements whose thickness is defined through real constant input or section properties. It is also applicable to beam elements whose thickness is defined through real constant input. However, KEYOPT(11) = 1 is not valid if the thickness of 2-D/3-D beams is defined through section properties.

KEYOPT(12)
Behavior of contact surface:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0 --
Standard

1 --
Rough

2 --
No separation (sliding permitted)

3 --
Bonded

4 --
No separation (always)

5 --
Bonded (always)

6 --
Bonded (initial contact)

Note

When KEYOPT(12) = 5 or 6 is used with the MPC algorithm to model surface-based constraints, the KEYOPT(12) setting will have an impact on the local coordinate system of the contact element nodes. See Specifying a Local Coordinate System in the *Contact Technology Guide* for more information.

KEYOPT(15)
Effect of contact stabilization damping:

0 --
Damping is activated only in the first load step (default).

1 --
Deactivate automatic damping.

2 --
Damping is activated for all load steps.

3 --
Damping is activated at all times regardless of the contact status of previous substeps.

Note

Stabilization damping is only applied to contact pairs in near-field contact. When KEYOPT(15) = 0, 1, or 2, stabilization damping will not be applied in the current substep if any contact detection point had a closed status in the previous substep. However, when KEYOPT(15) = 3, stabilization damping is always applied as long as the current contact status is near-field. See Applying Contact Stabilization Damping in the *Contact Technology Guide* for more information.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(16)

Squeal damping controls for interpretation of real constants FDMD and FDMS:

0 --

FDMD and FDMS are scaling factors for destabilizing and stabilizing damping (default).

1 --

FDMD is a constant friction-sliding velocity gradient. FDMS is the stabilization damping coefficient.

2 --

FDMD and FDMS are the destabilizing and stabilization damping coefficients.

Table 175.1: CONTA175 Real Constants

No.	Name	Description	For more information, see this section in the *Contact Technology Guide* . . .
1	R1	Target radius for cylinder, cone, or sphere	Defining the Target Surface
2	R2	Target radius at second node of cone	Defining the Target Surface
3	FKN[4]	Normal penalty stiffness factor [1] [2]	Determining Contact Stiffness and Penetration
4	FTOLN	Penetration tolerance factor	Determining Contact Stiffness and Penetration
5	ICONT	Initial contact closure	Adjusting Initial Contact Conditions
6	PINB	Pinball region	Determining Contact Status and the Pinball Region or Defining Influence Range (PINB)
7	PMAX	Upper limit of initial allowable penetration	Adjusting Initial Contact Conditions
8	PMIN	Lower limit of initial allowable penetration	Adjusting Initial Contact Conditions
9	TAUMAX	Maximum friction stress [1] [2]	Choosing a Friction Model
10	CNOF	Contact surface offset [1]	Adjusting Initial Contact Conditions
11	FKOP	Contact opening stiffness [1] [2]	Selecting Surface Interaction Models
12	FKT[4]	Tangent penalty stiffness factor [1] [2]	Determining Contact Stiffness

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

No.	Name	Description	For more information, see this section in the ***Contact Technology Guide*** . . .
13	COHE	Contact cohesion	Choosing a Friction Model
14	TCC[4]	Thermal contact conductance [1] [2]	Modeling Conduction
15	FHTG	Frictional heating factor	Modeling Heat Generation Due to Friction
16	SBCT	Stefan-Boltzmann constant	Modeling Radiation
17	RDVF	Radiation view factor [1] [2]	Modeling Radiation
18	FWGT	Heat distribution weighing factor	Modeling Heat Generation Due to Friction (thermal) or Heat Generation Due to Electric Current (electric)
19	ECC[4]	Electric contact conductance [1] [2]	Modeling Surface Interaction
20	FHEG	Joule dissipation weight factor	Heat Generation Due to Electric Current
21	FACT	Static/dynamic ratio	Static and Dynamic Friction Coefficients
22	DC	Exponential decay coefficient	Static and Dynamic Friction Coefficients
23	SLTO	Allowable elastic slip	Using FKT and SLTO
24	TNOP	Maximum allowable tensile contact force/pressure [3]	Chattering Control Parameters
25	TOLS	Target edge extension factor	Selecting Location of Contact Detection
26	MCC[4]	Magnetic contact permeance [1] [2]	Modeling Magnetic Contact
29	COR	Coefficient of restitution	Impact Between Rigid Bodies
30	STRM	Load step number for ramping penetration	Adjusting Initial Contact Conditions
31	FDMN	Normal stabilization damping factor [1] [2]	Applying Contact Stabilization Damping
32	FDMT	Tangential stabilization damping factor [1] [2]	Applying Contact Stabilization Damping
33	FDMD	Destabilization squeal damping factor	Forced Frictional Sliding Using Velocity Input
34	FDMS	Stabilization squeal damping factor	Forced Frictional Sliding Using Velocity Input

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

No.	Name	Description	For more information, see this section in the *Contact Technology Guide* . . .
35	TBND	Critical bonding temperature [1] [2]	Using TBND

1. This real constant can be defined as a function of certain primary variables.

2. This real constant can be defined by the user subroutine USERCNPROP.F.

3. For the contact force-based model (KEYOPT(3) = 0), TNOP is the allowable tensile contact force. For the contact traction-based model (KEYOPT(3) = 0), TNOP is the allowable tensile contact pressure.

4. For the contact force-based model (KEYOPT(3) = 0), the units of this real constant has a factor of AREA with respect to those used in the surface-to-surface contact elements. See Performing a Node-to-Surface Contact Analysis for more information.

CONTA175 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution

- Additional element output as shown in Table 175.2: CONTA175 Element Output Definitions (p. 729).

A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 175.2: CONTA175 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes I	Y	Y
XC, YC, (ZC)	Location where results are reported (same as nodal location)	Y	Y
TEMP	Temperature T(I)	Y	Y
VOLU	AREA for 3-D, Length for 2-D	Y	Y
NPI	Number of integration points	Y	-
ITRGET	Target surface number (assigned by ANSYS)	Y	-
ISOLID	Underlying solid or shell element number	Y	-
CONT:STAT	Current contact statuses	1	1
OLDST	Old contact statuses	1	1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
ISEG	Current contacting target element number	Y	Y
OLDSEG	Underlying old target number	Y	-
CONT:PENE	Current penetration (gap = 0; penetration = positive value)	Y	Y
CONT:GAP	Current gap (gap = negative value; penetration = 0)	Y	Y
NGAP	New or current gap at current converged substep (gap = negative value; penetration = positive value)	Y	-
OGAP	Old gap at previously converged substep (gap = negative value; penetration = positive value)	Y	-
IGAP	Initial gap at start of current substep (gap = negative value; penetration = positive value)	Y	Y
GGAP	Geometric gap at current converged substep (gap = negative value; penetration = positive value)	-	Y
CONT:PRES	Normal contact force/pressure	2	2
TAUR/TAUS[8]	Tangential contact forces/stresses	2	2
KN	Current normal contact stiffness (units: Force/Length for contact force-based model, Force/Length^3 for contract traction-based model)	5	5
KT	Current tangent contact stiffness (same units as KN)	5	5
MU[9]	Friction coefficient	Y	Y
TASS/TASR[8]	Total (algebraic sum) sliding in S and R directions (3-D only)	3	3
AASS/AASR[8]	Total (absolute sum) sliding in S and R directions (3-D only)	3	3
TOLN	Penetration tolerance	Y	Y
CONT:SFRIC	Frictional force/stress, SQRT (TAUR**2+TAUS**2) (3-D only)	2	2
CONT:STOTAL	Total force/stress, SQRT (PRES**2+TAUR**2+TAUS**2) (3-D only)	2	2
CONT:SLIDE	Amplitude of total accumulated sliding, SQRT (TASS**2+TASR**2) (3-D only)	3	3
NX, NY	Surface normal vector components (2-D only)	Y	-
CONT:SFRIC	Tangential contact force/stress (2-D only)	2	2
CONT:SLIDE	Total accumulated sliding (algebraic sum) (2-D only)	3	3
ASLIDE	Total accumulated sliding (absolute sum) (2-D only)	3	3
FDDIS	Frictional energy dissipation	7	7
ELSI	Total equivalent elastic slip distance	-	Y
PLSI	Total (accumulated) equivalent plastic slip due to frictional sliding	-	Y
GSLID	Amplitude of total accumulated sliding (including near-field)	-	10
VREL	Equivalent sliding velocity (slip rate)	-	Y
DBA	Penetration variation	Y	Y
PINB	Pinball Region	-	Y
CONT:CNOS	Total number of contact status changes during substep	Y	Y
TNOP	Maximum allowable tensile contact force/pressure	2	2

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
SLTO	Allowable elastic slip	Y	Y
CAREA	Contacting area	-	Y
DTSTART	Load step time during debonding	Y	Y
DPARAM	Debonding parameter	Y	Y
DENERI	Energy released due to separation in normal direction - mode I debonding	Y	Y
DENERII	Energy released due to separation in tangential direction - mode II debonding	Y	Y
CNFX[4]	Contact element force-X component	-	Y
CNFY	Contact element force-Y component	-	Y
CNFZ	Contact element force-Z component (3-D only)	-	Y
SDAMP	Squeal damping coefficient (3-D only) / Stabilization damping coefficient (2-D and 3-D)	-	Y
CONV	Convection coefficient	Y	Y
RAC	Radiation coefficient	Y	Y
TCC	Conductance coefficient	6	6
TEMPS	Temperature at contact point	Y	Y
TEMPT	Temperature at target surface	Y	Y
FXCV	Heat flux due to convection	Y	Y
FXRD	Heat flux due to radiation	Y	Y
FXCD	Heat flux due to conductance	Y	Y
CONT:FLUX	Total heat flux at contact surface	Y	Y
FXNP	Flux input	-	Y
CNFH	Contact element heat flow	-	Y
JCONT	Contact current density (Current/Unit Area)	Y	Y
CCONT	Contact charge density (Charge/Unit Area)	Y	Y
HJOU	Contact power/area	Y	Y
ECURT	Current per contact element	-	Y
ECHAR	Charge per contact element	-	Y
ECC	Electric contact conductance (for electric current DOF), or electric contact capacitance per unit area (for piezoelectric or electrostatic DOFs)	6	6
VOLTS	Voltage on contact nodes	Y	Y
VOLTT	Voltage on associated target	Y	Y
MCC	Magnetic contact permeance	6	6
MFLUX	Magnetic flux density	Y	Y
AZS/MAGS	2-D/3-D Magnetic potential on contact node	Y	Y
AZT/MAGT	2-D/3-D Magnetic potential on associated target	Y	Y
WEARX, WEARY, WEARZ	Wear correction - X, Y, and Z components	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. The possible values of *STAT* and *OLDST* are:

   0 = Open and not near contact
   1 = Open but near contact
   2 = Closed and sliding
   3 = Closed and sticking

2. For the force-based model (KEYOPT(3) = 0), the unit of this quantity is FORCE. For the traction-based model (KEYOPT(3) = 1), the unit is FORCE/AREA.

3. Only accumulates the sliding for sliding and closed contact (STAT = 2,3).

4. Contact element forces are defined in the global Cartesian system

5. For the force-based model, the unit of stiffness is FORCE/LENGTH. For the traction-based model, the unit is FORCE/LENGTH^3.

6. The units of TCC, ECC, and MCC in the traction-based model should be the units of TCC, ECC, and MCC of the force-based model per area.

7. FDDIS = (contact friction stress)*(sliding distance of substep)/(time increment of substep)

8. For the case of orthotropic friction, components are defined in the global Cartesian system (default) or in the local element coordinate system specified by **ESYS**.

9. For orthotropic friction, an equivalent coefficient of friction is output.

10. Accumulated sliding distance for near-field, sliding, and closed contact (STAT = 1,2,3).

Note

Contact results (including all element results) are generally not reported for elements that have a status of "open and not near contact" (far-field).

Table 175.3: CONTA175 (3-D) Item and Sequence Numbers (p. 733) and Table 175.4: CONTA175 (2-D) Item and Sequence Numbers (p. 734) list outputs available through the **ETABLE** command using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in the tables below:

Name
output quantity as defined in Table 175.2: CONTA175 Element Output Definitions (p. 729)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

I

sequence number for data at nodes I

Table 175.3: CONTA175 (3-D) Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	E	I
PRES	SMISC	13	1
TAUR	SMISC	-	5
TAUS	SMISC	-	9
FLUX	SMISC	-	14
FDDIS	SMISC	-	18
FXCV	SMISC		22
FXRD	SMISC	-	26
FXCD	SMISC	-	30
FXNP	SMISC	-	34
JCONT	SMISC	-	38
CCONT	SMISC	-	38
HJOU	SMISC	-	42
MFLUX	SMISC	-	46
STAT[1]	NMISC	41	1
OLDST	NMISC	-	5
PENE[2]	NMISC	-	9
DBA	NMISC	-	13
TASR	NMISC	-	17
TASS	NMISC	-	21
KN	NMISC	-	25
KT	NMISC	-	29
TOLN	NMISC	-	33
IGAP	NMISC	-	37
PINB	NMISC	42	-
CNFX	NMISC	43	-
CNFY	NMISC	44	-
CNFZ	NMISC	45	-
ISEG	NMISC	-	46
AASR	NMISC	-	50
AASS	NMISC	-	54
CAREA	NMISC	58	-
MU	NMISC	-	62
DTSTART	NMISC	-	66

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	E	I
DPARAM	NMISC	-	70
TEMPS	NMISC	-	78
TEMPT	NMISC	-	82
CONV	NMISC	-	86
RAC	NMISC	-	90
TCC	NMISC	-	94
CNFH	NMISC	98	-
ECURT	NMISC	99	-
ECHAR	NMISC	99	-
ECC	NMISC	-	100
VOLTS	NMISC	-	104
VOLTT	NMISC	-	108
CNOS	NMISC	-	112
TNOP	NMISC	-	116
SLTO	NMISC	-	120
MCC	NMISC	-	124
MAGS	NMISC	-	128
MAGT	NMISC	-	132
ELSI	NMISC	-	136
DENERI	NMISC	-	140
DENERII	NMISC	-	144
GGAP	NMISC	-	152
VREL	NMISC	-	156
SDAMP	NMISC	-	160
PLSI	NMISC	-	164
GSLID	NMISC	-	168
WEARX	NMISC	-	172
WEARY	NMISC	-	176
WEARZ	NMISC	-	180

Table 175.4: CONTA175 (2-D) Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	E	I
PRES	SMISC	5	1
SFRIC	SMISC	-	3

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	E	I
FLUX	SMISC	-	6
FDDIS	SMISC	-	8
FXCV	SMISC	-	10
FXRD	SMISC	-	12
FXCD	SMISC	-	14
FXNP	SMISC	-	16
JCONT	SMISC	-	18
CCONT	SMISC	-	18
HJOU	SMISC	-	20
MFLUX	SMISC	-	22
STAT[1]	NMISC	19	1
OLDST	NMISC	-	3
PENE[2]	NMISC	-	5
DBA	NMISC	-	7
SLIDE	NMISC	-	9
KN	NMISC	-	11
KT	NMISC	-	13
TOLN	NMISC	-	15
IPENE	NMISC	-	17
PINB	NMISC	20	-
CNFX	NMISC	21	-
CNFY	NMISC	22	-
ISEG	NMISC	-	23
CAREA	NMISC	27	-
MU	NMISC	-	29
DTSTART	NMISC	-	31
DPARAM	NMISC	-	33
TEMPS	NMISC	-	37
TEMPT	NMISC	-	39
CONV	NMISC	-	41
RAC	NMISC	-	43
TCC	NMISC	-	45
CNFH	NMISC	47	-
ECURT	NMISC	48	-
ECHAR	NMISC	48	-
ECC	NMISC	-	49

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	E	I
VOLTS	NMISC	-	51
VOLTT	NMISC	-	53
CNOS	NMISC	-	55
TNOP	NMISC	-	57
SLTO	NMISC	-	59
MCC	NMISC	-	61
AZS	NMISC	-	63
AZT	NMISC	-	65
ELSI	NMISC	-	67
DENERI	NMISC	-	69
DENERII	NMISC	-	71
GGAP	NMISC	-	75
VREL	NMISC	-	77
SDAMP	NMISC	-	79
PLSI	NMISC	-	81
GSLID	NMISC	-	83
WEARX	NMISC	-	85
WEARY	NMISC	-	87

1. Element Status = highest value of status of integration points within the element

2. Penetration = positive value, gap = negative value

You can display or list contact results through several POST1 postprocessor commands. The contact specific items for the **PLNSOL**, **PLESOL**, **PRNSOL**, and **PRESOL** commands are listed below:

STAT	Contact status
PENE	Contact penetration
PRES	Contact pressure for the traction-based model. Contact normal force for the force-based model.
SFRIC	Contact friction stress for the traction-based model. Friction force for the force-based model.
STOT	Contact total stress (pressure plus friction) for the traction-based model. Total contact force for the force-based model.
SLIDE	Contact sliding distance
GAP	Contact gap distance
CNOS	Total number of contact status changes during substep

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CONTA175 Assumptions and Restrictions

- This element is nonlinear and requires a full Newton iterative solution, regardless of whether large or small deflections are specified. An exception to this is when MPC bonded contact is specified (KEYOPT(2) = 2 and KEYOPT(12) = 5 or 6).
- The normal contact stiffness factor (FKN) must not be so large as to cause numerical instability.
- FTOLN, PINB, and FKOP can be changed between load steps or during restart stages.
- You can use this element in nonlinear static or nonlinear full transient analyses.
- In addition, you can use it in modal analyses, eigenvalue buckling analyses, and harmonic analyses. For these analysis types, the program assumes that the initial status of the element (i.e., the status at the completion of the static prestress analysis, if any) does not change.
- When the contact node is on the axis of symmetry in an axisymmetric analysis, the contact pressure on that node is not accurate since the area of the node is zero. The contact force is accurate in this situation.

CONTA175 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The MU material property is not allowed in the Nonlinear Thermal product.
- The birth and death special feature is not allowed in the Nonlinear Structural and Nonlinear Thermal products.

ANSYS Structural

- The VOLT DOF (KEYOPT(1) = 3 through 6) is not allowed.
- The AZ (2-D) and MAG (3-D) DOFs (KEYOPT(1) = 7) are not allowed.

ANSYS Mechanical

- The AZ (2-D) and MAG (3-D) DOFs (KEYOPT(1) = 7) are not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

CONTA176

3-D Line-to-Line Contact

MP ME ST PR PRN <> EM <> PP EME MFS
Product Restrictions

CONTA176 Element Description

CONTA176 is used to represent contact and sliding between 3-D line segments (TARGE170) and a deformable line segment, defined by this element. The element is applicable to 3-D beam-beam structural contact analyses. This element is located on the surfaces of 3-D beam or pipe elements with or without midside nodes (such as BEAM188 or BEAM189). Contact occurs when the element surface penetrates one of the 3-D straight line or parabolic line segment elements (TARGE170) on a specified target surface. Coulomb friction, shear stress friction, user-defined friction with the `USERFRIC` subroutine, and user-defined contact interaction with the `USERINTER` subroutine are allowed. This element also allows separation of bonded contact to simulate interface delamination. See CONTA176 in the *Mechanical APDL Theory Reference* for more details about this element. To model beam-to-surface contact, use the line-to-surface contact element, CONTA177.

Figure 176.1: CONTA176 Geometry

CONTA176 Input Data

The geometry and node locations are shown in Figure 176.1: CONTA176 Geometry (p. 739). The element is defined by two nodes (if the underlying beam element does not have a midside node) or three nodes (if the underlying beam element has a midside node). The element x-axis is along the I-J line of the element. Correct node ordering of the contact element is critical for proper detection of contact. The nodes must be ordered in a sequence that defines a continuous line. See Generating Contact Elements in the *Contact Technology Guide* for more information on generating elements automatically using the **ESURF** command.

Three different scenarios can be modeled by CONTA176:

- Internal contact where one beam (or pipe) slides inside another hollow beam (or pipe) (see Figure 176.2: Beam Sliding Inside a Hollow Beam (p. 740))
- External contact between two beams that lie next to each other and are roughly parallel (see Figure 176.3: Parallel Beams in Contact (p. 740))
- External contact between two beams that cross (see Figure 176.4: Crossing Beams in Contact (p. 741))

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use KEYOPT(3) = 0 for the first two scenarios (internal contact and parallel beams). In both cases, the contact condition is only checked at contact nodes.

Use KEYOPT(3) = 1 for the third scenario (beams that cross). In this case, the contact condition is checked along the entire length of the beams. The beams with circular cross sections are assumed to come in contact in a point-wise manner. Each contact element can potentially contact no more than one target element.

Figure 176.2: Beam Sliding Inside a Hollow Beam

Figure 176.3: Parallel Beams in Contact

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 176.4: Crossing Beams in Contact

The 3-D line-to-line contact elements are associated with the target line segment elements (LINE or PARA segment types for TARGE170) via a shared real constant set. The contact/target surface is assumed to be the surface of a cylinder. For a general beam cross section, use an equivalent circular beam (see Figure 176.5: Equivalent Circular Cross Section (p. 741)). Use the first real constant, R1, to define the radius on the target side (target radius r_t). Use the second real constant, R2, to define the radius on the contact side (contact radius r_c). Follow these guidelines to define the equivalent circular cross section:

- Determine the smallest cross section along the beam axis.
- Determine the largest circle embedded in that cross section.

Figure 176.5: Equivalent Circular Cross Section

The target radius can be entered as either a negative or positive value. Use a negative value when modeling internal contact (a beam sliding inside a hollow beam, or pipe sliding inside another pipe), with the input value equal to the inner radius of the outer beam (see Figure 176.2: Beam Sliding Inside a Hollow Beam (p. 740)). Use a positive value when modeling contact between the exterior surfaces of two cylindrical beams.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For the case of internal contact, the inner beam should usually be considered the contact surface and the outer beam should be the target surface. The inner beam can be considered as the target surface only when the inner beam is much stiffer than the outer beam.

Contact is detected when two circular beams touch or overlap each other. The non-penetration condition for beams with a circular cross section can be defined as follows.

For internal contact:

$$g = \left| r_t - r_c \right| - d \le 0$$

and for external contact:

$$g = d - (r_c + r_t) \le 0$$

where r_c and r_t are the radii of the cross sections of the beams on the contact and target sides, respectively; and d is the minimal distance between the two beams which also determines the contact normal direction (see Figure 176.4: Crossing Beams in Contact (p. 741)). Contact occurs for negative values of g.

ANSYS looks for contact only between contact and target surfaces with the same real constant set. For either rigid-flexible or flexible-flexible contact, one of the deformable surfaces must be represented by a contact surface. See Designating Contact and Target Surfaces in the *Contact Technology Guide* for more information. If more than one target surface will make contact with the same boundary of beam elements, you must define several contact elements that share the same geometry but relate to separate targets (targets which have different real constant numbers), or you must combine the two target surfaces into one (targets that share the same real constant numbers).

CONTA176 supports isotropic and orthotropic Coulomb friction. For isotropic friction, specify a single coefficient of friction, MU, using either **TB** command input (recommended) or the **MP** command. For orthotropic friction, specify two coefficients of friction, MU1 and MU2, in two principal directions using **TB** command input. (See Contact Friction in the *Material Reference* for more information.)

For isotropic friction, local element coordinates based on the nodal connectivity are used to define principal directions. In the case of two crossing beams in contact (KEYPT(3) = 1), the first principal direction is defined by 1/2($\mathbf{t_1}$ + $\mathbf{t_2}$). The first vector, $\mathbf{t_1}$, points from the first contact node to the second contact node, and the second vector, $\mathbf{t_2}$, points from the first target node to the second target node. In the case of two parallel beams in contact (KEYOPT(3) = 0), the first principal direction points from the first contact node to the second contact node. In both cases, the second principal direction is defined by taking a cross product of the first principal direction and the contact normal.

For orthotropic friction, the principal directions are determined as follows. The global coordinate system is used by default, or you may define a local element coordinate system with the **ESYS** command. The first principal direction is defined by projecting the first direction of the chosen coordinate system onto the contact element. The second principal direction is defined by taking a cross product of the first principal direction and the contact normal. These directions also follow the rigid body rotation of the contact element to correctly model the directional dependence of friction. Be careful to choose the coordinate system (global or local) so that the first direction of that system is within 45° of the tangent to the contact surface.

If you want to set the coordinate directions for isotropic friction (to the global Cartesian system or another system via **ESYS**), you can define orthotropic friction and set MU1 = MU2.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To define a coefficient of friction for isotropic or orthotropic friction that is dependent on temperature, time, normal pressure, sliding distance, or sliding relative velocity, use the **TBFIELD** command along with **TB**,FRIC. See Contact Friction in the *Material Reference* for more information.

To implement a user-defined friction model, use the **TB**,FRIC command with *TBOPT* = USER to specify friction properties and write a `USERFRIC` subroutine to compute friction forces. See Writing Your Own Friction Law (USERFRIC) in the *Mechanical APDL Contact Technology Guide* for more information on how to use this feature. See also the *Guide to User-Programmable Features* for a detailed description of the `USERFRIC` subroutine.

In addition to the user-defined friction subroutine, the contact interaction subroutine `USERINTER` is available for user-defined interface interactions, including interactions in the normal and tangential directions. See Defining Your Own Contact Interaction (USERINTER) in the *Mechanical APDL Contact Technology Guide* for more information on how to use this feature. See also the *Guide to User-Programmable Features* for a detailed description of the `USERINTER` subroutine.

To model proper momentum transfer and energy balance between contact and target surfaces, impact constraints should be used in transient dynamic analysis. See the description of KEYOPT(7) below and the contact element discussion in the *Mechanical APDL Theory Reference* for details.

To model separation of bonded contact with KEYOPT(12) = 2, 3, 4, 5, or 6, use the **TB** command with the CZM label. See Debonding in the *Contact Technology Guide* for more information.

In addition to controlling the type of beam contact, KEYOPT(3) allows you to choose between a contact force-based model (KEYOPT(3) = 0 or 1; default = 0) and a contact traction-based model (KEYOPT(3) = 2 or 3). The units for certain real constants (FKN, FKT, TNOP) and postprocessing items (PRES, TAUR, TAUS, SFRIC, and so on) vary by a factor of AREA, depending on which model is specified. (For details, see the real constant table and output definitions table.) For more information on using KEYOPT(3) with CONTA176, see KEYOPT(3) in the *Contact Technology Guide*.

See the *Contact Technology Guide* for a detailed discussion on contact and using the contact elements. 3-D Beam-to-Beam Contact discusses CONTA176 specifically, including the use of real constants and KEYOPTs.

The following table summarizes the element input. Element Input (p. 43) gives a general description of element input.

CONTA176 Input Summary

Nodes

I, J, (K)

Degrees of Freedom

UX, UY, UZ

Real Constants

R1, R2, FKN, FTOLN, ICONT, PINB,
PMAX, PMIN, TAUMAX, CNOF, FKOP, FKT,
COHE, (Blank), (Blank), (Blank), (Blank), (Blank),
(Blank), (Blank), FACT, DC, SLTO, TNOP,
TOLS, (Blank), (Blank), (Blank), COR, STRM
FDMN, FDMT, , , TBND

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See Table 176.1: CONTA176 Real Constants (p. 748) for descriptions of the real constants.

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: MU

Special Features

Birth and death
Debonding
Isotropic friction
Large deflection
Linear perturbation
Nonlinearity
Orthotropic friction
User-defined contact interaction
User-defined friction

KEYOPTs

Presented below is a list of KEYOPTS available for this element. Included are links to sections in the *Contact Technology Guide* where more information is available on a particular topic.

KEYOPT(1)

Selects degrees of freedom. Currently, the default (UX, UY, UZ) is the only valid option:

0 --
UX, UY, UZ

KEYOPT(2)

Contact algorithm:

0 --
Augmented Lagrangian (default)

1 --
Penalty function

2 --
Multipoint constraint (MPC); see Multipoint Constraints and Assemblies in the *Contact Technology Guide* for more information

3 --
Lagrange multiplier on contact normal and penalty on tangent

4 --
Pure Lagrange multiplier on contact normal and tangent

KEYOPT(3)

Beam contact type:

0 --
Parallel beams or beam inside beam (contact forced-based model)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
Crossing beams (contact forced-based model)

2 --
Parallel beams or beam inside beam (contact traction-based model)

3 --
Crossing beams (contact traction-based model)

KEYOPT(4)
Type of surface-based constraint (see Surface-based Constraints for more information):

0 --
Rigid surface constraint

1 --
Force-distributed constraint

3 --
Coupling constraint

KEYOPT(5)
CNOF/ICONT Automated adjustment:

0 --
No automated adjustment

1 --
Close gap with auto CNOF

2 --
Reduce penetration with auto CNOF

3 --
Close gap/reduce penetration with auto CNOF

4 --
Auto ICONT

KEYOPT(6)
Contact stiffness variation (used to enhance stiffness updating when KEYOPT(10) > 0):

0 --
Use default range for stiffness updating

1 --
Make a nominal refinement to the allowable stiffness range

2 --
Make an aggressive refinement to the allowable stiffness range

KEYOPT(7)
Element level time incrementation control / impact constraints:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0 --
No control

1 --
Automatic bisection of increment

2 --
Change in contact predictions are made to maintain a reasonable time/load increment

3 --
Change in contact predictions made to achieve the minimum time/load increment whenever a change in contact status occurs

4 --
Use impact constraints for standard or rough contact (KEYOPT(12) = 0 or 1) in a transient dynamic analysis with automatic adjustment of time increment

---

Note

KEYOPT(7) = 2, 3, and 4 include an automatic adjustment of the time increment. This is activated only if the command **SOLCONTROL**,ON,ON was issued prior to the solution.

KEYOPT(8)
Asymmetric contact selection:

0 --
No action

2 --
ANSYS internally selects which asymmetric contact pair is used at the solution stage (used only when symmetry contact is defined).

KEYOPT(9)
Effect of initial penetration or gap:

0 --
Include both initial geometrical penetration or gap and offset

1 --
Exclude both initial geometrical penetration or gap and offset

2 --
Include both initial geometrical penetration or gap and offset, but with ramped effects

3 --
Include offset only (exclude initial geometrical penetration or gap)

4 --
Include offset only (exclude initial geometrical penetration or gap), but with ramped effects

5 --
Include offset only (exclude initial geometrical penetration or gap) regardless of the initial contact status (near-field or closed)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6 --
Include offset only (exclude initial geometrical penetration or gap), but with ramped effects regardless of the initial contact status (near-field or closed)

Note

The effects of KEYOPT(9) are dependent on settings for other KEYOPTs. The indicated initial gap effect is considered only if KEYOPT(12) = 4 or 5. See the discussion on using KEYOPT(9) in the *Contact Technology Guide* for more information.

KEYOPT(10)
Contact Stiffness Update:

0 --
Each load step if FKN is redefined during load step (pair based).

2 --
Each iteration based on current mean stress of underlying elements (pair based).

KEYOPT(12)
Behavior of contact surface:

0 --
Standard

1 --
Rough

2 --
No separation (sliding permitted)

3 --
Bonded

4 --
No separation (always)

5 --
Bonded (always)

6 --
Bonded (initial contact)

Note

When KEYOPT(12) = 5 or 6 is used with the MPC algorithm to model surface-based constraints, the KEYOPT(12) setting will have an impact on the local coordinate system of the contact element nodes. See Specifying a Local Coordinate System in the *Contact Technology Guide* for more information.

KEYOPT(15)
Effect of contact stabilization damping:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0 --
Damping is activated only in the first load step (default).

1 --
Deactivate automatic damping.

2 --
Damping is activated for all load steps.

3 --
Damping is activated at all times regardless of the contact status of previous substeps.

Note

Stabilization damping is only applied to contact pairs in near-field contact. When KEYOPT(15) = 0, 1, or 2, stabilization damping will not be applied in the current substep if any contact detection point had a closed status in the previous substep. However, when KEYOPT(15) = 3, stabilization damping is always applied as long as the current contact status is near-field. See Applying Contact Stabilization Damping in the *Contact Technology Guide* for more information.

Table 176.1: CONTA176 Real Constants

No.	Name	Description	For more information, see this section in the ***Contact Technology Guide*** . . .
1	R1	Target radius	Real Constants R1, R2
2	R2	Contact radius	Real Constants R1, R2
3	FKN[1]	Normal penalty stiffness factor [2] [3]	Determining Contact Stiffness and Penetration
4	FTOLN	Penetration tolerance factor	Determining Contact Stiffness and Penetration
5	ICONT	Initial contact closure	Adjusting Initial Contact Conditions
6	PINB	Pinball region	Determining Contact Status and the Pinball Region or Defining Influence Range (PINB)
7	PMAX	Upper limit of initial allowable penetration	Adjusting Initial Contact Conditions
8	PMIN	Lower limit of initial allowable penetration	Adjusting Initial Contact Conditions
9	TAUMAX	Maximum friction stress [2] [3]	Choosing a Friction Model

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

No.	Name	Description	For more information, see this section in the *Contact Technology Guide* . . .
10	CNOF	Contact surface offset [2]	Adjusting Initial Contact Conditions
11	FKOP	Contact opening stiffness [2] [3]	Selecting Surface Interaction Models
12	FKT[1]	Tangent penalty stiffness factor [2] [3]	Determining Contact Stiffness
13	COHE	Contact cohesion	Choosing a Friction Model
21	FACT	Static/dynamic ratio	Static and Dynamic Friction Coefficients
22	DC	Exponential decay coefficient	Static and Dynamic Friction Coefficients
23	SLTO	Allowable elastic slip	Using FKT and SLTO
24	TNOP	Maximum allowable tensile contact force/pressure [4]	Chattering Control Parameters
25	TOLS	Target edge extension factor	Real Constant TOLS
29	COR	Coefficient of restitution	Impact Between Rigid Bodies
30	STRM	Load step number for ramping penetration	Adjusting Initial Contact Conditions
31	FDMN	Normal stabilization damping factor [2] [3]	Applying Contact Stabilization Damping
32	FDMT	Tangential stabilization damping factor [2] [3]	Applying Contact Stabilization Damping
35	TBND	Critical bonding temperature [2] [3]	Using TBND

1. For the contact force-based model (KEYOPT(3) = 0 or 1), the units of real constants FKN and FKT have a factor of AREA with respect to those used in the surface-to-surface contact elements. See Performing a 3-D Beam-to-Beam Contact Analysis for more information.

2. This real constant can be defined as a function of certain primary variables.

3. This real constant can be defined by the user subroutine USERCNPROP.F.

4. For the contact force-based model (KEYOPT(3) = 0 or 1), TNOP is the allowable tensile contact force. For the contact traction-based model (KEYOPT(3) = 2 or 3), TNOP is the allowable tensile contact pressure.

CONTA176 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 176.2: CONTA176 Element Output Definitions (p. 750).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 176.2: CONTA176 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes I, J, K	Y	Y
XC, YC, ZC	Location where results are reported (same as nodal location)	Y	Y
TEMP	Temperature T(I)	Y	Y
VOLU	Length	Y	Y
NPI	Number of integration points	Y	-
ITRGET	Target surface number (assigned by ANSYS)	Y	-
ISOLID	Underlying beam element number	Y	-
CONT:STAT	Current contact statuses	1	1
OLDST	Old contact statuses	1	1
ISEG	Current contacting target element number	Y	Y
OLDSEG	Underlying old target number	Y	-
CONT:PENE	Current penetration (gap = 0; penetration = positive value)	Y	Y
CONT:GAP	Current gap (gap = negative value; penetration = 0)	Y	Y
NGAP	New or current gap at current converged substep (gap = negative value; penetration = positive value)	Y	-
OGAP	Old gap at previously converged substep (gap = negative value; penetration = positive value)	Y	-
IGAP	Initial gap at start of current substep (gap = negative value; penetration = positive value)	Y	Y
GGAP	Geometric gap at current converged substep (gap = negative value; penetration = positive value)	-	Y
CONT:PRES	Normal contact force/pressure	2	2
TAUR/TAUS[7]	Tangential contact forces/stresses	2	2
KN	Current normal contact stiffness (units: FORCE/LENGTH for contact force model, FORCE/LENGTH^3 for contact traction model)	5	5
KT	Current tangent contact stiffness (same units as KN)	5	5
MU[8]	Friction coefficient	Y	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
TASS/TASR[7]	Total (algebraic sum) sliding in S and R directions	3	3
AASS/AASR[7]	Total (absolute sum) sliding in S and R directions	3	3
TOLN	Penetration tolerance	Y	Y
CONT:SFRIC	Frictional force/stress, SQRT (TAUR**2+TAUS**2)	2	2
CONT:STOTAL	Total force/stress, SQRT (PRES**2+TAUR**2+TAUS**2)	2	2
CONT:SLIDE	Amplitude of total accumulated sliding, SQRT (TASS**2+TASR**2)	3	3
FDDIS	Frictional energy dissipation	6	6
ELSI	Total equivalent elastic slip distance	-	Y
PLSI	Total (accumulated) equivalent plastic slip due to frictional sliding	-	Y
GSLID	Amplitude of total accumulated sliding (including near-field)	-	9
VREL	Equivalent sliding velocity (slip rate)	-	Y
DBA	Penetration variation	Y	Y
PINB	Pinball Region	-	Y
CONT:CNOS	Total number of contact status changes during substep	Y	Y
TNOP	Maximum allowable tensile contact force/pressure	2	2
SLTO	Allowable elastic slip	Y	Y
CAREA	Contacting area	-	Y
DTSTART	Load step time during debonding	Y	Y
DPARAM	Debonding parameter	Y	Y
DENERI	Energy released due to separation in normal direction - mode I debonding	Y	Y
DENERII	Energy released due to separation in tangential direction - mode II debonding	Y	Y
CNFX[4]	Contact element force-X component	-	Y
CNFY	Contact element force-Y component	-	Y
CNFZ	Contact element force-Z component	-	Y
SDAMP	Stabilization damping coefficient	-	Y

1. The possible values of *STAT* and *OLDST* are:

   0 = Open and not near contact
   1 = Open but near contact
   2 = Closed and sliding
   3 = Closed and sticking

2. For the force-based model (KEYOPT(3) = 0 or 1), the unit of this quantity is FORCE. For the traction-based model (KEYOPT(3) = 2 or 3), the unit is FORCE/AREA.

3. Only accumulates the sliding for sliding and closed contact (STAT = 2,3).

4. Contact element forces are defined in the global Cartesian system

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. For the force-based model (KEYOPT(3) = 0 or 1), the unit of stiffness is FORCE/LENGTH. For the traction-based model (KEYOPT(3) = 2 or 3), the unit is FORCE/LENGTH^3.
6. FDDIS = (contact friction stress)*(sliding distance of substep)/(time increment of substep)
7. For the case of orthotropic friction in contact between beams, components are defined in the global Cartesian system.
8. For orthotropic friction, an equivalent coefficient of friction is output.
9. Accumulated sliding distance for near-field, sliding, and closed contact (STAT = 1,2,3).

Note

Contact results (including all element results) are generally not reported for elements that have a status of "open and not near contact" (far-field).

The following table lists output available through the **ETABLE** command using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information.

Name
output quantity as defined in Table 176.2: CONTA176 Element Output Definitions (p. 750)

Item
predetermined item label for **ETABLE** command

E
sequence number for single-valued or constant element data

NMISC

I, J, K
sequence number for data at nodes I, J, K

Table 176.3: CONTA176 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	E	I	J	K
PRES	SMISC	13	1	2	3
TAUR	SMISC	-	5	6	7
TAUS	SMISC	-	9	10	11
FDDIS	SMISC	-	18	19	20
STAT[1]	NMISC	41	1	2	3
OLDST	NMISC	-	5	6	7
PENE[2]	NMISC	-	9	10	11
DBA	NMISC	-	13	14	15
TASR	NMISC	-	17	18	19

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quant-ity Name	ETABLE and ESOL Command Input				
	Item	E	I	J	K
TASS	NMISC	-	21	22	23
KN	NMISC	-	25	26	27
KT	NMISC	-	29	30	31
TOLN	NMISC	-	33	34	35
IGAP	NMISC	-	37	38	39
PINB	NMISC	42	-	-	-
CNFX	NMISC	43	-	-	-
CNFY	NMISC	44	-	-	-
CNFZ	NMISC	45	-	-	-
ISEG	NMISC	-	46	47	48
AASR	NMISC	-	50	51	52
AASS	NMISC	-	54	55	56
CAREA	NMISC	58	-	-	-
MU	NMISC	-	62	63	64
DTSTART	NMISC	-	66	67	68
DPARAM	NMISC	-	70	71	72
CNOS	NMISC	-	112	113	114
TNOP	NMISC	-	116	117	118
SLTO	NMISC	-	120	121	122
ELSI	NMISC	-	136	137	138
DENERI	NMISC	-	140	141	142
DENERII	NMISC	-	144	145	146
GGAP	NMISC	-	152	153	154
VREL	NMISC	-	156	157	158
SDAMP	NMISC	-	160	161	162
PLSI	NMISC	-	164	165	166
GSLID	NMISC	-	168	169	170

1. Element Status = highest value of status of integration points within the element

2. Penetration = positive value, gap = negative value

You can display or list contact results through several POST1 postprocessor commands. The contact specific items for the **PLNSOL**, **PLESOL**, **PRNSOL**, and **PRESOL** commands are listed below:

STAT	Contact status
PENE	Contact penetration
PRES	Contact pressure for the traction-based model. Contact normal force for the force-based model.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SFRIC	Contact friction stress for the traction-based model. Friction force for the force-based model.
STOT	Contact total stress (pressure plus friction) for the traction-based model. Total contact force for the force-based model.
SLIDE	Contact sliding distance
GAP	Contact gap distance
CNOS	Total number of contact status changes during substep

CONTA176 Assumptions and Restrictions

- The main restriction is the assumption of constant circular beam cross section. The contact radius is assumed to be the same for all elements in the contact pair.
- For KEYOPT(3) = 1 (crossing beams), contact between the beams is pointwise, and each contact element contacts no more than one target element.
- This element is nonlinear and requires a full Newton iterative solution, regardless of whether large or small deflections are specified. An exception to this is when MPC bonded contact is specified (KEYOPT(2) = 2 and KEYOPT(12) = 5 or 6).
- The normal contact stiffness factor (FKN) must not be so large as to cause numerical instability.
- FTOLN, PINB, and FKOP can be changed between load steps or during restart stages.
- You can use this element in nonlinear static or nonlinear full transient analyses.
- In addition, you can use it in modal analyses, eigenvalue buckling analyses, and harmonic analyses. For these analysis types, the program assumes that the initial status of the element (i.e., the status at the completion of the static prestress analysis, if any) does not change.

CONTA176 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The MU material property (input via **MP**,MU or **TB**,FRIC) is not allowed in the Nonlinear Thermal product.
- The birth and death special feature is not allowed in the Nonlinear Structural and Nonlinear Thermal products.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CONTA177

3-D Line-to-Surface Contact

MP ME ST PR PRN <> <> <> PP EME MFS
Product Restrictions

CONTA177 Element Description

CONTA177 is used to represent contact and sliding between 3-D surface segments (TARGE170) and a deformable line segment, defined by this element. The element is applicable to 3-D beam-to-solid and 3-D shell edge-to-solid structural contact analyses. This element is located on the surfaces of 3-D beam or pipe elements with or without midside nodes (such as BEAM188, BEAM189, PIPE288, PIPE289, and ELBOW290). It can also be located on the edge of 3-D shell elements with or without midside nodes, such as SHELL181 and SHELL281. Contact occurs when the element surface penetrates one of the target segment elements (TARGE170) on a specified target surface. Coulomb friction, shear stress friction, user-defined friction with the `USERFRIC` subroutine, and user-defined contact interaction with the `USERINTER` subroutine are allowed. This element also allows separation of bonded contact to simulate interface delamination. See CONTA177 in the *Mechanical APDL Theory Reference* for more details about this element. To model beam-to-beam contact, use the line-to-line contact element, CONTA176.

Figure 177.1: CONTA177 Geometry

CONTA177 Input Data

The geometry and node locations are shown in Figure 177.1: CONTA177 Geometry (p. 755). The element is defined by two nodes (if the underlying beam or shell element does not have a midside node) or three nodes (if the underlying beam or shell element has a midside node). The element x-axis is along the I-J line of the element. Correct node ordering of the contact element is critical for proper detection of contact. The nodes must be ordered in a sequence that defines a continuous line. See Generating Contact Elements in the *Contact Technology Guide* for more information on generating elements automatically using the **ESURF** command.

ANSYS looks for contact only between contact and target surfaces with the same real constant set. For either rigid-flexible or flexible-flexible contact, one of the deformable "surfaces" (beam or shell edge) must be represented by a contact surface. See Designating Contact and Target Surfaces in the *Contact*

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Technology Guide for more information. If more than one target surface will make contact with the same boundary of line elements, you must define several contact elements that share the same geometry but relate to separate targets (targets which have different real constant numbers), or you must combine the two target surfaces into one (targets that share the same real constant numbers).

CONTA177 supports isotropic and orthotropic Coulomb friction. For isotropic friction, specify a single coefficient of friction, MU, using either **TB** command input (recommended) or the **MP** command. For orthotropic friction, specify two coefficients of friction, MU1 and MU2, in two principal directions using **TB** command input. (See Contact Friction in the *Mechanical APDL Material Reference* for more information.)

For isotropic friction, local element coordinates based on the nodal connectivity are used as principal directions. The first principal direction points from node I to node J. The second principal direction is defined by taking a cross product of the first principal direction and the contact normal.

For orthotropic friction, the principal directions are determined as follows. The global coordinate system is used by default, or you may define a local element coordinate system with the **ESYS** command. The first principal direction is defined by projecting the first direction of the chosen coordinate system onto the contact element. The second principal direction is defined by taking a cross product of the first principal direction and the contact normal. These directions also follow the rigid body rotation of the contact element to correctly model the directional dependence of friction. Be careful to choose the coordinate system (global or local) so that the first direction of that system is within 45° of the tangent to the contact surface.

If you want to set the coordinate directions for isotropic friction (to the global Cartesian system or another system via **ESYS**), you can define orthotropic friction and set MU1 = MU2.

To define a coefficient of friction for isotropic or orthotropic friction that is dependent on temperature, time, normal pressure, sliding distance, or sliding relative velocity, use the **TBFIELD** command along with **TB**,FRIC. See Contact Friction in the *Mechanical APDL Material Reference* for more information.

To implement a user-defined friction model, use the **TB**,FRIC command with *TBOPT* = USER to specify friction properties and write a `USERFRIC` subroutine to compute friction forces. See Writing Your Own Friction Law (USERFRIC) in the *Mechanical APDL Contact Technology Guide* for more information on how to use this feature. See also the *Guide to User-Programmable Features* for a detailed description of the `USERFRIC` subroutine.

In addition to the user-defined friction subroutine, the contact interaction subroutine `USERINTER` is available for user-defined interface interactions, including interactions in the normal and tangential directions. See Defining Your Own Contact Interaction (USERINTER) in the *Mechanical APDL Contact Technology Guide* for more information on how to use this feature. See also the *Guide to User-Programmable Features* for a detailed description of the `USERINTER` subroutine.

To model proper momentum transfer and energy balance between contact and target surfaces, impact constraints should be used in transient dynamic analysis. See the description of KEYOPT(7) below and the contact element discussion in the *Mechanical APDL Theory Reference* for details.

To model separation of bonded contact with KEYOPT(12) = 2, 3, 4, 5, or 6, use the **TB** command with the CZM label. See Debonding in the *Contact Technology Guide* for more information.

KEYOPT(3) allows you to choose between a contact force-based model (KEYOPT(3) = 0, default) and a contact traction-based model (KEYOPT(3) = 1). The units for certain real constants (FKN, FKT, TNOP) and postprocessing items (PRES, TAUR, TAUS, SFRIC, and so on) vary by a factor of AREA, depending on which model is specified. (For details, see the real constant table and output definitions table.) For more information on using KEYOPT(3) with CONTA177, see KEYOPT(3) in the *Contact Technology Guide*.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See the *Contact Technology Guide* for a detailed discussion on contact and using the contact elements. Line-to-Surface Contact discusses CONTA177 specifically, including the use of real constants and KEYOPTs.

The following table summarizes the element input. Element Input (p. 43) gives a general description of element input.

CONTA177 Input Summary

Nodes

I, J, (K)

Degrees of Freedom

UX, UY, UZ

Real Constants

R1, R2, FKN, FTOLN, ICONT, PINB,
PMAX, PMIN, TAUMAX, CNOF, FKOP, FKT,
COHE, (Blank), (Blank), (Blank), (Blank), (Blank),
(Blank), (Blank), FACT, DC, SLTO, TNOP,
TOLS, (Blank), (Blank), (Blank), COR, STRM
FDMN, FDMT, , , TBND
See Table 177.1: CONTA177 Real Constants (p. 761) for descriptions of the real constants.

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: MU

Special Features

Birth and death
Debonding
Isotropic friction
Large deflection
Linear perturbation
Nonlinearity
Orthotropic friction
User-defined contact interaction
User-defined friction

KEYOPTs

Presented below is a list of KEYOPTS available for this element. Included are links to sections in the *Contact Technology Guide* where more information is available on a particular topic.

KEYOPT(1)

Selects degrees of freedom. Currently, the default (UX, UY, UZ) is the only valid option:

0 --

UX, UY, UZ

KEYOPT(2)

Contact algorithm:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0 --
Augmented Lagrangian (default)

1 --
Penalty function

2 --
Multipoint constraint (MPC); see Multipoint Constraints and Assemblies in the *Contact Technology Guide* for more information

3 --
Lagrange multiplier on contact normal and penalty on tangent

4 --
Pure Lagrange multiplier on contact normal and tangent

KEYOPT(3)
Contact model:

0 --
Contact force-based model (default)

1 --
Contact traction-based model

KEYOPT(4)
Type of surface-based constraint (see Surface-based Constraints for more information):

0 --
Rigid surface constraint

1 --
Force-distributed constraint

3 --
Coupling constraint

KEYOPT(5)
CNOF/ICONT Automated adjustment:

0 --
No automated adjustment

1 --
Close gap with auto CNOF

2 --
Reduce penetration with auto CNOF

3 --
Close gap/reduce penetration with auto CNOF

4 --
Auto ICONT

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(6)

Contact stiffness variation (used to enhance stiffness updating when KEYOPT(10) > 0):

0 --
Use default range for stiffness updating

1 --
Make a nominal refinement to the allowable stiffness range

2 --
Make an aggressive refinement to the allowable stiffness range

KEYOPT(7)

Element level time incrementation control / impact constraints:

0 --
No control

1 --
Automatic bisection of increment

2 --
Change in contact predictions are made to maintain a reasonable time/load increment

3 --
Change in contact predictions made to achieve the minimum time/load increment whenever a change in contact status occurs

4 --
Use impact constraints for standard or rough contact (KEYOPT(12) = 0 or 1) in a transient dynamic analysis with automatic adjustment of time increment

---

Note

KEYOPT(7) = 2, 3, and 4 include an automatic adjustment of the time increment. This is activated only if the command **SOLCONTROL**,ON,ON was issued prior to the solution.

KEYOPT(9)

Effect of initial penetration or gap:

0 --
Include both initial geometrical penetration or gap and offset

1 --
Exclude both initial geometrical penetration or gap and offset

2 --
Include both initial geometrical penetration or gap and offset, but with ramped effects

3 --
Include offset only (exclude initial geometrical penetration or gap)

4 --
Include offset only (exclude initial geometrical penetration or gap), but with ramped effects

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5 --
Include offset only (exclude initial geometrical penetration or gap) regardless of the initial contact status (near-field or closed)

6 --
Include offset only (exclude initial geometrical penetration or gap), but with ramped effects regardless of the initial contact status (near-field or closed)

Note

The effects of KEYOPT(9) are dependent on settings for other KEYOPTs. The indicated initial gap effect is considered only if KEYOPT(12) = 4 or 5. See the discussion on using KEYOPT(9) in the *Contact Technology Guide* for more information.

KEYOPT(10)
Contact Stiffness Update:

0 --
Each load step if FKN is redefined during load step (pair based).

2 --
Each iteration based on current mean stress of underlying elements (pair based).

KEYOPT(11)
Shell thickness effect (target side only):

0 --
Exclude

1 --
Include

KEYOPT(12)
Behavior of contact surface:

0 --
Standard

1 --
Rough

2 --
No separation (sliding permitted)

3 --
Bonded

4 --
No separation (always)

5 --
Bonded (always)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6 --
Bonded (initial contact)

Note

When KEYOPT(12) = 5 or 6 is used with the MPC algorithm to model surface-based constraints, the KEYOPT(12) setting will have an impact on the local coordinate system of the contact element nodes. See Specifying a Local Coordinate System in the *Contact Technology Guide* for more information.

KEYOPT(15)
Effect of contact stabilization damping:

0 --
Damping is activated only in the first load step (default).

1 --
Deactivate automatic damping.

2 --
Damping is activated for all load steps.

3 --
Damping is activated at all times regardless of the contact status of previous substeps.

Note

Stabilization damping is only applied to contact pairs in near-field contact. When KEYOPT(15) = 0, 1, or 2, stabilization damping will not be applied in the current substep if any contact detection point had a closed status in the previous substep. However, when KEYOPT(15) = 3, stabilization damping is always applied as long as the current contact status is near-field. See Applying Contact Stabilization Damping in the *Contact Technology Guide* for more information.

Table 177.1: CONTA177 Real Constants

No.	Name	Description	For more information, see this section in the ***Contact Technology Guide*** . . .
1	R1	Target radius for cylinder, cone, or sphere	Defining the Target Surface
2	R2	Target radius at second node of cone	Defining the Target Surface
3	FKN[1]	Normal penalty stiffness factor [2] [3]	Determining Contact Stiffness and Penetration
4	FTOLN	Penetration tolerance factor	Determining Contact Stiffness and Penetration

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

No.	Name	Description	For more information, see this section in the ***Contact Technology Guide*** . . .
5	ICONT	Initial contact closure	Adjusting Initial Contact Conditions
6	PINB	Pinball region	Determining Contact Status and the Pinball Region or Defining Influence Range (PINB)
7	PMAX	Upper limit of initial allowable penetration	Adjusting Initial Contact Conditions
8	PMIN	Lower limit of initial allowable penetration	Adjusting Initial Contact Conditions
9	TAUMAX	Maximum friction stress [2] [3]	Choosing a Friction Model
10	CNOF	Contact surface offset [2]	Accounting for Thickness Effect (CNOF and KEYOPT(11))
11	FKOP	Contact opening stiffness [2] [3]	Selecting Surface Interaction Models
12	FKT[1]	Tangent penalty stiffness factor [2] [3]	Determining Contact Stiffness
13	COHE	Contact cohesion	Choosing a Friction Model
21	FACT	Static/dynamic ratio	Static and Dynamic Friction Coefficients
22	DC	Exponential decay coefficient	Static and Dynamic Friction Coefficients
23	SLTO	Allowable elastic slip	Using FKT and SLTO
24	TNOP	Maximum allowable tensile contact force/pressure [4]	Chattering Control Parameters
25	TOLS	Target edge extension factor	Real Constant TOLS
29	COR	Coefficient of restitution	Impact Between Rigid Bodies
30	STRM	Load step number for ramping penetration	Adjusting Initial Contact Conditions
31	FDMN	Normal stabilization damping factor [2] [3]	Applying Contact Stabilization Damping
32	FDMT	Tangential stabilization damping factor [2] [3]	Applying Contact Stabilization Damping
35	TBND	Critical bonding temperature [2] [3]	Using TBND

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. For the contact force-based model (KEYOPT(3) = 0), the units of real constants FKN and FKT have a factor of AREA with respect to those used in the surface-to-surface contact elements. See Performing a 3-D Line-to-Surface Contact Analysis for more information.

2. This real constant can be defined as a function of certain primary variables.

3. This real constant can be defined by the user subroutine USERCNPROP.F.

4. For the contact force-based model (KEYOPT(3) = 0), TNOP is the allowable tensile contact force. For the contact traction-based model (KEYOPT(3) = 1), TNOP is the allowable tensile contact pressure.

CONTA177 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 177.2: CONTA177 Element Output Definitions (p. 763).

A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 177.2: CONTA177 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes I, J, K	Y	Y
XC, YC, ZC	Location where results are reported (same as nodal location)	Y	Y
TEMP	Temperature T(I)	Y	Y
VOLU	Length	Y	Y
NPI	Number of integration points	Y	-
ITRGET	Target surface number (assigned by ANSYS)	Y	-
ISOLID	Underlying beam or shell element number	Y	-
CONT:STAT	Current contact statuses	1	1
OLDST	Old contact statuses	1	1
ISEG	Current contacting target element number	Y	Y
OLDSEG	Underlying old target number	Y	-
CONT:PENE	Current penetration (gap = 0; penetration = positive value)	Y	Y
CONT:GAP	Current gap (gap = negative value; penetration = 0)	Y	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
NGAP	New or current gap at current converged substep (gap = negative value; penetration = positive value)	Y	-
OGAP	Old gap at previously converged substep (gap = negative value; penetration = positive value)	Y	-
IGAP	Initial gap at start of current substep (gap = negative value; penetration = positive value)	Y	Y
GGAP	Geometric gap at current converged substep (gap = negative value; penetration = positive value)	-	Y
CONT:PRES	Normal contact force/pressure	2	2
TAUR/TAUS[7]	Tangential contact forces/stresses	2	2
KN	Current normal contact stiffness (units: FORCE/LENGTH for contact force model, FORCE/LENGTH^3 for contact traction model)	5	5
KT	Current tangent contact stiffness (same units as KN)	5	5
MU[8]	Friction coefficient	Y	Y
TASS/TASR[7]	Total (algebraic sum) sliding in S and R directions	3	3
AASS/AASR[7]	Total (absolute sum) sliding in S and R directions	3	3
TOLN	Penetration tolerance	Y	Y
CONT:SFRIC	Frictional force/stress, SQRT (TAUR**2+TAUS**2)	2	2
CONT:STOTAL	Total force/stress, SQRT (PRES**2+TAUR**2+TAUS**2)	2	2
CONT:SLIDE	Amplitude of total accumulated sliding, SQRT (TASS**2+TASR**2)	3	3
FDDIS	Frictional energy dissipation	6	6
ELSI	Total equivalent elastic slip distance	-	Y
PLSI	Total (accumulated) equivalent plastic slip due to frictional sliding	-	Y
GSLID	Amplitude of total accumulated sliding (including near-field)	-	9
VREL	Equivalent sliding velocity (slip rate)	-	Y
DBA	Penetration variation	Y	Y
PINB	Pinball Region	-	Y
CONT:CNOS	Total number of contact status changes during substep	Y	Y
TNOP	Maximum allowable tensile contact force/pressure	2	2
SLTO	Allowable elastic slip	Y	Y
CAREA	Contacting area	-	Y
DTSTART	Load step time during debonding	Y	Y
DPARAM	Debonding parameter	Y	Y
DENERI	Energy released due to separation in normal direction - mode I debonding	Y	Y
DENERII	Energy released due to separation in tangential direction - mode II debonding	Y	Y
CNFX[4]	Contact element force-X component	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
CNFY	Contact element force-Y component	-	Y
CNFZ	Contact element force-Z component	-	Y
SDAMP	Stabilization damping coefficient	-	Y

1. The possible values of *STAT* and *OLDST* are:

   0 = Open and not near contact
   1 = Open but near contact
   2 = Closed and sliding
   3 = Closed and sticking

2. For the force-based model (KEYOPT(3) = 0), the unit of this quantity is FORCE. For the traction-based model (KEYOPT(3) = 1), the unit is FORCE/AREA.

3. Only accumulates the sliding for sliding and closed contact (STAT = 2,3).

4. Contact element forces are defined in the global Cartesian system

5. For the force-based model (KEYOPT(3) = 0), the unit of stiffness is FORCE/LENGTH. For the traction-based model (KEYOPT(3) = 1), the unit is FORCE/LENGTH^3.

6. FDDIS = (contact friction stress)*(sliding distance of substep)/(time increment of substep)

7. For the case of orthotropic friction in contact between beams (or shell edges) and a 3-D surface, components are defined in the global Cartesian system.

8. For orthotropic friction, an equivalent coefficient of friction is output.

9. Accumulated sliding distance for near-field, sliding, and closed contact (STAT = 1,2,3).

Note

Contact results (including all element results) are generally not reported for elements that have a status of "open and not near contact" (far-field).

The following table lists output available through the **ETABLE** command using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information.

Name
output quantity as defined in Table 177.2: CONTA177 Element Output Definitions (p. 763)

Item
predetermined item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

I, J, K

sequence number for data at nodes I, J, K

Table 177.3: CONTA177 (3-D) Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	E	I	J	K
PRES	SMISC	13	1	2	3
TAUR	SMISC	-	5	6	7
TAUS	SMISC	-	9	10	11
FDDIS	SMISC	-	18	19	20
STAT[1]	NMISC	41	1	2	3
OLDST	NMISC	-	5	6	7
PENE[2]	NMISC	-	9	10	11
DBA	NMISC	-	13	14	15
TASR	NMISC	-	17	18	19
TASS	NMISC	-	21	22	23
KN	NMISC	-	25	26	27
KT	NMISC	-	29	30	31
TOLN	NMISC	-	33	34	35
IGAP	NMISC	-	37	38	39
PINB	NMISC	42	-	-	-
CNFX	NMISC	43	-	-	-
CNFY	NMISC	44	-	-	-
CNFZ	NMISC	45	-	-	-
ISEG	NMISC	-	46	47	48
AASR	NMISC	-	50	51	52
AASS	NMISC	-	54	55	56
CAREA	NMISC	58	-	-	-
MU	NMISC	-	62	63	64
DTSTART	NMISC	-	66	67	68
DPARAM	NMISC	-	70	71	72
CNOS	NMISC	-	112	113	114
TNOP	NMISC	-	116	117	118
SLTO	NMISC	-	120	121	122
ELSI	NMISC	-	136	137	138
DENERI	NMISC	-	140	141	142
DENERII	NMISC	-	144	145	146
GGAP	NMISC	-	152	153	154
VREL	NMISC	-	156	157	158

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quant-ity Name	ETABLE and ESOL Command Input				
	Item	E	I	J	K
SDAMP	NMISC	-	160	161	162
PLSI	NMISC	-	164	165	166
GSLID	NMISC	-	168	169	170

1. Element Status = highest value of status of integration points within the element

2. Penetration = positive value, gap = negative value

You can display or list contact results through several POST1 postprocessor commands. The contact specific items for the **PLNSOL**, **PLESOL**, **PRNSOL**, and **PRESOL** commands are listed below:

STAT	Contact status
PENE	Contact penetration
PRES	Contact pressure for the traction-based model. Contact normal force for the force-based model.
SFRIC	Contact friction stress for the traction-based model. Friction force for the force-based model.
STOT	Contact total stress (pressure plus friction) for the traction-based model. Total contact force for the force-based model.
SLIDE	Contact sliding distance
GAP	Contact gap distance
CNOS	Total number of contact status changes during substep

CONTA177 Assumptions and Restrictions

- The thickness effects of underlying beam elements on the contact side can be taken into account by specifying the contact surface offset CNOF.
- The thickness effects of underlying shell elements on the target side can be taken into account by setting KEYOPT(11) = 1.
- This element is nonlinear and requires a full Newton iterative solution, regardless of whether large or small deflections are specified. An exception to this is when MPC bonded contact is specified (KEYOPT(2) = 2 and KEYOPT(12) = 5 or 6).
- The normal contact stiffness factor (FKN) must not be so large as to cause numerical instability.
- FTOLN, PINB, and FKOP can be changed between load steps or during restart stages.
- You can use this element in nonlinear static or nonlinear full transient analyses.
- In addition, you can use it in modal analyses, eigenvalue buckling analyses, and harmonic analyses. For these analysis types, the program assumes that the initial status of the element (i.e., the status at the completion of the static prestress analysis, if any) does not change.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CONTA177 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The MU material property (input via **MP**,MU or **TB**,FRIC) is not allowed in the Nonlinear Thermal product.
- The birth and death special feature is not allowed in the Nonlinear Structural and Nonlinear Thermal products.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CONTA178

3-D Node-to-Node Contact

MP ME ST PR PRN <> <> <> PP EME MFS
Product Restrictions

CONTA178 Element Description

CONTA178 represents contact and sliding between any two nodes of any types of elements. The element has two nodes with three degrees of freedom at each node with translations in the X, Y, and Z directions. It can also be used in 2-D and axisymmetric models by constraining the UZ degree of freedom. The element is capable of supporting compression in the contact normal direction and Coulomb friction in the tangential direction. User-defined friction with the `USERFRIC` subroutine and user-defined interaction with the `USERINTER` subroutine are also allowed. The element may be initially preloaded in the normal direction or it may be given a gap specification. A longitudinal damper option can also be included. See CONTA178 in the *Mechanical APDL Theory Reference* for more details about this element. Other contact elements, such as COMBIN40, are also available.

Figure 178.1: CONTA178 Geometry

CONTA178 Input Data

The geometry, node locations, and the coordinate system for this element are shown in the CONTA178 figure above. The element is defined by two nodes, an initial gap or interference (GAP), an initial element status (START), and damping coefficients CV1 and CV2. The orientation of the interface is defined by the node locations (I and J) or by a user specified contact normal direction. The interface is assumed to be perpendicular to the I-J line or to the specified gap direction. The element coordinate system has its origin at node I and the x-axis is directed toward node J or in the user specified gap direction. The

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

interface is parallel to the element y-z plane. See Generating Contact Elements in the *Contact Technology Guide* for more information on generating elements automatically using the **EINTF** command.

To model proper momentum transfer and energy balance between contact and target surfaces, impact constraints should be used in transient dynamic analysis. See the description of KEYOPT(7) in the Input Summary and the contact element discussion in the *Mechanical APDL Theory Reference* for details.

Contact Algorithms

Four different contact algorithms can be selected:

- Pure Lagrange multiplier method (KEYOPT(2) = 4)
- Lagrange multiplier on contact normal and penalty on frictional (tangential) direction (KEYOPT(2) = 3)
- Augmented Lagrange method (KEYOPT(2) = 0)
- Pure Penalty method (KEYOPT(2) = 1)

The following sections outline these four algorithms.

Pure Lagrange Multiplier

The pure Lagrange multiplier method does not require contact stiffness FKN, FKS. Instead it requires chattering control parameters TOLN, FTOL, by which ANSYS assumes that the contact status remains unchanged. TOLN is the maximum allowable penetration and FTOL is the maximum allowable tensile contact force.

Note

A negative contact force occurs when the contact status is closed. A tensile contact force (positive) refers to a separation between the contact surfaces, but not necessarily an open contact status.

The behavior can be described as follows:

- If the contact status from the previous iteration is open and the current calculated penetration is smaller than TOLN, then contact remains open. Otherwise the contact status switches to closed and another iteration is processed.
- If the contact status from the previous iteration is closed and the current calculated contact force is positive, but smaller than FTOL, then contact remains closed. If the tensile contact force is larger than FTOL, then the contact status changes from closed to open and ANSYS continues to the next iteration.

ANSYS will provide reasonable defaults for TOLN and FTOL. Keep in mind the following when providing values for TOLN and FTOL:

- A positive value is a scaling factor applied to the default values.
- A negative value is used as an absolute value (which overrides the default).

The objective of TOLN and FTOL is to provide stability to models which exhibit contact chattering due to changing contact status. If the values you use for these tolerances are too small, the solution will

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

require more iterations. However, if the values are too big it will affect the accuracy of the solution, since a certain amount of penetration or tensile contact force are allowed.

Theoretically, the pure Lagrange multiplier method enforces zero penetration when contact is closed and "zero slip" when sticking contact occurs. However the pure Lagrange multiplier method adds additional degrees of freedom to the model and requires additional iterations to stabilize contact conditions. This will increase the computational cost and may even lead to solution divergence if many contact points are oscillating between sticking and sliding conditions during iterations.

Lagrange Multiplier on Normal and Penalty on Tangent Plane

An alternative algorithm is the Lagrange multiplier method applied on the contact normal and the penalty method (tangential contact stiffness) on the frictional plane. This method only allows a very small amount of slip for a sticking contact condition. It requires chattering control parameters TOLN, FTOL as well as the maximum allowable elastic slip parameter SLTOL. Again, ANSYS provides default tolerance values which work well in most cases. You can override the default value for SLTOL by defining a scaling factor (positive value) or an absolute value (negative value). Based on the tolerance, current normal contact force, and friction coefficient, the tangential contact stiffness FKS can be obtained automatically. In a few cases, you can override FKS by defining a scaling factor (positive input) or absolute value (negative input). Use care when specifying values for SLTOL and FKS. If the value for SLTOL is too large and the value for FKS too small, too much elastic slip can occur. If the value for SLTOL is too small or the value for FKS too large, the problem may not converge.

Augmented Lagrange Method

The third contact algorithm is the augmented Lagrange method, which is basically the penalty method with additional penetration control. This method requires contact normal stiffness FKN, maximum allowable penetration TOLN, and maximum allowable slip SLTOL. FKS can be derived based on the maximum allowable slip SLTOL and the current normal contact force. ANSYS provides a default normal contact stiffness FKN which is based on the Young's modulus E and the size of the underlying elements. If Young's modulus E is not found, $E = 1x10^9$ will be assumed.

You can override the default normal contact stiffness FKN by defining a scaling factor (positive input) or absolute value (negative input with unit force/length). If you specify a large value for TOLN, the augmented Lagrange method works as the penalty method. Use care when specifying values for FKN and TOLN. If the value for FKN is too small and the value for TOLN too large, too much penetration can occur. If the value for FKN is too large or the value for TOLN too small, the problem may not converge.

Penalty Method

The last algorithm is the pure penalty method. This method requires both contact normal and tangential stiffness values FKN, FKS. Real constants TOLN, FTOLN, and SLTOL are not used and penetration is no longer controlled in this method. Default FKN is provided as the one used in the augmented Lagrange method. The default FKS is given by MU x FKN. You can override the default values for FKN and FKS by inputting a scaling factor (positive input) or absolute value (negative input) for these real constants.

Contact Normal Definition

The contact normal direction is of primary importance in a contact analysis. By default [KEYOPT(5) = 0 and NX, NY, NZ = 0], ANSYS will calculate the contact normal direction based on the initial positions of the I and J nodes, such that a positive displacement (in the element coordinate system) of node J relative to node I opens the gap. However, you must specify the contact normal direction for any of the following conditions:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If nodes I and J have the same initial coordinates.
- If the model has an initial interference condition in which the underlying elements' geometry overlaps.
- If the initial open gap distance is very small.

In the above cases, the ordering of nodes I and J is critical. The correct contact normal usually points from node I toward node J unless contact is initially overlapped.

You can specify the contact normal by means of real constants NX, NY, NZ (direction cosines related to the global Cartesian system) or element KEYOPT(5). The following lists the various options for KEYOPT(5):

KEYOPT(5) = 0
The contact normal is either based on the real constant values of NX, NY, NZ or on node locations when NX, NY, NZ are not defined. For 2-D contact, NZ = 0.

KEYOPT(5) = 1 (2,3)
The contact normal points in a direction which averages the direction cosines of the X (Y, Z) axis of the nodal coordinates on both nodes I and J. The direction cosines on nodes I and J should be very close. This option may be supported by the **NORA** and **NORL** commands, which rotate the X axis of the nodal coordinate system to point to the surface normal of solid models.

KEYOPT(5) = 4 (5,6)
The contact normal points to X (Y, Z) of the element coordinate system issued by the **ESYS** command. If you use this option, make sure that the element coordinate system specified by **ESYS** is the Cartesian system. Otherwise, the global Cartesian system is assumed.

Note that the above discussion pertains to the unidirectional gap option (KEYOPT(1) = 0). For the definition of the contact normal for other gap options, see "Cylindrical Gap" (p. 774) and "Spherical Gap" (p. 775).

Contact Status

The initial gap defines the gap size (if positive) or the displacement interference (if negative). If KEYOPT(4) = 0, the default, the gap size can be automatically calculated from the GAP real constant and the node locations (projection of vector points from node I to J on the contact normal), that is, the gap size is determined from the additive effect of the geometric gap and the value of GAP.

If KEYOPT(4) = 1, the initial gap size is only based on real constant GAP (node locations are ignored).

By default KEYOPT(9) is set to 0, which means the initial gap size is applied in the first load step. To ramp the initial gap size with the first load step (to model initial interference problems, for example), set KEYOPT(9) = 1. Also, set **KBC**,0 and do not specify any external loads over the first load step.

The force deflection relationships for the contact element can be separated into the normal and tangential (sliding) directions. In the normal direction, when the normal force (FN) is negative, the contact status remains closed (STAT = 3 or 2). In the tangential direction, for FN < 0 and the absolute value of the tangential force (FS) less than $\mu|FN|$, contact "sticks" (STAT = 3). For FN < 0 and FS = $\mu|FN|$, sliding occurs (STAT = 2). As FN becomes positive, contact is broken (STAT = 1) and no force is transmitted (FN = 0, FS = 0).

The contact condition at the beginning of the first substep can be determined from the START parameter. The initial element status (START) is used to define the "previous" condition of the interface at the start of the first substep. This value overrides the condition implied by the interference specification and can

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

be useful in anticipating the final interface configuration and reducing the number of iterations required for convergence. However, specifying unrealistic START values can sometimes degrade the convergence behavior.

If START = 0.0 or blank, the initial status of the element is determined from either the GAP value or the KEYOPT(4) setting. If START = 3.0, contact is initially closed and not sliding ($\mu \neq 0$), or sliding (if $\mu = 0.0$). If START = 2.0, contact is initially closed and sliding. If START = 1.0, contact is initially open.

Friction

The only structural material property used is the interface coefficient of friction μ (MU). A zero value should be used for frictionless surfaces. Temperatures may be specified at the element nodes (via the **BF** command in a structural analysis or the **D** command in a coupled thermal-structural analysis). The coefficient of friction μ is evaluated at the average of the two node temperatures. The node I temperature T(I) defaults to TUNIF. The node J temperature defaults to T(I).

For analyses involving friction, using **NROPT**,UNSYM is useful (and, in fact, sometimes required if the coefficient of friction μ is > 0.2) for problems where the normal and tangential (sliding) motions are strongly coupled.

To define a coefficient of friction for isotropic friction that is dependent on temperature, time, normal pressure, sliding distance, or sliding relative velocity, use the **TBFIELD** command along with **TB**,FRIC. See Contact Friction in the *Material Reference* for more information.

Rigid Coulomb friction can be modeled with KEYOPT(10) = 7. In penalty-based methods, there is some elastic sliding present based on FKS. With the rigid Coulomb friction option, no sliding stiffness is used, so the element is always in a sliding state. See Rigid Coulomb Friction for details.

This element also supports user-defined friction. To implement a user-defined friction model, use the **TB**,FRIC command with *TBOPT* = USER to specify friction properties and write a `USERFRIC` subroutine to compute friction forces. See Writing Your Own Friction Law (USERFRIC) in the *Mechanical APDL Contact Technology Guide* for more information on how to use this feature. See also the *Guide to User-Programmable Features* for a detailed description of the `USERFRIC` subroutine.

In addition to the user-defined friction subroutine, the contact interaction subroutine `USERINTER` is available for user-defined interface interactions, including interactions in the normal and tangential directions as well coupled-field interactions. See Defining Your Own Contact Interaction (USERINTER) in the *Mechanical APDL Contact Technology Guide* for more information on how to use this feature. See also the *Guide to User-Programmable Features* for a detailed description of the `USERINTER` subroutine.

Cross-sectional Area

You may specify the contact area (defaults to 1.0) associated with this element by specifying real constant AREA. This area is used to compute the normal pressure and the tangential stress values. For axisymmetric problems, the area should be input on a full 360° basis.

Weak Spring

KEYOPT(3) can be used to specify a "weak spring" across an open or free sliding interface, which is useful for preventing rigid body motion that could occur in a static analysis. The weak spring stiffness is computed by multiplying the normal stiffness KN by a reduction factor if the real constant REDFACT is positive (which defaults to 1×10^{-6}). The weak spring stiffness can be overridden if REDFACT has a negative value. Set KEYOPT(3) = 1 to add weak spring stiffness only to the contact normal direction

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

when contact is open. Set KEYOPT(3) = 2 to add weak spring stiffness to the contact normal direction for open contact and tangent plane for frictionless or open contact.

The weak spring only contributes to global stiffness, which prevents a "singularity" condition from occurring during the solution phase if KEYOPT(3) = 1,2. By setting KEYOPT(3) = 3,4, the weak spring will contribute both to the global stiffness and the internal nodal force which holds two separated nodes.

Note

The weak spring option should *never* be used in conjunction with either the no-separation or bonded contact options defined by KEYOPT(10).

Contact Behavior

Use KEYOPT(10) to model the following different contact surface behaviors:

KEYOPT(10) = 0
Models standard unilateral contact; that is, normal pressure equals zero if separation occurs.

KEYOPT(10) = 1
Models rough frictional contact where there is no sliding. This case corresponds to an infinite friction coefficient and ignores the material property input MU.

KEYOPT(10) = 2
Models no separation contact, in which two gap nodes are tied (although sliding is permitted) for the remainder of the analysis once contact is established.

KEYOPT(10) = 3
Models bonded contact, in which two gap nodes are bonded in all directions (once contact is established) for the remainder of the analysis.

KEYOPT(10) = 4
Models no separation contact, in which two gap nodes are always tied (sliding is permitted) throughout the analysis.

KEYOPT(10) = 5
Models bonded contact, in which two gap nodes are bonded in all directions throughout the analysis.

KEYOPT(10) = 6
Models bonded contact, in which two gap nodes that are initially in a closed state will remain closed and two gap nodes that are initially in an open state will remain open throughout the analysis.

KEYOPT(10) = 7
Models rigid Coulomb frictional contact in which sliding always occurs regardless of the magnitude of the normal contact force and the friction coefficient. Sticking contact never occurs and FKS is not used in this case. This option is useful for displacement-controlled problems or for certain dynamic problems where sliding dominates.

Cylindrical Gap

The cylindrical gap option (KEYOPT(1) = 1) is useful where the final contact normal is not fixed during the analysis, such as in the interaction between concentric pipes. With this option, you define the real

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

constants NX, NY, NZ as the direction cosines of the cylindrical axis ($\vec{N}$) in the global Cartesian coordinate system. The contact normal direction lies in a cross section that is perpendicular to the cylindrical axis. The program measures the relative distance |XJ - XI| between the current position of node I and the current position of node J projected onto the cross section. NX, NY, NZ defaults to (0,0,1), which is the case for a 2-D circular gap. With the cylindrical gap option, KEYOPT(4) and KEYOPT(5) are ignored and node ordering can be arbitrary. Real constant GAP is no long referred as the initial gap size and a zero value is not allowed. The following explanation defines the model based on the sign of the GAP value.

Figure 178.2: CONTA178 - Cylindrical Gap

- A positive GAP value models contact when one smaller cylinder inserted into another parallel larger cylinder. GAP is equal to the difference between the radii of the cylinders (|RJ - RI|) and it represents the maximum allowable distance projected on the cross-section. The contact constraint condition can be written as : $|XJ - XI| \leq |GAP|$

- A negative GAP value models external contact between two parallel cylinders. GAP is equal to the sum of the radii of the cylinders (|RJ + RI|) and it represents the minimum allowable distance projected on the cross-section. The contact constraint condition can be written as: $|XJ - XI| \geq |GAP|$

Spherical Gap

The spherical gap option (KEYOPT(1) = 2) is useful when the final contact normal is not fixed during the analysis, such as in the interaction between two rigid spheres. With this spherical gap option, KEY-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OPT(4), KEYOPT(5), and real constants NX, NY, NZ are ignored. In addition, node ordering can be arbitrary. Real constant GAP is no longer referenced as the initial gap size, and a zero value is not allowed.

Figure 178.3: CONTA178 - Spherical Gap

The following explanation defines the model based on the sign of the GAP value:

- A positive GAP value models contact when one smaller sphere is inside another large sphere. GAP is equal to the difference between the radii of the spheres (|RJ - RI|) and it represents the maximum allowable distance. The contact constraint condition can be written as: $|XJ - XI| \leq |GAP|$
- A negative GAP value models external contact between two spheres. GAP is equal to the sum of the radii of the spheres (|RJ + RI|) and it represents the minimum allowable distance. The contact constraint condition can be written as: $|XJ - XI| \geq |GAP|$

Damper

The damping capability is only used for modal and transient analyses. By default, the damping capability is removed from the element. Damping is only active in the contact normal direction when contact is closed. The damping coefficient units are Force (Time/Length). For a 2-D axisymmetric analysis, the coefficient should be on a full 360° basis. The damping force is computed as $F_x = -C_v \frac{du_x}{dt}$, where C_v is the damping coefficient given by $C_v = C_{v1} + C_{v2}xV$. V is the velocity calculated in the previous substep. The second damping coefficient (C_{v2}) is available to produce a nonlinear damping effect.

Modeling Thermal Contact

You can use the node-to-node contact element, in combination with thermal-structural coupled field solid elements or thermal elements, to model heat transfer that occurs between two contact nodes. To activate both the structural and thermal DOFs, set KEYOPT(6) = 1. To activate the thermal DOF only, set KEYOPT(6) = 2.

The contact normal direction must be specified, even in a purely thermal analysis (KEYOPT(6) = 2).

In the discussions below, note that the contacting area (AREA) is input as a real constant; it is not automatically calculated by the program.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modeling Conduction

To take into account the conductive heat transfer between two contacting nodes, you need to specify the thermal contact conductance coefficient, which is real constant TCC.

The conduction function is defined as follows:

$$q(I) = -q(J) = TCC * AREA * (T(I) - T(J))$$

where:

q(I) = heat flow to node I (Heat/Time)
q(J) = heat flow to node J (Heat/Time)
AREA = cross sectional area ($Length^2$) (defaults to 1)
TCC = thermal contact conductance coefficient ($Heat/Length^2 * Time * Deg$)
T(I) = temperature at node I (Deg)
T(J) = temperature at node J (Deg)

The TCC value is input through a real constant, which can be made a function of temperature [(T(I) + T(J))/2], pressure, time, penetration (positive input with GAP as a table parameter), and location by using the %*tabname*% option. If contact occurs, a small value of TCC yields a measured amount of imperfect contact and a temperature discontinuity across the two contacting nodes. For large values of TCC, the resulting temperature discontinuity tends to vanish, and perfect thermal contact is approached. When not in contact, however, it is assumed that no heat is transferred across the interface. To model contact conduction between two surfaces where a small gap exists, use KEYOPT(10) = 4 or 5 to define either the "bonded contact" or "no-separation contact" option.

Modeling Convection

The standard convection function is defined as follows:

$$q(I) = -q(J) = -CONV * AREA * (T(I) - T(J))$$

where:

CONV = Heat convection coefficient ($Heat/Length^2 * Time * Deg$))

To model convective heat transfer between two opening nodes, you must specify the heat convection coefficient CONV using the **SFE** command (with *KVAL* = 1 and CONV as a table parameter). CONV can be a constant value (only uniform is allowed) or a function of temperature [T(I) + T(J))/2], time, or location as specified through tabular input. If CONV is not defined through the **SFE** command, real constant TCC will be used instead only if tabular input or a user subroutine is used. TCC can be made a function of temperature [(T(I) + T(J))/2], pressure, time, gap (negative value input with GAP as a table parameter), and location by using the %*tabname*% option.

Modeling Radiation

The standard radiation function is defined as follows:

$$q(I) = -q(J) = -RDVF * EMIS * SBCT * AREA* [(T(I) + TOFFST)^4 - (T(J)) + TOFFST)^4]$$

where:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TOFFST = the temperature offset from absolute zero to zero (defined through the **TOFFST** command)
EMIS = surface emissivity (input as a material property)
SBCT = The Stefan-Boltzmann constant (input as a real constant). There is no default for SBCT; if it is not defined, the radiation effect is excluded. (BTU/Hr * in^2 * R^4)
RDVF = The radiation view factor input as a real constant (defaults to 1). RDVF can be defined as a function of temperature, gap distance (negative value input with GAP as a table parameter), time, and location by using the %*tabname*% option.

Modeling Heat Generation Due to Friction

The standard frictional heat function is defined as follows:

q(I) = FHTG * FWGT * FDDIS

q(J) = FHTG * (1.0 - FWGT) * FDDIS

where:

FHTG = the fraction of frictionally dissipated energy converted into heat (defaults to 1.0). For an input of true zero, you must enter a very small value (for example, 1E-8). If you enter 0, the program interprets this input as the default value.
FWGT = the weight factor for the distribution of heat between node I and node J (defaults to 0.5)
FDDIS = frictional energy dissipation (Heat/Time)

In order to model heat generation due to frictionally dissipated energy, you would typically perform a coupled transient thermal-structural analysis. If inertia effects are negligible, you can turn off transient effects on structural DOFs by using the command **TIMINT**,STRUC,OFF. However, you should generally include transient effects on the thermal DOF.

In a steady-state coupled thermal-structural analysis, the friction-induced heat is still accounted for. The program computes the frictional energy dissipation as follows:

FDDIS = (total accumulated frictional energy)/(total time of the current load step)

In this case, the time of the current load step cannot be described as arbitrary. If you wish to neglect the friction-induced heat generation in the steady-state coupled thermal-structural analysis, use FHTG to specify a very small fraction of the frictionally dissipated energy (for example, FHTG = 1E-8).

Modeling External Heat Flux

You can apply an external flux through the **SFE** command. Only a uniform flux can be applied. The node J heat flux, HFLUX(J), defaults to the node I heat flux, HFLUX(I). The external heat flux is active only for opening contact.

The heat flow is defined as follows:

q(I) = q(J) = FLUX * AREA

where:

FLUX = applied heat flux, HFLUX(I)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modeling Electric Contact

You can use the node-to-node contact element, in combination with thermal-electric elements and solid coupled field elements, to model electric current conduction between the two contacting nodes. You can also use the node-to-node contact element, in combination with piezoelectric and electrostatic elements, to model electric charge between the two contacting nodes.

KEYOPT(6) provides degree of freedom options for modeling electric contact. For combined structural/thermal/electric contact, set KEYOPT(6) = 3 to activate the structural, thermal, and electric current DOFs. For pure thermal/electric contact, set KEYOPT(6) = 4 to activate the thermal and electric current DOFs. For piezoelectric contact, set KEYOPT(6) = 5 to activate the structural and piezoelectric DOFs. For electrostatic contact, set KEYOPT(6) = 6 to activate the electrostatic DOF.

The contact normal direction must be specified, even in a purely electrical analysis (KEYOPT(6) = 6).

In the discussions below, note that the contacting area (AREA) is input as a real constant; it is not automatically calculated by the program.

Modeling Electric Conduction

To take into account the surface interaction for electric contact, you need to specify the electric contact conductance if you are using the electric current degree of freedom, or the electric contact capacitance if you are using the piezoelectric or electrostatic degrees of freedom. For either case, this parameter is specified through a real constant ECC.

The interaction between the two contacting surfaces is defined by:

J = ECC * AREA * (V(I) - V(J))

where:

J = current density for the electric potential (VOLT) degree of freedom (KEYOPT(1) = 3 or 4), or the electric charge density (KEYOPT(1) = 5, or 6)
V(I) and V(J) = voltages at node I and node J
ECC = electric contact conductance for the electric potential (VOLT) degree of freedom (KEYOPT(1) = 3 or 4), or the electric contact capacitance per unit area for (KEYOPT(1) = 5, or 6)

The ECC value is input through a real constant, which can be a function of temperature [(T(I) + T(J))/2], pressure (PRESSURE as a table parameter), penetration (positive value input with GAP as a table parameter), gap (negative value input with GAP as a table parameter) , and time by using the %*tabname*% option. For the current conduction option, the electric contact conductance ECC has units of electric conductivity/length. For the piezoelectric and electrostatic options, the electric contact capacitance ECC has units of capacitance per unit area. If the “bonded contact” or “no-separation contact” option is set, contact interaction can occur between two nodes separated by a narrow gap.

Modeling Heat Dissipation Due to Electric Current

For electric current field analyses (KEYOPT(6) = 3 or 4), the heat dissipation at node I and node J due to electric current is given by:

q(I) = FHEG * FWGT * J * (V(I) - V(J))

q(J) = FHEG * (1.0 - FWGT) * J * (V(I) - V(J))

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where:

FHEG = the fraction of electrically dissipated energy converted into heat (Joule heating). This value defaults to 1 and can be input as a real constant. For an input of true zero, you must enter a very small value (for example, 1E-8). If you enter 0, the program interprets this input as the default value.
J = the current density
FWGT = the weight factor for the contact heat dissipation between node I and node J (input as a real constant). FWGT is the same real constant used for frictional heat generation. By default, FWGT = 0.5. For an input of true zero, you must enter a very small value (for example, 1E-8). If you enter 0, the program interprets this input as the default value.

Defining Real Constants in Tabular Format

For certain real constants, you can define the real constants as a function of primary variables by using tabular input. Table 178.2: Real Constants and Corresponding Primary Variables for CONTA178 (p. 785) lists real constants that allow tabular input and their associated primary variables.

Use the ***DIM** command to dimension the table and identify the variables. Keep in mind that you should follow the positive and negative convention for the real constant values. The possible primary variables used in tabular input are:

Time (TIME)
X location (X) in local/global coordinates
Y location (Y) in local/global coordinates
Z location (Z) in local/global coordinates
Temperature (TEMP degree of freedom)
Contact normal force (PRESSURE); specify positive index values for compression, negative index values for tension
Geometric contact gap/penetration (GAP); specify positive index values for penetration, negative index values for an open gap

Note that the primary variables X, Y, and Z represent the coordinates of the contact detection points. Coordinate system applicability is determined by the ***DIM** command.

When defining the tables, the primary variables must be in ascending order in the table indices (as in any table array).

When defining real constants via the **R**, **RMORE**, or **RMODIF** command, enclose the table name in % signs (e.g., %*tabname*%). For example, given a table named CNREAL1 that contains normal contact stiffness (FKN) values, you would issue a command similar to the following:

```
RMODIF,NSET,1,%CNREAL1%      ! NSET is the real constant set ID associated with the contact pair
```

For more information and examples of using table inputs, see TABLE Type Array Parameters in the *ANSYS Parametric Design Language Guide*; Applying Loads Using TABLE Type Array Parameters in the *Basic Analysis Guide*; and the ***DIM** command in the *Command Reference*.

Defining Real Constants via a User Subroutine

You can write a USERCNPROP subroutine to define your own real constants. The real constants that can be defined via this user subroutine are noted in Table 178.1: CONTA178 Real Constants (p. 785). For additional details, see Defining Your Own Real Constant (USERCNPROP) in the *Contact Technology Guide*, and the description of USERCNPROP in the *Guide to User-Programmable Features*.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Monitoring Contact Status

By default, the program will not print out contact status and contact stiffness for each individual element. Use KEYOPT(12) = 1 to print out such information, which may help in solving problems that are difficult to converge.

See the *Contact Technology Guide* for a detailed discussion on contact and using the contact elements. Node-to-Node Contact discusses CONTA178 specifically, including the use of real constants and KEYOPTs.

A summary of the element input is given in "CONTA178 Input Summary" (p. 781). A general description of element input is given in Element Input (p. 43).

CONTA178 Input Summary

Nodes

I, J

Degrees of Freedom

UX, UY, UZ (if KEYOPT(6) = 0)
UX, UY, UZ, TEMP (if KEYOPT(6) = 1)
TEMP (if KEYOPT(6) = 2)
UX, UY, UZ, TEMP, VOLT (if KEYOPT(6) = 3)
TEMP, VOLT (if KEYOPT(6) = 4)
UX, UY, UZ, VOLT (if KEYOPT(6) = 5)
VOLT (if KEYOPT(6) = 6)

Real Constants

FKN, GAP, START, FKS, REDFACT, NX,
NY, NZ, TOLN, FTOL, SLTOL, CV1,
CV2, COR, AREA, TCC, FHTG, SBCT
RDVF, FWGT, ECC, FHEG
See Table 178.1: CONTA178 Real Constants (p. 785) for a description of the real constants

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: MU, EMIS

Surface Loads

Convection, Face 1 (input with **SFE** command)
Heat Flux, Face 1 (I-J) (input with **SFE** command)

Body Loads

Temperatures - T(I), T(J)

Special Features

Isotropic friction
Linear perturbation
Nonlinear gap type
User-defined contact interaction
User-defined friction

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(1)

Gap type:

0 --

Unidirectional gap

1 --

Cylindrical gap

2 --

Spherical gap

KEYOPT(2)

Contact algorithm:

0 --

Augmented Lagrange method (default)

1 --

Pure Penalty method

3 --

Lagrange multiplier on contact normal and penalty on tangent (uses U/P formulation for normal contact, non-U/P formulation for tangential contact)

4 --

Lagrange multiplier method

KEYOPT(3)

Weak Spring:

0 --

Not used

1 --

Acts across an open contact (only contributes to stiffness)

2 --

Acts across an open contact or free sliding plane (only contributes to stiffness)

3 --

Acts across an open contact (contributes to stiffness and internal force)

4 --

Acts across an open contact or free sliding plane (contributes to stiffness and force)

KEYOPT(4)

Gap size:

0 --

Gap size based on real constant GAP + initial node locations

1 --

Gap size based on real constant GAP (ignore node locations)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(5)

Basis for contact normal:

0 --

Node locations or real constants NX, NY, NZ

1 --

X - component of nodal coordinate system (averaging on two contact nodes)

2 --

Y - component of nodal coordinate system (averaging on two contact nodes)

3 --

Z - component of nodal coordinate system (averaging on two contact nodes)

4 --

X - component of defined element coordinate system (ESYS)

5 --

Y - component of defined element coordinate system (ESYS)

6 --

Z - component of defined element coordinate system (ESYS)

KEYOPT(6)

Selects degrees of freedom:

0 --

UX, UY, UZ

1 --

UX, UY, UZ, TEMP

2 --

TEMP

3 --

UX, UY, UZ, TEMP, VOLT

4 --

TEMP, VOLT

5 --

UX, UY, UZ, VOLT

6 --

VOLT

KEYOPT(7)

Element level time incrementation control / impact control:

0 --

No control.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
Change in contact predictions are made to maintain a reasonable time/load increment. This option is activated only if the command **SOLCONTROL**,ON,ON was issued prior to the solution.

2 --
Change in contact predictions are made to achieve the minimum time/load increment whenever a change in contact status occurs. This option is activated only if the command **SOLCONTROL**,ON,ON was issued prior to the solution.

4 --
Use impact constraints for standard or rough contact (KEYOPT(12) = 0 or 1) in a transient dynamic analysis with automatic adjustment of the time increment. Automatic adjustment of the time increment is performed only if the command **SOLCONTROL**,ON,ON was issued prior to the solution.

KEYOPT(9)
Initial gap step size application:

0 --
Initial gap size is step applied

1 --
Initial gap size is ramped in the first load step

KEYOPT(10)
Behavior of contact surface:

0 --
Standard

1 --
Rough

2 --
No separation (sliding permitted)

3 --
Bonded

4 --
No separation (always)

5 --
Bonded (always)

6 --
Bonded (initial)

7 --
Rigid Coulomb friction (resisting force only if MU > 0.0)

KEYOPT(12)
Contact Status:

0 --
Does not print contact status

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
Monitor and print contact status, contact stiffness

Table 178.1: CONTA178 Real Constants

No.	Name	Description
1	FKN	Normal stiffness [1] [2]
2	GAP	Initial gap size [1]
3	START	Initial contact status
4	FKS	Sticking stiffness [1] [2]
5	REDFACT	KN/KS reduction factor
6	NX	Defined gap normal - X component
7	NY	Defined gap normal - Y component
8	NZ	Defined gap normal - Z component
9	TOLN	Penetration tolerance
10	FTOL	Maximum tensile contact force
11	SLTOL	Maximum elastic slip
12	CV1	Damping coefficient
13	CV2	Nonlinear damping coefficient
14	COR	Coefficient of restitution
15	AREA	Cross-sectional area (defaults to 1.0)
16	TCC	Thermal contact conductance [1] [2]
17	FHTG	Fraction of frictionally dissipation energy converted into heat
18	SBCT	Stefan-Boltzmann constant. There is no default for SBCT. If this real constant is not defined, the radiation effect is excluded. The Stefan-Boltzmann constant should correspond to the units of the FE model to be solved (e.g., 0.1190E-10 Btu/hr-in^2-R^4; 5.67E-8 W/m^2-K^4)
19	RDVF	Radiation view factor [1] [2]
20	FWGT	Heat distribution weighing factor
21	ECC	Electric contact conductance [1] [2]
22	FHEG	The fraction of Joule dissipation energy converted into heat

1. This real constant can be defined as a function of certain primary variables.
2. This real constant can be defined by the user subroutine USERCNPROP.F.

Table 178.2: Real Constants and Corresponding Primary Variables for CONTA178

Real Constants		**TIME**	**Location (X,Y,Z)**	**TEMP**	**PRESSURE [1]**	**GAP [2]**	**%_CNPROP% [4]**
Number	**Name**						
1	FKN	x	x	x	x [3]	x	x
2	GAP	x	x	x	x	x	x
4	FKS	x	x	x	x	x	x

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Real Constants		TIME	Location (X,Y,Z)	TEMP	PRESSURE [1]	GAP [2]	%_CNPROP% [4]
16	TCC	x	x	x	x	x	x
19	RDVF	x	x	x		x	x
21	ECC	x	x	x	x	x	x

1. Contact normal force via the primary variable PRESSURE. For PRESSURE index values, specify positive values for compression, negative value for tension.
2. Geometric contact gap/penetration via the primary variable GAP. For GAP index values, specify positive values for penetration, negative values for an open gap.
3. Contact normal force at the end of the previous iteration, if it exists.
4. Real constant can be defined via the user subroutine USERCNPROP.

CONTA178 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution.
- Additional element output as shown in Element Output Definitions.

The value of USEP is determined from the normal displacement (UN), in the element x-direction, between the contact nodes at the end of a substep. This value is used in determining the normal force, FN. The values represented by UT(Y, Z) are the total translational displacements in the element y and z directions. The maximum value printed for the sliding force, FS, is μ|FN|. Sliding may occur in both the element y and z directions. STAT describes the status of the element at the end of a substep.

- If STAT = 3, contact is closed and no sliding occurs
- If STAT = 1, contact is open
- If STAT = 2, node J slides relative to node I

For a frictionless surface (μ = 0.0), the converged element status is either STAT = 2 or 1.

The element coordinate system orientation angles α and β (shown in Figure 178.1: CONTA178 Geometry (p. 769)) are computed by the program from the node locations. These values are printed as ALPHA and BETA respectively. α ranges from 0° to 360° and β from -90° to +90°. Elements lying along the Z-axis are assigned values of α = 0°, β = ± 90°, respectively. Elements lying off the Z-axis have their coordinate system oriented as shown for the general α , β position.

Note

For α = 90°, $\beta \rightarrow$ 90°, the element coordinate system flips 90° about the Z-axis. The value of ANGLE represents the principal angle of the friction force in the element y-z plane. A general description of solution output is given in Element Solution (p. 51). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 178.3: CONTA178 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
XC, YC, ZC	Location where results are reported	Y	3
TEMP	T(I), T(J)	Y	Y
USEP	Gap size (gap = negative value; penetration = positive value)	Y	Y
FN	Normal force (along I-J line)	Y	Y
STAT	Element status	1	1
OLDST	Old contact status	1	1
ALPHA, BETA	Element orientation angles	Y	Y
MU	Coefficient of friction	2	2
UT(Y, Z)	Displacement (node J - node I) in element y and z directions	2	2
FS(Y, Z)	Tangential (friction) force in element y and z directions	2	2
ANGLE	Principal angle of friction force in element y-z plane	2	2
AREA	Cross-sectional area	Y	Y
VELOC	Normal velocity used for damping calculation	Y	Y
DAMPF	Damping force	Y	Y
AUTY	Total sliding in y direction		Y
AUTZ	Total sliding in z direction		Y
ELSI	Total equivalent elastic slip distance		Y
PLSI	Total (accumulated) equivalent plastic slip due to frictional sliding		Y
TEMP1	Temperature at node I	Y	Y
TEMP2	Temperature at node J	Y	Y
TCC	Conductance coefficient	Y	Y
CONV	Convection coefficient	Y	Y
RAC	Radiation coefficient	Y	Y
HFLUX1	Applied heat flux at node I	Y	Y
HFLUX2	Applied heat flux at node J	Y	Y
FLUX1	Total heat flow at node I	Y	Y
FLUX2	Total heat flow at node J	Y	Y
FDDIS	Frictional energy dissipation	Y	Y
FXCV	Heat flow due to convection	Y	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
FXRD	Heat flow due to radiation	Y	Y
FXCD	Heat flow due to conductance	Y	Y
ECC	Electric contact conductance (for electric current DOF), or electric contact capacitance per unit area (for piezoelectric or electrostatic DOF)	Y	Y
VOLT1	Voltage at node I	Y	Y
VOLT2	Voltage at node J	Y	Y
HJOU	Contact power/area	Y	Y
CCONT	Contact charge density (charge/area)	Y	Y
JCONT	Contact current density (current/area)	Y	Y

1. If the value of STAT is:

   1 - Open contact

   2 - Sliding contact

   3 - Sticking contact (no sliding)

2. If MU>0.0

3. Available only at centroid as a ***GET** item.

Table 178.4: CONTA178 Item and Sequence Numbers (p. 788) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* for more information. The following notation is used in Table 178.4: CONTA178 Item and Sequence Numbers (p. 788) :

Name
output quantity as defined in the Element Output Definition

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 178.4: CONTA178 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FN	SMISC	1
FSY	SMISC	2
FSZ	SMISC	3
FLUX1	SMISC	4
FLUX2	SMISC	5

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FDDIS	SMISC	6
HFLUX1	SMISC	7
HFLUX2	SMISC	8
FXCV	SMISC	9
FXRD	SMISC	10
FXCD	SMISC	11
HJOU	SMISC	12
CCONT	SMISC	13
JCONT	SMISC	13
STAT	NMISC	1
OLDST	NMISC	2
USEP	NMISC	3
ALPHA	NMISC	4
BETA	NMISC	5
UTY	NMISC	6
UTZ	NMISC	7
MU	NMISC	8
ANGLE	NMISC	9
KN	NMISC	10
KS	NMISC	11
TOLN	NMISC	12
FTOL	NMISC	13
SLTOL	NMISC	14
ELSI	NMISC	15
PLSI	NMISC	16
AREA	NMISC	17
VELOC	NMISC	18
DAMPF	NMISC	19
AUTY	NMISC	20
AUTZ	NMISC	21
TEMP1	NMISC	22
TEMP2	NMISC	23
CONV	NMISC	24
RAC	NMISC	25
TCC	NMISC	26
ECC	NMISC	27

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
VOLT1	NMISC	28
VOLT2	NMISC	29

You can list contact results through several POST1 postprocessor commands. The contact specific items for the **PRNSOL** and **PRESOL** commands are listed below:

STAT	Contact status
PENE	Contact penetration
PRES	Contact force
SFRIC	Contact tangential force
STOT	Total contact tangential force
SLIDE	Contact sliding distance
GAP	Contact gap distance
FLUX	Total heat flow at contact node

CONTA178 Assumptions and Restrictions

- The element operates bilinearly only in static and nonlinear transient dynamic analyses. If used in other analysis types, the element maintains its initial status throughout the analysis.
- The element is nonlinear and requires an iterative solution.
- Nonconverged substeps are not in equilibrium.
- Unless the contact normal direction is specified by (NX, NY, NZ) or KEYOPT(5), nodes I and J must not be coincident or overlapped since the nodal locations define the interface orientation. In this case the node ordering is not an issue. On the other hand, if the contact normal is not defined by nodal locations, the node ordering is critical. Use **/PSYMB**, **ESYS** to verify the contact normal and use **EINTF**,,,REVE to reverse the normal if wrong ordering is detected. To determine which side of the interface contains the nodes, use **ESEL**,,ENAM,,178 and then **NSLE**,,POS,1.
- The element maintains its original orientation in either a small or a large deflection analysis unless the cylindrical gap option is used.
- For real constants FKN, REDFACT, TOLN, FTOL, SLTOL and FKS, you can specify either a positive or negative value. ANSYS interprets a positive value as a scaling factor and interprets a negative value as the absolute value. These real constants can be changed between load steps or during restart stages.
- The Lagrange multiplier methods introduce zero diagonal terms in the stiffness matrix. The PCG solver may encounter precondition matrix singularity. The Lagrange multiplier methods often overconstrain the model if boundary conditions, coupling, and constraint equations applied on the contact nodes overlay the contact constraints. Chattering is most likely to occur due to change of contact status, typically for contact impact problems. The Lagrange multipliers also introduce more degrees of freedom which may result in spurious modes for modal and linear eigenvalue bucking analysis. Therefore, the augmented Lagrange method option is the best choice for: PCG iterative solver, transient analysis for impact problems, modal, and eigenvalue bucking analysis.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The element may not be deactivated with the **EKILL** command.

CONTA178 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

PRETS179

Pretension

MP ME ST PR PRN DS <> <> PP EME MFS
Product Restrictions

PRETS179 Element Description

PRETS179 is used to define a 2-D or 3-D pretension section within a meshed structure. The structure can be built from any 2-D or 3-D structural elements (solid, beam, shell, pipe, or link). The PRETS179 element has one translation degree of freedom, UX. (UX represents the defined pretension direction. ANSYS transforms the geometry of the problem so that, internally, the pretension force is applied in the specified pretension load direction, regardless of how the model is defined.) Loads can be applied using the **SLOAD** command. These loads will overwrite any F or D command specifications on the same nodes at solution time. Only tension loads can be applied; bending or torsion loads are ignored. See PRETS179 in the *Mechanical APDL Theory Reference* for more details about the element. See Defining Pretension in a Joint Fastener in the *Basic Analysis Guide* for a discussion of how to generate PRETS179 elements automatically using the **PSMESH** command. Keep in mind when creating the PRETS179 elements that the pretension load direction is specified relative to surface A. (For backward compatibility, it is also possible to generate such elements using the **EINTF** command.)

Figure 179.1: PRETS179 Geometry

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PRETS179 Input Data

The pretension section is modeled by a set of pretension elements. The geometry, node locations, and the coordinate system for the pretension element are shown in Figure 179.1: PRETS179 Geometry (p. 793). The pretension element is defined by three nodes I, J, K and the section data NX, NY, NZ which define the pretension load direction relative to surface A. The pretension load direction is constant and is not updated for large displacements. Although it is not recommended, the pretension load direction can be changed between load steps by changing the section data. For large-deflection problems, you could track the deflection and change the pretension load direction accordingly.

Nodes I and J are initially coincident and they should be defined in the same nodal coordinate system. No boundary conditions apply on node J. For each pretension section, the node ordering of the pretension elements is critical. The I and J nodes must be ordered so that all nodes I are on surface A and all nodes J are on surface B.

Node K is the pretension node. This pretension node provides a convenient way to assign boundary conditions on an entire pretension section. Node K can be anywhere in space; however, its nodal coordinate system must be global Cartesian. Each pretension section has only one pretension node associated with it. Node K should only connect to pretension elements that use the same section number.

The pretension node K has only one translation degree of freedom UX, which defines the relative displacement between the two sections A and B in the pretension load direction. Sliding motion is prevented automatically. If the pretension node and the bolted structure are not well constrained, rigid body motion can occur. Therefore, in the beginning of each load step, you should verify the boundary conditions for bolt structures carefully.

The following table summarizes the element input. Element Input (p. 43) gives a general description of element input.

PRETS179 Input Summary

Nodes
I, J, K

Degrees of Freedom
UX (tightening adjustment of the pretension section)

Real Constants
None

Material Properties
MP command: BETD

Surface Loads
None

Body Loads
None

Special Features

Linear perturbation
Nonlinearity

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPTs

None

PRETS179 Output Data

Nodal displacements are included in the overall displacement solution. There is no printed or post element data output for the pretension element. ANSYS automatically determines the deformations of the bolt structure. The underlying elements connected to both sides of cutting surfaces appear overlap under the pretension load. The displacement of the pretension node gives the adjustment of the pretension. Use **PRNSOL** to list the adjustment. The reaction force on the pretension node provides the total normal force across the pretension section. Use **PRRSOL** or **PRRFOR** command to list the tension force. The stress distribution of underlying elements provides a good estimation of the stress across the pretension section.

PRETS179 Assumptions and Restrictions

- The nodal coordinate system of the pretension node K must be global Cartesian.
- You cannot apply any constraint equations (or coupling) on any pretension element nodes.
- The **NROTAT** command can not be applied on pretension node K. **NROTAT** can be applied to the other nodes I and J in such way that they are rotated into the same nodal coordinate system. If K has been mistakenly rotated into another coordinate system, ANSYS will issue a warning and will automatically rotate it back into the global Cartesian system. Similarly, if I and J are rotated into different coordinate systems, ANSYS will issue a warning and will automatically rotate J to be consistent with I.
- The pretension normal NX, NY, NZ must be specified through section data. You should not change section data either between load steps or during restart stages; otherwise ANSYS assumes the pretension normal varies between the load steps.
- The pretension force is actually applied in the direction of the pretension section normal. However, the pretension force is always specified and displayed in the global X direction since the element has only one degree of freedom, UX.
- The structure can be composed of superelements. However, all the pretension nodes must remain as the master nodes.
- The element may not be deactivated with the **EKILL** command.
- Use of this element is limited to structural analyses only.

PRETS179 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

LINK180

3-D Spar (or Truss)

MP ME ST PR PRN <> <> <> PP EME MFS
Product Restrictions

LINK180 Element Description

LINK180 is a 3-D spar that is useful in a variety of engineering applications. The element can be used to model trusses, sagging cables, links, springs, and so on. The element is a uniaxial tension-compression element with three degrees of freedom at each node: translations in the nodal x, y, and z directions. Tension-only (cable) and compression-only (gap) options are supported. As in a pin-jointed structure, no bending of the element is considered. Plasticity, creep, rotation, large deflection, and large strain capabilities are included.

By default, LINK180 includes stress-stiffness terms in any analysis that includes large-deflection effects. Elasticity, isotropic hardening plasticity, kinematic hardening plasticity, Hill anisotropic plasticity, Chaboche nonlinear hardening plasticity, and creep are supported. To simulate the tension-/compression-only options, a nonlinear iterative solution approach is necessary; therefore, large-deflection effects must be activated (**NLGEOM**,ON) prior to the solution phase of the analysis. Added mass, hydrodynamic added mass and loading, and buoyant loading are available.

See LINK180 in the *Mechanical APDL Theory Reference* for more information about this element.

Figure 180.1: LINK180 Geometry

LINK180 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 180.1: LINK180 Geometry (p. 797). The element is defined by two nodes, the cross-sectional area (*A*) input via the **SECTYPE** and **SECDATA** commands, added mass per unit length (*ADDMAS*) input via the **SECCONTROL** command, and the material properties.

The element x-axis is oriented along the length of the element from node I toward node J. If ocean loading is present, the global origin is normally at the mean sea level, with the global Z-axis pointing away from the center of the earth; however, the vertical location can be adjusted via `Zmsl` (`Val6`) on the **OCDATA** command (following the **OCTYPE**,BASIC command).

Element loads are described in Nodal Loading (p. 49). Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. The node J temperature T(J) defaults to T(I).

LINK180 allows a change in cross-sectional area as a function of axial elongation. By default, the cross-sectional area changes such that the volume of the element is preserved, even after deformation. The

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

default is suitable for elastoplastic applications. By using KEYOPT(2), you may choose to keep the cross section constant or rigid.

LINK180 offers compression-and-tension, tension-only, and compression-only options. Specify the desired behavior via KEYOPT(3). (See "LINK180 Input Summary" (p. 798) for details.) A nonlinear solution procedure is necessary for these options; for more information, see the documentation for the **SOLCONTROL** command.

For ocean loading, hydrodynamic added mass and loading, and buoyant loading, are available via the **OCDATA** and **OCTABLE** commands.

When ocean loading is applied, the loading is nonlinear (that is, based on the square of the relative velocity between the structure and the water). Accordingly, the full Newton-Raphson option (**NROPT**,FULL) may be necessary to achieve optimal results. (Full Newton-Raphson is applied automatically in an analysis involving large-deflection effects [**NLGEOM**,ON].)

You can apply an initial stress state to this element via the **INISTATE** command. For more information, see Initial State in the *Basic Analysis Guide*.

The "LINK180 Input Summary" (p. 798) table summarizes the element input. Element Input (p. 43) gives a general description of element input.

LINK180 Input Summary

Nodes
I, J

Degrees of Freedom
UX, UY, UZ

Material Properties
TB command: See Element Support for Material Models (p. 59) for this element.

MP command: EX, (PRXY or NUXY), ALPX (or CTEX *or* THSX), DENS, GXY, ALPD, BETD

Surface Loads
None

Body Loads

Temperatures --
T(I), T(J)

Special Features

Birth and death
Initial state
Large deflection
Large strain
Linear perturbation
Nonlinear stabilization
Ocean loading
Stress stiffening

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(2)

Cross-section scaling (applies only when large-deflection effects [**NLGEOM**,ON] are specified):

0 --

Enforce incompressibility; cross section is scaled as a function of axial stretch (default).

1 --

Section is assumed to be rigid.

KEYOPT(3)

Tension and/or compression option:

0 --

Tension and compression (default).

1 --

Tension only.

2 --

Compression only.

KEYOPT(12)

Hydrodynamic output (not available in harmonic analyses that include ocean wave effects (**HROCEAN**)):

0 --

None (default)

1 --

Additional centroidal hydrodynamic printout

LINK180 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 180.1: LINK180 Element Output Definitions (p. 800).

Several items are illustrated in Figure 180.2: LINK180 Stress Output (p. 799). A general description of solution output is given in Solution Output (p. 50). Element results can be viewed in POST1 with **PRESOL**,ELEM. See the *Basic Analysis Guide* for details.

Figure 180.2: LINK180 Stress Output

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 180.1: LINK180 Element Output Definitions

Name	Definition	O	R
EL	Element number	Y	Y
NODES	Nodes - I, J	Y	Y
MAT	Material number	Y	Y
SECID	Section number	Y	-
XC, YC, ZC	Center location	Y	1
TEMP	Temperatures T(I), T(J)	Y	Y
AREA	Cross-sectional area	Y	Y
FORCE	Member force in the element coordinate system	Y	Y
Sxx	Axial stress	Y	Y
EPELxx	Axial elastic strain	Y	Y
EPTOxx	Total strain	Y	Y
EPEQ	Plastic equivalent strain	2	2
Cur.Yld.Flag	Current yield flag	2	2
Plwk	Plastic strain energy density	2	2
Pressure	Hydrostatic pressure	2	2
Creq	Creep equivalent strain	2	2
Crwk_Creep	Creep strain energy density	2	2
EPPLxx	Axial plastic strain	2	2
EPCRxx	Axial creep strain	2	2
EPTHxx	Axial thermal strain	3	3
EXT PRESS	External pressure at integration point	4	4
EFFECTIVE TENS	Effective tension on link	4	4
The following values apply to ocean loading only:			
GLOBAL COORD	Element centroid location	5	Y
VR, VZ	Radial and vertical fluid particle velocities (VR is always > 0)	5	Y
AR, AZ	Radial and vertical fluid particle accelerations	5	Y
PHDYN	Dynamic fluid pressure head	5	Y
ETA	Wave amplitude over integration point	5	Y
TFLUID	Fluid temperature (printed if VISC is nonzero)	5	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
VISC	Viscosity (output if VISC is nonzero)	5	Y
REN, RET	Normal and tangential Reynolds numbers (if VISC is nonzero)	5	Y
CT	Input tangential drag coefficients evaluated at Reynolds numbers	5	Y
CDY, CDZ	Input normal drag coefficients evaluated at Reynolds numbers	5	Y
CMY, CMZ	Input inertia coefficients evaluated at Reynolds numbers	5	Y
URT, URN	Tangential (parallel to element axis) and normal relative velocities	5	Y
ABURN	Vector sum of normal (URN) velocities	5	Y
AN	Accelerations normal to element	5	Y
FX, FY, FZ	Hydrodynamic tangential and normal forces in element coordinates	5	Y
ARGU	Effective position of wave (radians)	5	Y

1. Available only at the centroid as a ***GET** item.
2. Available only if the element has an appropriate nonlinear material.
3. Available only if the element temperatures differ from the reference temperature.
4. Available only if ocean loading is present.
5. See KEYOPT(12) description.

The element printout also includes 'INT, SEC PTS' (which are always '1, Y Z' where Y and Z both have values of 0.0). These values are printed to maintain formatting consistency with the output printouts of the BEAM188, BEAM189, PIPE288, and PIPE289 elements.

Table 180.2: LINK180 Item and Sequence Numbers (p. 802) lists output available through **ETABLE** using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 180.2: LINK180 Item and Sequence Numbers (p. 802):

Name
output quantity as defined in Table 180.1: LINK180 Element Output Definitions (p. 800)

Item
predetermined Item label for **ETABLE** and

ESOL

E
sequence number for single-valued or constant element data

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

I,J

sequence number for data at nodes I and J

Table 180.2: LINK180 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
Sxx	LS	-	1	2
EPELxx	LEPEL	-	1	2
EPTOxx	LEPTO	-	1	2
EPTHxx	LEPTH	-	1	2
EPPLxx	LEPPL	-	1	2
EPCRxx	LEPCR	-	1	2
FORCE	SMISC	1	-	-
AREA	SMISC	2	-	-
EXT PRESS [1]	SMISC	3	-	-
EFFECTIVE TENS [1]	SMISC	4	-	-
TEMP	LBFE	-	1	2
The following output quantities are valid for ocean loading only:				
GLOBAL COORD	NMISC	1, 2, 3	--	--
VR, VZ	NMISC	4, 5	--	--
AR, AZ	NMISC	6, 7 [2]	--	--
PHDYN	NMISC	8 [2]	--	--
ETA	NMISC	9 [2]	--	--
TFLUID	NMISC	10	--	--
VISC	NMISC	11	--	--
REN, RET	NMISC	12, 13 [3]	--	--
CT	NMISC	14	--	--
CDY, CDZ	NMISC	15, 16	--	--
CMY, CMZ	NMISC	17, 18 [2]	--	--
URT, URN	NMISC	19, 20, 21	--	--
ABURN	NMISC	22 [2]	--	--

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
AN	NMISC	23, 24 [2]	--	--
FX, FY, FZ	NMISC	25, 26, 27	--	--
ARGU	NMISC	28 [2]	--	--

1. External pressure (EXT PRESS) and effective tension (EFFECTIVE TENS) occur at mid-length.
2. See KEYOPT(12) description.
3. These quantities are output only if a Reynold's number dependency is used.

LINK180 Assumptions and Restrictions

- The spar element assumes a straight bar, axially loaded at its ends, and of uniform properties from end to end.
- The length of the spar must be greater than zero, so nodes I and J must not be coincident.
- The cross-sectional area must be greater than zero.
- The temperature is assumed to vary linearly along the length of the spar.
- The displacement shape function implies a uniform stress in the spar.
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). Prestress effects can be activated by the **PSTRES** command.
- To simulate the tension-/compression-only options, a nonlinear iterative solution approach is necessary.
- The output axial force may not be exact when using ocean loading with nonlinear materials.
- When the element is used in an ocean environment:
  - Hydrodynamic output via KEYOPT(12) is not available in harmonic analyses that include ocean wave effects (**HROCEAN**).
  - The three-dimensional effect of water pressure on the element is adjusted, as the element has only one direct stress. For more information, see Hydrostatic Loads in the *Mechanical APDL Theory Reference*.

LINK180 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

SHELL181

4-Node Structural Shell

MP ME ST PR PRN DS <> <> PP EME MFS
Product Restrictions

SHELL181 Element Description

SHELL181 is suitable for analyzing thin to moderately-thick shell structures. It is a four-node element with six degrees of freedom at each node: translations in the x, y, and z directions, and rotations about the x, y, and z-axes. (If the membrane option is used, the element has translational degrees of freedom only). The degenerate triangular option should only be used as filler elements in mesh generation.

SHELL181 is well-suited for linear, large rotation, and/or large strain nonlinear applications. Change in shell thickness is accounted for in nonlinear analyses. In the element domain, both full and reduced integration schemes are supported. SHELL181 accounts for follower (load stiffness) effects of distributed pressures.

SHELL181 can be used for layered applications for modeling composite shells or sandwich construction. The accuracy in modeling composite shells is governed by the first-order shear-deformation theory (usually referred to as Mindlin-Reissner shell theory).

The element formulation is based on logarithmic strain and true stress measures. The element kinematics allow for finite membrane strains (stretching). However, the curvature changes within a time increment are assumed to be small.

See SHELL181 in the *Mechanical APDL Theory Reference* for more details about this element.

SHELL181 Input Data

The following figure shows the geometry, node locations, and the element coordinate system for this element. The element is defined by shell section information and by four nodes (I, J, K, and L).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 181.1: SHELL181 Geometry

x_o = Element x-axis if ESYS is not provided.

x = Element x-axis if ESYS is provided.

Single-Layer Definition

To define the thickness (and other information), use section definition, as follows:

SECTYPE,,SHELL
SECDATA,*THICKNESS*, ...

A single-layer shell section definition provides flexible options. For example, you can specify the number of integration points used and the material orientation.

Multilayer Definition

The shell section commands allow for layered shell definition. Options are available for specifying the thickness, material, orientation, and number of integration points through the thickness of the layers.

You can designate the number of integration points (1, 3, 5, 7, or 9) located through the thickness of each layer when using section input. When only one, the point is always located midway between the top and bottom surfaces. If three or more points, two points are located on the top and bottom surfaces respectively and the remaining points are distributed equal distance between the two points. The default number of integration points for each layer is three; however, when a single layer is defined and plasticity is present, the number of integration points is changed to a minimum of five during solution.

The following additional capabilities are available when defining shell layers:

- SHELL181 accepts the preintegrated shell section type (**SECTYPE**,,GENS).

  When the element is associated with the GENS section type, thickness or material definitions are not required.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You can use the function tool to define thickness as a function of global/local coordinates or node numbers (**SECFUNCTION**).
- You can specify offsets (**SECOFFSET**).
- A section can be partially defined using data from a FiberSIM `.xml` file (**SECTYPE**,,SHELL,FIBERSIM).

Other Input

The default orientation for this element has the S_1 (shell surface coordinate) axis aligned with the first parametric direction of the element at the center of the element, which connects the midsides of edges LI and JK and is shown as x_o in Figure 181.1: SHELL181 Geometry (p. 806). In the most general case, the axis can be defined as:

$$S_1 = \frac{\partial\{x\}}{\partial s} \Big/ \left(\left| \frac{\partial\{x\}}{\partial s} \right| \right)$$

where:

$$\{x\} = \sum_{i=1}^{8} h^i(s,r)\{x\}^i$$

$\{x\}^I, \{x\}^J, \{x\}^K, \{x\}^L$ = global nodal coordinates

For undistorted elements, the default orientation is the same as described in Coordinate Systems (p. 55) (the first surface direction is aligned with the IJ side). For spatially warped or otherwise distorted elements, the default orientation represents the stress state better because the element uses a single point of quadrature (by default) in the element domain.

The first surface direction S_1 can be rotated by angle θ (in degrees) for the layer via the **SECDATA** command. For an element, you can specify a single value of orientation in the plane of the element. Layer-wise orientation is supported.

You can also define element orientation via the **ESYS** command. See Coordinate Systems (p. 55).

The element supports degeneration into a triangular form; however, use of the triangular form is *not* recommended, except when used as mesh filler elements or with the membrane option (KEYOPT(1) = 1). The triangle form is generally more robust when using the membrane option with large deflections.

To evaluate stresses and strains on exterior surfaces, use KEYOPT(1) = 2. When used as overlaid elements on the faces of 3-D elements, this option is similar to the surface stress option (described in the *Mechanical APDL Theory Reference*), but is more general and applicable to nonlinear analysis. The element used with this option does not provide any stiffness, mass, or load contributions. This option should only be used in single-layer shells. Irrespective of other settings, SHELL181 provides stress and strain output at the center of the layer.

SHELL181 uses a penalty method to relate the independent rotational degrees of freedom about the normal (to the shell surface) with the in-plane components of displacements. The program chooses an appropriate penalty stiffness by default. A drill stiffness factor can be specified via the **SECCONTROL** command.

Element loads are described in Nodal Loading (p. 49). Pressures can be input as surface loads on the element faces as shown by the circled numbers on Figure 181.1: SHELL181 Geometry (p. 806). Positive

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

pressures act into the element. Because shell edge pressures are input on a per-unit-length basis, per-unit-area quantities must be multiplied by the shell thickness.

Temperatures can be input as element body loads at the corners of the outside faces of the element and at the corners of the interfaces between layers. The first corner temperature T1 defaults to TUNIF. If all other temperatures are unspecified, they default to T1. If KEYOPT(1) = 0 and if exactly NL+1 temperatures are input, one temperature is used for the four bottom corners of each layer, and the last temperature is used for the four top corner temperatures of the top layer. If KEYOPT(1) = 1 and if exactly NL temperatures are input, one temperature is used for the four corners of each layer. That is, T1 is used for T1, T2, T3, and T4; T2 (as input) is used for T5, T6, T7, and T8, etc. For any other input pattern, unspecified temperatures default to TUNIF.

Using KEYOPT(3), SHELL181 supports uniform reduced integration and full integration with incompatible modes. By default, this element uses the uniform reduced integration for performance reasons in non-linear applications.

Using reduced integration with hourglass control creates some usage restrictions, although minimal. For example, to capture the in-plane bending of a cantilever or a stiffener (see Figure 181.2: SHELL181 Typical Bending Applications (p. 809)), a number of elements through the thickness direction is necessary. The performance gains achieved by using uniform reduced integration are significant enough to offset the need to use more elements. In relatively well-refined meshes, hourglassing issues are largely irrelevant.

When the reduced integration option is used, you can check the accuracy of the solution by comparing the total energy (SENE label in **ETABLE**) and the artificial energy (AENE label in **ETABLE**) introduced by hourglass control. If the ratio of artificial energy to total energy is less than 5%, the solution is generally acceptable. The total energy and artificial energy can also be monitored by using **OUTPR**,VENG in the solution phase.

Bilinear elements, when fully integrated, are too stiff in in-plane bending.SHELL181 uses the method of incompatible modes to enhance the accuracy in bending-dominated problems. This approach is also called "extra shapes" or "bubble" modes approach. SHELL181 uses the formulation that ensures satisfaction of the patch test (J. C. Simo and F. Armero, "Geometrically nonlinear enhanced strain mixed methods and the method of incompatible modes," *IJNME*, Vol. 33, pp. 1413-1449, 1992).

When including incompatible modes in the analysis, you must use full integration. KEYOPT(3) = 2 implies the inclusion of incompatible modes and the use of full (2x2) quadrature.

SHELL181, with KEYOPT(3) = 2 specified, does not have any spurious energy mechanisms. This specific form of SHELL181 is highly accurate, even with coarse meshes. Use KEYOPT(3) = 2 if you encounter any hourglass-related difficulties with the default options. KEYOPT(3) = 2 is also necessary if the mesh is coarse and in-plane bending of the elements dominate the response. We recommend this option with all layered applications.

KEYOPT(3) = 2 imposes the fewest usage restrictions. You can always choose this option. However, you can improve element performance by choosing the best option for your problem. Consider the problems illustrated in Figure 181.2: SHELL181 Typical Bending Applications (p. 809)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 181.2: SHELL181 Typical Bending Applications

The cantilever beam and the beam cross-section to be modeled with shells are typical examples of in-plane bending-dominated problems. The use of KEYOPT(3) = 2 is the most effective choice in these circumstances. Reduced integration would require refined meshes. For example, reduced integration for the cantilever beam problem requires four elements through the thickness, whereas the full integration with incompatible modes only requires one element through the thickness.

For the stiffened shell, the most effective choice is to use KEYOPT(3) = 0 for the shell and KEYOPT(3) = 2 for the stiffener.

When KEYOPT(3) = 0 is specified, SHELL181 uses an hourglass control method for membrane and bending modes. By default, SHELL181 calculates the hourglass parameters for both metal and hyperelastic applications. To specify the hourglass stiffness scaling factors, use the **SECCONTROL** command.

When KEYOPT(5) = 1, the element incorporates initial curvature effects. The calculation for effective shell curvature change accounts for both shell-membrane and thickness strains. The formulation generally offers improved accuracy in curved shell structure simulations, especially when thickness strain is significant or the material anisotropy in the thickness direction cannot be ignored, or in thick shell structures with unbalanced laminate construction or with shell offsets. The initial curvature of each element is calculated from the nodal shell normals. The shell normal at each node is obtained by averaging the shell normals from the surrounding SHELL181 elements. A coarse or highly distorted shell mesh can

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

lead to significant error in the recovered element curvature; therefore, this option should only be used with a smooth and adequately refined mesh. To ensure proper representation of the original mesh, a nodal normal is replaced by the element shell normal in the curvature calculation if the subtended angle between these two is greater than 25 degrees.

SHELL181 includes the linear effects of transverse shear deformation. An assumed shear strain formulation of Bathe-Dvorkin is used to alleviate shear locking. The transverse shear stiffness of the element is a 2x2 matrix as shown below:

$$E = \begin{bmatrix} E_{11} & E_{12} \\ \text{sym} & E_{22} \end{bmatrix}$$

To define transverse shear stiffness values, use the **SECCONTROL** command.

For a single-layer shell with isotropic material, default transverse shear stiffnesses are:

$$E = \begin{bmatrix} kGh & 0 \\ 0 & kGh \end{bmatrix}$$

In the above matrix, k = 5/6, G = shear modulus, and h = thickness of the shell.

SHELL181 can be associated with linear elastic, elastoplastic, creep, or hyperelastic material properties. Only isotropic, anisotropic, and orthotropic linear elastic properties can be input for elasticity. The von Mises isotropic hardening plasticity models can be invoked with BISO (bilinear isotropic hardening), MISO (multilinear isotropic hardening), and NLISO (nonlinear isotropic hardening) options. The kinematic hardening plasticity models can be invoked with BKIN (bilinear kinematic hardening), MKIN and KINH (multilinear kinematic hardening), and CHABOCHE (nonlinear kinematic hardening). Invoking plasticity assumes that the elastic properties are isotropic (that is, if orthotropic elasticity is used with plasticity, ANSYS assumes the isotropic elastic modulus = EX and Poisson's ratio = NUXY).

Hyperelastic material properties (2, 3, 5, or 9 parameter Mooney-Rivlin material model, Neo-Hookean model, Polynomial form model, Arruda-Boyce model, and user-defined model) can be used with this element. Poisson's ratio is used to specify the compressibility of the material. If less than 0, Poisson's ratio is set to 0; if greater than or equal to 0.5, Poisson's ratio is set to 0.5 (fully incompressible).

Both isotropic and orthotropic thermal expansion coefficients can be input using **MP**,ALPX. When used with hyperelasticity, isotropic expansion is assumed.

Use the **TREF** command to specify the global value of reference temperature. If **MP**,REFT is defined for the material number of the element, it is used for the element instead of the value from the **TREF** command. But if **MP**,REFT is defined for the material number of the layer, it is used instead of either the global or element value.

With reduced integration and hourglass control (KEYOPT(3) = 0), low frequency spurious modes can appear if the mass matrix employed is not consistent with the quadrature rule. SHELL181 uses a projection scheme that effectively filters out the inertia contributions to the hourglass modes of the element. To be effective, a consistent mass matrix must be used. We recommend setting **LUMPM**,OFF for a modal analysis using this element type. The lumped mass option can, however, be used with the full integration options (KEYOPT(3) = 2).

KEYOPT(8) = 2 stores midsurface results in the results file for single or multi-layer shell elements. If you use **SHELL**,MID, you will see these calculated values, rather than the average of the TOP and BOTTOM

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

results. You should use this option to access these correct midsurface results (membrane results) for those analyses where averaging TOP and BOTTOM results is inappropriate; examples include midsurface stresses and strains with nonlinear material behavior, and midsurface results after mode combinations that involve squaring operations such as in spectrum analyses.

KEYOPT(9) = 1 reads initial thickness data from a user subroutine.

You can apply an initial stress state to this element via the **INISTATE** command. For more information, see Initial State in the *Basic Analysis Guide*.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

A summary of the element input is given in "SHELL181 Input Summary" (p. 811). A general description of element input is given in Element Input (p. 43).

SHELL181 Input Summary

Nodes

I, J, K, L

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ if KEYOPT(1) = 0

UX, UY, UZ if KEYOPT(1) = 1

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, EY, EZ, (PRXY, PRYZ, PRXZ, or NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ),
DENS, GXY, GYZ, GXZ, ALPD
Specify BETD, ALPD, and DMPR for the element (all layers) by issuing the **MAT** command to assign the material property set.
REFT can be specified once for the element, or it can be assigned on a per-layer basis. See the discussion in "SHELL181 Input Summary" (p. 811) for more information.

Surface Loads

Pressures --

face 1 (I-J-K-L) (bottom, in +N direction),
face 2 (I-J-K-L) (top, in -N direction),
face 3 (J-I), face 4 (K-J), face 5 (L-K), face 6 (I-L)

Body Loads

Temperatures --

For KEYOPT(1) = 0 (Bending and membrane stiffness):

T1, T2, T3, T4 (at bottom of layer 1), T5, T6, T7, T8 (between layers 1-2); similarly for between next layers, ending with temperatures at top of layer NL(4*(NL+1) maximum). Hence, for one-layer elements, 8 temperatures are used.

For KEYOPT(1) = 1 (Membrane stiffness only):

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

T1, T2, T3, T4 for layer 1, T5, T6, T7, T8 for layer 2, similarly for all layers (4*NL maximum). Hence, for one-layer elements, 4 temperatures are used.

Special Features

Birth and death
Element technology autoselect
Initial state
Large deflection
Large strain
Linear perturbation
Nonlinear stabilization
Section definition for layered shells and preintegrated general shell sections for input of homogeneous section stiffnesses
Stress stiffening

KEYOPT(1)
Element stiffness:

0 --
Bending and membrane stiffness (default)

1 --
Membrane stiffness only

2 --
Stress/strain evaluation only

KEYOPT(3)
Integration option:

0 --
Reduced integration with hourglass control (default)

2 --
Full integration with incompatible modes

KEYOPT(5)
Curved shell formulation:

0 --
Standard shell formulation (default)

1 --
Advanced shell formation to account for initial shell-curvature effects

KEYOPT(8)
Specify layer data storage:

0 --
Store data for bottom of bottom layer and top of top layer (multi-layer elements) (default)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
Store data for TOP and BOTTOM, for all layers (multi-layer elements)

> **Note**
>
> Volume of data may be excessive.

2 --
Store data for TOP, BOTTOM, and MID for all layers; applies to single- and multi-layer elements

KEYOPT(9)
User thickness option:

0 --
No user subroutine to provide initial thickness (default)

1 --
Read initial thickness data from user subroutine UTHICK

> **Note**
>
> See the *Guide to User-Programmable Features* for user written subroutines

SHELL181 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 181.1: SHELL181 Element Output Definitions (p. 815)

Several items are illustrated in Figure 181.3: SHELL181 Stress Output (p. 814).

KEYOPT(8) controls the amount of data output to the results file for processing with the **LAYER** command. Interlaminar shear stress is available as SYZ and SXZ evaluated at the layer interfaces. KEYOPT(8) must be set to either 1 or 2 to output these stresses in POST1. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to review results.

The element stress resultants (N11, M11, Q13, etc.) are parallel to the element coordinate system, as are the membrane strains and curvatures of the element. Such generalized strains are available through the SMISC option at the element centroid only. The transverse shear forces Q13, Q23 are available only in resultant form: that is, use SMISC,7 (or 8). Likewise, the transverse shear strains, γ_{13} and γ_{23}, are constant through the thickness and are only available as SMISC items (SMISC,15 and SMISC,16, respectively).

ANSYS computes moments (M11, M22, M12) with respect to the shell reference plane. By default, ANSYS adopts the shell midplane as the reference plane. To offset the reference plane to any other specified location, issue the **SECOFFSET** command. When there is a nonzero offset (L) from the reference plane to the midplane, moments with respect to the midplane ($\overline{M11}, \overline{M22}, \overline{M12}$) can be recovered from stress resultants with respect to the reference plane as follows:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\overline{M11} = M11 - L \times N11$$
$$\overline{M22} = M22 - L \times N22$$
$$\overline{M12} = M12 - L \times N12$$

SHELL181 does not support extensive basic element printout. POST1 provides more comprehensive output processing tools; therefore, ANSYS suggests using the **OUTRES** command to ensure that the required results are stored in the database.

Figure 181.3: SHELL181 Stress Output

x_o = Element x-axis if ESYS is not provided.
x = Element x-axis if ESYS is provided.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 181.1: SHELL181 Element Output Definitions

Name	Definition	O	R
EL	Element number and name	Y	Y
NODES	Nodes - I, J, K, L	-	Y
MAT	Material number	-	Y
THICK	Average thickness	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	-	4
PRES	Pressures P1 at nodes I, J, K, L; P2 at I, J, K, L; P3 at J,I; P4 at K,J; P5 at L,K; P6 at I,L	-	Y
TEMP	T1, T2, T3, T4 at bottom of layer 1, T5, T6, T7, T8 between layers 1-2, similarly for between next layers, ending with temperatures at top of layer NL(4*(NL+1) maximum)	-	Y
LOC	TOP, MID, BOT, or integration point location	-	1
S:X, Y, Z, XY, YZ, XZ	Stresses	3	1
S:1, 2, 3	Principal stresses	-	1
S:INT	Stress intensity	-	1
S:EQV	Equivalent stress	-	1
EPEL:X, Y, Z, XY	Elastic strains	3	1
EPEL:EQV	Equivalent elastic strains [7]	-	1
EPTH:X, Y, Z, XY	Thermal strains	3	1
EPTH:EQV	Equivalent thermal strains [7]	-	1
EPPL:X, Y, Z, XY	Average plastic strains	3	2
EPPL:EQV	Equivalent plastic strains [7]	-	2
EPCR:X, Y, Z, XY	Average creep strains	3	2
EPCR:EQV	Equivalent creep strains [7]	-	2
EPTO:X, Y, Z, XY	Total mechanical strains (EPEL + EPPL + EPCR)	3	-
EPTO:EQV	Total equivalent mechanical strains (EPEL + EPPL + EPCR)	-	-
NL:SEPL	Plastic yield stress	-	2
NL:EPEQ	Accumulated equivalent plastic strain	-	2
NL:CREQ	Accumulated equivalent creep strain	-	2
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	-	2
NL:PLWK	Plastic work/volume	-	2
NL:HPRES	Hydrostatic pressure	-	2

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
SEND:ELASTIC, PLASTIC, CREEP	Strain energy densities	-	2
N11, N22, N12	In-plane forces (per unit length)	-	Y
M11, M22, M12	Out-of-plane moments (per unit length)	-	8
Q13, Q23	Transverse shear forces (per unit length)	-	8
ε_{11}, ε_{22}, ε_{12}	Membrane strains	-	Y
k_{11}, k_{22}, k_{12}	Curvatures	-	8
γ_{13}, γ_{23}	Transverse shear strains	-	8
LOCI:X, Y, Z	Integration point locations	-	5
SVAR:1, 2, ... , N	State variables	-	6
ILSXZ	SXZ interlaminar shear stress	-	Y
ILSYZ	SYZ interlaminar shear stress	-	Y
ILSUM	Magnitude of the interlaminar shear stress vector	-	Y
ILANG	Angle of interlaminar shear stress vector (measured from the element x-axis toward the element y-axis in degrees)	-	Y
Sm: 11, 22, 12	Membrane stresses	-	Y
Sb: 11, 22, 12	Bending stresses	-	Y
Sp: 11, 22, 12	Peak stresses	-	Y
St: 13, 23	Averaged transverse shear stresses	-	Y

1. The following stress solution repeats for top, middle, and bottom surfaces.

2. Nonlinear solution output for top, middle, and bottom surfaces, if the element has a nonlinear material, or if large-deflection effects are enabled (**NLGEOM**,ON) for SEND.

3. Stresses, total strains, plastic strains, elastic strains, creep strains, and thermal strains in the element coordinate system are available for output (at all section points through thickness). If layers are in use, the results are in the layer coordinate system.

4. Available only at centroid as a ***GET** item.

5. Available only if **OUTRES**,LOCI is used.

6. Available only if the `UserMat` subroutine and **TB**,STATE command are used.

7. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.

8. Not available if the membrane element option is used (KEYOPT(1) = 1).

Table 181.2: SHELL181 Item and Sequence Numbers (p. 817) lists output available through **ETABLE** using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 181.2: SHELL181 Item and Sequence Numbers (p. 817):

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name

output quantity as defined in the Table 181.1: SHELL181 Element Output Definitions (p. 815)

Item

predetermined Item label for **ETABLE**

E

sequence number for single-valued or constant element data

I,J,K,L

sequence number for data at nodes I, J, K, L

Table 181.2: SHELL181 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
N11	SMISC	1	-	-	-	-
N22	SMISC	2	-	-	-	-
N12	SMISC	3	-	-	-	-
M11	SMISC	4	-	-	-	-
M22	SMISC	5	-	-	-	-
M12	SMISC	6	-	-	-	-
Q13	SMISC	7	-	-	-	-
Q23	SMISC	8	-	-	-	-
ε_{11}	SMISC	9	-	-	-	-
ε_{22}	SMISC	10	-	-	-	-
ε_{12}	SMISC	11	-	-	-	-
k_{11}	SMISC	12	-	-	-	-
k_{22}	SMISC	13	-	-	-	-
k_{12}	SMISC	14	-	-	-	-
γ_{13}	SMISC	15	-	-	-	-
γ_{23}	SMISC	16	-	-	-	-
THICK	SMISC	17	-	-	-	-
P1	SMISC	-	18	19	20	21
P2	SMISC	-	22	23	24	25
P3	SMISC	-	27	26	-	-
P4	SMISC	-	-	29	28	-
P5	SMISC	-	-	-	31	30
P6	SMISC	-	32	-	-	33
Sm: 11	SMISC	34	-	-	-	-
Sm: 22	SMISC	35	-	-	-	-
Sm: 12	SMISC	36	-	-	-	-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
Sb: 11	SMISC	37	-	-	-	-
Sb: 22	SMISC	38	-	-	-	-
Sb: 12	SMISC	39	-	-	-	-
Sp: 11 (at shell bottom)	SMISC	40	-	-	-	-
Sp: 22 (at shell bottom)	SMISC	41	-	-	-	-
Sp: 12 (at shell bottom)	SMISC	42	-	-	-	-
Sp: 11 (at shell top)	SMISC	43	-	-	-	-
Sp: 22 (at shell top)	SMISC	44	-	-	-	-
Sp: 12 (at shell top)	SMISC	45	-	-	-	-
St: 13	SMISC	46	-	-	-	-
St: 23	SMISC	47	-	-	-	-

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	Bottom of Layer i	Top of Layer NL
ILSXZ	SMISC	8 * (i - 1) + 51	8 * (NL - 1) + 52
ILSYZ	SMISC	8 * (i - 1) + 53	8 * (NL - 1) + 54
ILSUM	SMISC	8 * (i - 1) + 55	8 * (NL - 1) + 56
ILANG	SMISC	8 * (i - 1) + 57	8 * (NL - 1) + 58

SHELL181 Assumptions and Restrictions

- ANSYS, Inc. recommends against using this element in triangular form, except as a filler element. Avoid triangular form especially in areas with high stress gradients.
- Zero-area elements are not allowed. (Zero-area elements occur most often whenever the elements are numbered improperly.)
- Zero thickness elements or elements tapering down to a zero thickness at any corner are not allowed (but zero thickness layers are allowed).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If multiple load steps are used, the number of layers cannot change between load steps.
- When the element is associated with preintegrated shell sections (**SECTYPE**,,GENS), additional restrictions apply. For more information, see Considerations for Using Preintegrated Shell Sections.
- If reduced integration is used (KEYOPT(3) = 0) SHELL181 ignores rotary inertia effects when an unbalanced laminate construction is used, and all inertial effects are assumed to be in the nodal plane (that is, an unbalanced laminate construction and offsets have no effect on the mass properties of the element).
- For most composite analyses, ANSYS, Inc. recommends setting KEYOPT(3) = 2 (necessary to capture the stress gradients).
- No slippage is assumed between the element layers. Shear deflections are included in the element; however, normals to the center plane before deformation are assumed to remain straight after deformation.
- Transverse shear stiffness of the shell section is estimated by an energy equivalence procedure (of the generalized section forces and strains vs. the material point stresses and strains). The accuracy of this calculation may be adversely affected if the ratio of material stiffnesses (Young's moduli) between adjacent layers is very high.
- The calculation of interlaminar shear stresses is based on simplifying assumptions of unidirectional, un-coupled bending in each direction. If accurate edge interlaminar shear stresses are required, shell-to-solid submodeling should be used.
- The section definition permits use of hyperelastic material models and elastoplastic material models in laminate definition; however, the accuracy of the solution is primarily governed by fundamental assumptions of shell theory. The applicability of shell theory in such cases is best understood by using a comparable solid model.
- The layer orientation angle has no effect if the material of the layer is hyperelastic.
- Before using this element in a simulation containing curved thick shell structures with unbalanced laminate construction or shell offsets, validate the usage via full 3-D modeling with a solid element in a simpler representative model. The element may underestimate the curved thick shell stiffness, particularly when the offset is large and the structure is under torsional load. Consider using curved-shell formulation (KEYOPT(5) = 1).
- The through-thickness stress, SZ, is always zero.
- This element works best with the full Newton-Raphson solution scheme (**NROPT**,FULL,,ON).
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). Prestress effects can be activated by the **PSTRES** command.
- In a nonlinear analysis, the solution process terminates if the thickness at any integration point that was defined with a nonzero thickness vanishes (within a small numerical tolerance).
- If a shell section has only one layer and the number of section integration points is equal to one, or if KEYOPT(1) = 1, then the shell has no bending stiffness, a condition that can result in solver and convergence problems.

SHELL181 Product Restrictions

ANSYS Professional

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The only special features allowed are stress stiffening, large deflections, and plasticity (BISO, BKIN).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PLANE182

2-D 4-Node Structural Solid

MP ME ST PR PRN DS <> <> PP EME MFS
Product Restrictions

PLANE182 Element Description

PLANE182 is used for 2-D modeling of solid structures. The element can be used as either a plane element (plane stress, plane strain or generalized plane strain) or an axisymmetric element. It is defined by four nodes having two degrees of freedom at each node: translations in the nodal x and y directions. The element has plasticity, hyperelasticity, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials.

See PLANE182 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 182.1: PLANE182 Geometry

PLANE182 Input Data

The geometry and node locations for this element are shown in Figure 182.1: PLANE182 Geometry (p. 821). The element input data includes four nodes, a thickness (for the plane stress option only), and the orthotropic material properties. The default element coordinate system is along global directions. You may define an element coordinate system using **ESYS**, which forms the basis for orthotropic material directions.

Element loads are described in Nodal Loading (p. 49). Pressures may be input as surface loads on the element faces as shown by the circled numbers on Figure 182.1: PLANE182 Geometry (p. 821). Positive pressures act into the element. Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

Input the nodal forces, if any, per unit of depth for a plane analysis (except for KEYOPT(3) = 3 or KEYOPT(3) = 5) and on a full 360° basis for an axisymmetric analysis.

KEYOPT(3) = 5 is used to enable generalized plane strain. For more information about the generalized plane strain option, see Generalized Plane Strain (p. 68) in the *Element Reference*.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(6) = 1 sets the element for using mixed formulation. For details on the use of mixed formulation, see Applications of Mixed u-P Formulations (p. 73) in the *Element Reference*.

You can apply an initial stress state to this element via the **INISTATE** command. For more information, see Initial State in the *Basic Analysis Guide*.

As described in Coordinate Systems (p. 55), you can use **ESYS** to orient the material properties and strain/stress output. Use **RSYS** to choose output that follows the material coordinate system or the global coordinate system. For the case of hyperelastic materials, the output of stress and strain is always with respect to the global Cartesian coordinate system rather than following the material/element coordinate system.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

"PLANE182 Input Summary" (p. 822) contains a summary of the element input. For a general description of element input, see Element Input (p. 43). For axisymmetric applications see Harmonic Axisymmetric Elements (p. 83).

PLANE182 Input Summary

Nodes

I, J, K, L

Degrees of Freedom

UX, UY

Real Constants

THK - Thickness (used only if KEYOPT(3) = 3)
HGSTF - Hourglass stiffness scaling factor (used only if KEYOPT(1) = 1); default is 1.0 (if you input 0.0, the default value is used)

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ),
DENS, GXY, GYZ, GXZ, ALPD, BETD

Surface Loads

Pressures --

face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads

Temperatures --

T(I), T(J), T(K), T(L)

Body force densities --

The element values in the global X and Y directions.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Special Features --

Birth and death
Element technology autoselect
Initial state
Large deflection
Large strain
Linear perturbation
Material force evaluation
Nonlinear stabilization
Rezoning
Stress stiffening

KEYOPT(1)
Element technology:

0 --
Full integration with $\bar{B}$ method

1 --
Uniform reduced integration with hourglass control

2 --
Enhanced strain formulation

3 --
Simplified enhanced strain formulation

KEYOPT(3)
Element behavior:

0 --
Plane stress

1 --
Axisymmetric

2 --
Plane strain (Z strain = 0.0)

3 --
Plane stress with thickness input

5 --
Generalized plane strain

KEYOPT(6)
Element formulation:

0 --
Use pure displacement formulation (default)

1 --
Use mixed u-P formulation (not valid with plane stress)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PLANE182 Element Technology

PLANE182 uses the full-integration $\bar{B}$ method (also known as the selective reduced integration method), enhanced strain formulation, simplified enhanced strain formulation, or uniform reduced integration.

When enhanced strain formulation (KEYOPT(1) = 2) is selected, the element introduces four *internal* (user-inaccessible) degrees of freedom to handle shear locking, and one internal degree of freedom to handle volumetric locking.

For more information, see Element Technologies.

PLANE182 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 182.1: PLANE182 Element Output Definitions (p. 824)

Several items are illustrated in Figure 182.2: PLANE182 Stress Output (p. 824).

The element stress directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

Figure 182.2: PLANE182 Stress Output

Stress directions are shown for Global.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 182.1: PLANE182 Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
NODES	Nodes - I, J, K, L	-	Y
MAT	Material number	-	Y
THICK	Thickness	-	Y
VOLU:	Volume	-	Y
XC, YC	Location where results are reported	Y	3
PRES	Pressures P1 at nodes J,I; P2 at K,J; P3 at L,K; P4 at I,L	-	Y
TEMP	Temperatures T(I), T(J), T(K), T(L)	-	Y
S:X, Y, Z, XY	Stresses (SZ = 0.0 for plane stress elements)	Y	Y
S:1, 2, 3	Principal stresses	-	Y
S:INT	Stress intensity	-	Y
S:EQV	Equivalent stress	Y	Y
EPEL:X, Y, Z, XY	Elastic strains	Y	Y
EPEL:EQV	Equivalent elastic strain [6]	Y	Y
EPTH:X, Y, Z, XY	Thermal strains	2	2
EPTH:EQV	Equivalent thermal strain [6]	2	2
EPPL:X, Y, Z, XY	Plastic strains[7]	1	1
EPPL:EQV	Equivalent plastic strain [6]	1	1
EPCR:X, Y, Z, XY	Creep strains	1	1
EPCR:EQV	Equivalent creep strains [6]	1	1
EPTO:X, Y, Z, XY	Total mechanical strains (EPEL + EPPL + EPCR)	Y	-
EPTO:EQV	Total equivalent mechanical strains (EPEL + EPPL + EPCR)	Y	-
NL:SEPL	Plastic yield stress	1	1
NL:EPEQ	Accumulated equivalent plastic strain	1	1
NL:CREQ	Accumulated equivalent plastic strain	1	1
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	1	1
NL:PLWK	Plastic work/volume	1	1
NL:HPRES	Hydrostatic pressure	1	1
SEND:ELASTIC, PLASTIC, CREEP	Strain energy densities	-	1
LOCI:X, Y, Z	Integration point locations	-	4
SVAR:1, 2, ... , N	State variables	-	5

1. Nonlinear solution, output only if the element has a nonlinear material, or if large-deflection effects are enabled (**NLGEOM**,ON) for SEND.

2. Output only if element has a thermal load.

3. Available only at centroid as a ***GET** item.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Available only if **OUTRES**,LOCI is used.

5. Available only if the `UserMat` subroutine and **TB**,STATE command are used.

6. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.

7. For the shape memory alloy material model, transformation strains are reported as plasticity strain EPPL.

For axisymmetric solutions in a global coordinate system, the X, Y, Z, and XY stress and strain outputs correspond to the radial, axial, hoop, and in-plane shear stresses and strains, respectively.

Table 182.2: PLANE182 Item and Sequence Numbers (p. 826) lists output available through **ETABLE** using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 182.2: PLANE182 Item and Sequence Numbers (p. 826):

Name
output quantity as defined in the Table 182.1: PLANE182 Element Output Definitions (p. 824)

Item
predetermined Item label for **ETABLE**

E
sequence number for single-valued or constant element data

I,J,K,L
sequence number for data at nodes I, J, K, L

Table 182.2: PLANE182 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
P1	SMISC	-	2	1	-	-
P2	SMISC	-	-	4	3	-
P3	SMISC	-	-	-	6	5
P4	SMISC	-	7	-	-	8
THICK	NMISC	1	-	-	-	-

PLANE182 Assumptions and Restrictions

- The area of the element must be nonzero.
- The element must lie in a global X-Y plane as shown in Figure 182.1: PLANE182 Geometry (p. 821) and the Y-axis must be the axis of symmetry for axisymmetric analyses. An axisymmetric structure should be modeled in the +X quadrants.
- You can form a triangular element by defining duplicate K and L node numbers (see Degenerated Shape Elements (p. 39)). For triangular elements where the $\bar{B}$ or enhanced strain formulations are specified, degenerated shape functions and a conventional integration scheme are used.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you use the mixed formulation (KEYOPT(6) = 1), you must use the sparse solver.
- For modal cyclic symmetry analyses, ANSYS, Inc. recommends using enhanced strain formulation.
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). Prestress effects can be activated by the **PSTRES** command.

PLANE182 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

PLANE183

2-D 8-Node or 6-Node Structural Solid

MP ME ST PR PRN DS <> <> PP EME MFS
Product Restrictions

PLANE183 Element Description

PLANE183 is a higher order 2-D, 8-node or 6-node element. PLANE183 has quadratic displacement behavior and is well suited to modeling irregular meshes (such as those produced by various CAD/CAM systems).

This element is defined by 8 nodes or 6 nodes having two degrees of freedom at each node: translations in the nodal x and y directions. The element may be used as a plane element (plane stress, plane strain and generalized plane strain) or as an axisymmetric element. This element has plasticity, hyperelasticity, creep, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials. Initial state is supported. Various printout options are also available. See PLANE183 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 183.1: PLANE183 Geometry

PLANE183 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 183.1: PLANE183 Geometry (p. 829).

Although a degenerated triangular-shaped element may be formed by defining the same node number for nodes K, L and O when KEYOPT(1) = 1, it is better to use KEYOPT(1) = 1 for triangular shaped elements. In addition to the nodes, the element input data includes a thickness (TK) (for the plane stress option only) and the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is described in Coordinate Systems (p. 55).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Element loads are described in Nodal Loading (p. 49). Pressures may be input as surface loads on the element faces as shown by the circled numbers in Figure 183.1: PLANE183 Geometry (p. 829). Positive pressures act into the element. Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). If all corner node temperatures are specified, each midside node temperature defaults to the average temperature of its adjacent corner nodes. For any other input temperature pattern, unspecified temperatures default to TUNIF.

The nodal forces, if any, should be input per unit of depth for a plane analysis (except for KEYOPT(3) = 3 or KEYOPT(3) = 5) and on a full 360° basis for an axisymmetric analysis.

As described in Coordinate Systems (p. 55), you can use **ESYS** to orient the material properties and strain/stress output. Use **ESYS** to choose output that follows the material coordinate system or the global coordinate system. For the case of hyperelastic materials, the output of stress and strain is always with respect to the global Cartesian coordinate system rather than following the material/element coordinate system.

KEYOPT(3) = 5 is used to enable generalized plane strain. For more information about the generalized plane strain option, see Generalized Plane Strain (p. 68) in the *Element Reference*.

KEYOPT(6) = 1 sets the element for using mixed formulation. For details on the use of mixed formulation, see Applications of Mixed u-P Formulations (p. 73) in the *Element Reference*.

You can apply an initial stress state to this element via the **INISTATE** command. For more information, see Initial State in the *Basic Analysis Guide*.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

The next table summarizes the element input. Element Input (p. 43) gives a general description of element input. For axisymmetric applications see Harmonic Axisymmetric Elements (p. 83).

PLANE183 Input Summary

Nodes

I, J, K, L, M, N, O, P when KEYOPT(1) = 0

I, J, K, L, M, N when KEYOPT(1) = 1)

Degrees of Freedom

UX, UY

Real Constants

None, if KEYOPT (3) = 0, 1, or 2
THK - Thickness if KEYOPT (3) = 3

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ),
DENS, GXY, GYZ, GXZ, ALPD, BETD

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface Loads

Pressures --
face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L) when KEYOPT(1) = 0

face 1 (J-I), face 2 (K-J), face 3 (I-K) when KEYOPT(1) = 1

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P) when KEYOPT(1) = 0

T(I), T(J), T(K), T(L), T(M), T(N) when KEYOPT(1) = 1

Body force densities --
The element values in the global X and Y directions.

Special Features

Birth and death
Element technology autoselect
Initial state
Large deflection
Large strain
Linear perturbation
Material force evaluation
Nonlinear stabilization
Rezoning
Stress stiffening

KEYOPT(1)
Element shape:

0 --
8-node quadrilateral

1 --
6-node triangle

KEYOPT(3)
Element behavior:

0 --
Plane stress

1 --
Axisymmetric

2 --
Plane strain (Z strain = 0.0)

3 --
Plane stress with thickness (TK) real constant input

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5 --
Generalized plane strain

KEYOPT(6)
Element formulation:

0 --
Use pure displacement formulation (default)

1 --
Use mixed u-P formulation (not valid with plane stress)

PLANE183 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 183.1: PLANE183 Element Output Definitions (p. 832).

Several items are illustrated in Figure 183.2: PLANE183 Stress Output (p. 832).

The element stress directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

Figure 183.2: PLANE183 Stress Output

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 183.1: PLANE183 Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Nodes - I, J, K, L (for KEYOPT(1) = 0 and I, J, K (for KEYOPT(1) = 1)	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
MAT	Material number	-	Y
THICK	Thickness	-	Y
VOLU:	Volume	-	Y
XC, YC	Location where results are reported	Y	4
PRES	Pressures P1 at nodes J, I; P2 at K, J; P3 at L, K; P4 at I, L (P4 only for KEYOPT(1) = 0	-	Y
TEMP	Temperatures T(I), T(J), T(K), T(L) (T(L) only when KEYOPT(1) = 0)	-	Y
S:X, Y, Z, XY	Stresses (SZ = 0.0 for plane stress elements)	Y	Y
S:1, 2, 3	Principal stresses	-	Y
S: INT	Stress intensity	-	Y
S:EQV	Equivalent stress	-	Y
EPEL:X, Y, Z, XY	Elastic strains	Y	Y
EPEL:EQV	Equivalent elastic strain [7]	-	Y
EPTH:X, Y, Z, XY	Thermal strains	3	3
EPTH:EQV	Equivalent thermal strain [7]	-	3
EPPL:X, Y, Z, XY	Plastic strains[8]	1	1
EPPL:EQV	Equivalent plastic strain [7]	-	1
EPCR:X, Y, Z, XY	Creep strains	2	2
EPCR:EQV	Equivalent creep strains [7]	2	2
EPTO:X, Y, Z, XY	Total mechanical strains (EPEL + EPPL + EPCR)	Y	-
EPTO:EQV	Total equivalent mechanical strains (EPEL + EPPL + EPCR)	Y	-
NL:SEPL	Plastic yield stress	1	1
NL:EPEQ	Accumulated equivalent plastic strain	1	1
NL:CREQ	Accumulated equivalent creep strain	1	1
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	1	1
NL:PLWK	Plastic work/volume	1	1
NL:HPRES	Hydrostatic pressure	1	1
SEND:ELASTIC, PLASTIC, CREEP	Strain energy densities	-	1
LOCI:X, Y, Z	Integration point locations	-	5
SVAR:1, 2, ... , N	State variables	-	6

1. Nonlinear solution, output only if the element has a nonlinear material, or if large-deflection effects are enabled (**NLGEOM**,ON) for SEND.

2. Output only if element has a creep load.

3. Output only if element has a thermal load.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Available only at centroid as a ***GET** item.

5. Available only if **OUTRES**,LOCI is used.

6. Available only if the `UserMat` subroutine and **TB**,STATE command are used.

7. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.

8. For the shape memory alloy material model, transformation strains are reported as plasticity strain EPPL.

For axisymmetric solutions, the X, Y, XY, and Z stress and strain outputs correspond to the radial, axial, in-plane shear, and hoop stresses and strains.

Table 183.2: PLANE183 Item and Sequence Numbers (p. 834) lists output available through **ETABLE** using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 183.2: PLANE183 Item and Sequence Numbers (p. 834):

Name
output quantity as defined in Table 183.1: PLANE183 Element Output Definitions (p. 832)

Item
predetermined Item label for **ETABLE**

E
sequence number for single-valued or constant element data

I,J,...,P
sequence number for data at nodes I, J, ..., P

Table 183.2: PLANE183 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input									
	Item	E	I	J	K	L	M	N	O	P
P1	SMISC	-	2	1	-	-	-	-	-	-
P2	SMISC	-	-	4	3	-	-	-	-	-
P3	SMISC	-	-	-	6	5	-	-	-	-
P4[1]	SMISC	-	7	-	-	8	-	-	-	-
THICK	NMISC	1	-	-	-	-	-	-	-	-

1. P4 is only for KEYOPT(1) = 0

PLANE183 Assumptions and Restrictions

- The area of the element must be positive.

- The element must lie in a global X-Y plane as shown in Figure 183.1: PLANE183 Geometry (p. 829) and the Y-axis must be the axis of symmetry for axisymmetric analyses. An axisymmetric structure should be modeled in the +X quadrants.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- A face with a removed midside node implies that the displacement varies linearly, rather than parabolically, along that face. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.
- Use at least two elements to avoid hourglass mode for KEYOPT(1) = 0.
- A triangular element may be formed by defining duplicate K-L-O node numbers (see Degenerated Shape Elements (p. 39)). For these degenerated elements, the triangular shape function is used and the solution is the same as for the regular triangular 6-node elements, but might be slightly less efficient for KEYOPT(1) = 0. Since these degenerated elements are less efficient, the triangle shape option (KEYOPT(1) = 1) is suggested for this case.
- When mixed formulation is used (KEYOPT(6) = 1), no midside nodes can be missed. If you use the mixed formulation (KEYOPT(6) = 1), you must use the sparse solver (default).
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). Prestress effects can be activated by the **PSTRES** command.

PLANE183 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

MPC184

Multipoint Constraint Element

MP ME ST PR PRN <> <> <> PP EME MFS
Product Restrictions

MPC184 Element Description

MPC184 comprises a general class of multipoint constraint elements that apply kinematic constraints between nodes. The elements are loosely classified here as "constraint elements" (rigid link, rigid beam, etc.) and "joint elements" (revolute, universal, etc.). The constraint may be as simple as that of identical displacements between nodes. Constraints can also be more complicated, such as those modeling rigid parts, or those transmitting motion between flexible bodies in a particular way. For example, a structure may consist of rigid parts and moving parts connected together by rotational or sliding connections. The rigid part of the structure may be modeled with the MPC184 link/beam elements, while the moving parts may be connected with any of the MPC184 joint elements.

The kinematic constraints are imposed using one of the following two methods:

- The **direct elimination method**, wherein the kinematic constraints are imposed by internally generated constraint equations. The degrees of freedom of a dependent node in the equations are eliminated in favor of an independent node.
  - The dependent degrees of freedom are eliminated. Therefore, the constraint forces and moments are not available from the element output table (**ETABLE**) for output purposes. However, the global constraint reaction forces are available at independent nodes in the results file, Jobname.rst (**PRRSOL** command, etc.).
  - The direct elimination method should be used whenever it is available since the degrees of freedom at the dependent nodes are eliminated, thereby reducing the problem size and solution time.
- The **Lagrange multiplier method**, wherein the kinematic constraints are imposed using Lagrange multipliers. In this case, all the participating degrees of freedom are retained.
  - The Lagrange multiplier method should be used when the direct elimination method is not available or not suitable for the analysis purposes.
  - In this method, the constraint forces and moments are available from the element output table (**ETABLE**).
  - The disadvantage of the Lagrange multiplier method is that the Lagrange multipliers are additional solution variables and, hence, the problem size and solution time become larger when compared with the direct elimination method.

Currently, the MPC184 rigid link/beam elements can use the direct elimination method or the Lagrange multiplier method. All other MPC184 element options use the Lagrange multiplier method only.

Constraint Elements

The following types of constraint elements are available:

Rigid link/beam
Slider

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Joint Elements

Numerical simulations often involve modeling of joints between two parts. These joints or connections may need simple kinematic constraints such as identical displacements between the two parts at the junction or more complicated kinematic constraints that allow for transmission of motion between two flexible bodies. These complex joints may also include some sort of control mechanism like limits or stops, and locks on the components of relative motion between the two bodies. In many instances, these joints may also have stiffness, damping, or friction forces based on the unconstrained components of relative motion between the two bodies. For detailed information on how to use joint elements, see Connecting Multibody Components with Joint Elements in the *Multibody Analysis Guide*.

The following types of joint elements are available:

x-axis Revolute joint
z-axis Revolute joint
Universal joint
Slot joint
Point-in-plane joint
Translational joint
x-axis Cylindrical joint
z-axis Cylindrical joint
x-axis Planar joint
z-axis Planar joint
Weld joint
Orient joint
Spherical joint
General joint
Screw joint

These elements are well suited for linear, large rotation, and/or large strain nonlinear applications. If finite rotations and/or large strain effects are to be considered, the **NLGEOM**,ON command must be used; otherwise, linear behavior is assumed. For example, if a revolute joint element is used in an analysis and **NLGEOM**,ON is not set, the calculations are carried out in the original configuration and the end result may not reflect the expected deformed configuration. However, if the **NLGEOM**,ON command is used, the calculations will take into account the rotation of the revolute joint element.

Two nodes define these joint elements. Depending on the joint to be defined, the kinematic constraints are imposed on some of the quantities that define the relative motion between the two nodes. These kinematic constraints are applied using Lagrange multipliers. In some instances, one of the nodes is required to be "grounded" or attached to "ground" or some other reference location that is not moving. In such cases, only one of the two nodes may be specified. The specified node and the "grounded" node are assumed to be coincident in the element calculations.

The joint element has six degrees of freedom at each node, defining six components of relative motion: three relative displacements and three relative rotations. These six components of relative motion are of primary interest in simulations that involve joint elements. Some of these components may be constrained by the kinematic constraints relevant to a particular joint element, while the other components are "free" or "unconstrained". For example, in the case of universal and revolute joint elements the two nodes are assumed to be connected, and thus the relative displacements are zero. For the revolute joint only one rotational component of the relative motion (rotation about the revolute axis) is unconstrained, while for the universal joint two such components are available.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The capabilities of these elements include certain control features such as stops, locks, and actuating loads/boundary conditions that can be imposed on the components of relative motion between the two nodes of the element. For example, in a revolute joint, stops can be specified for the rotation about the revolute axis. This limits the rotation around the revolute axis to be within a certain range. Displacement, force, velocity, and acceleration boundary conditions may be imposed on the components of relative motion between the two nodes allowing for "actuation" of the joints. The driving force or displacements arise from the actuating mechanisms like an electric or hydraulic system that drives these joints.

You can impose linear and nonlinear elastic stiffness and damping behavior or hysteretic friction behavior on the available components of relative motion of a joint element. The properties can be made temperature dependent if necessary.

In addition to the existing output options available in ANSYS, outputs related to the components of relative motion are available for joint elements.

Joint Input Data

Certain input requirements are common to most MPC184 joint elements. Any specific requirements for individual joint elements are highlighted in the description for that element.

The following types of input data should be considered:

- Element Connectivity Definition - A joint element is typically defined by specifying two nodes, I and J. One of these nodes may be a "grounded" node.
- Section Definition - Each joint element must have an associated section definition (**SECTYPE** command).
- Local Coordinate System Specification - Local coordinate systems at the nodes are often required to define the kinematic constraints of a joint element (**SECJOINT** command).
- Stops or Limits - You can impose stops or limits on the available components of relative motion between the two nodes of a joint element (**SECSTOP** command).
- Locks - Locking limits may also be imposed on the available components of relative motion between the two nodes of a joint element to "freeze" the joint in a desired configuration (**SECLOCK** command).
- Material Behavior - The JOIN material option on the **TB** command allows you to impose linear and nonlinear elastic stiffness and damping behavior or hysteretic friction behavior on the available components of relative motion of a joint element.
- Reference Lengths and Angles - These correspond to the free relative degrees of freedom in a joint element for which constitutive calculations are performed and are used when stiffness, damping, or hysteretic friction are specified for the joint elements (**SECDATA** command).
- Boundary Conditions - You can impose boundary conditions (**DJ** command) or apply concentrated forces (**FJ** command) on the available components of relative motion of the joint element.

MPC184 Input Data

Use KEYOPT(1) to specify the type of MPC184 constraint or joint element you want to use. The remaining input data will vary depending on the type of constraint or joint element specified. The individual MPC184 element descriptions each contain an input summary that applies only to that particular element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

It is recommended that you review these element-specific input summaries after you determine which constraint or joint element you will be using.

KEYOPT(1)

Element behavior:

0 --
Rigid link (default)

1 --
Rigid beam

3 --
Slider element

6 --
x-axis or z-axis revolute joint element

7 --
Universal joint element

8 --
Slot joint element

9 --
Point-in-plane joint element

10 --
Translational joint element

11 --
x-axis or z-axis cylindrical joint element

12 --
x-axis or z-axis planar joint element

13 --
Weld joint element

14 --
Orient joint element

15 --
Spherical joint element

16 --
General joint element

17 --
Screw joint element

MPC184 Output Data

The solution output associated with the constraint and joint elements is in two forms:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Nodal displacements included in the overall nodal solution.
- Additional element output as shown in the individual constraint and joint element descriptions. This output is available via the **ETABLE** command using the Sequence Number method.

Refer to the individual element descriptions for complete listings of the output for each element.

MPC184 Assumptions and Restrictions

The following restrictions apply to all forms of the MPC184element:

- For MPC184, the element coordinate system (**/PSYMB**,ESYS) is not relevant.

There are additional assumptions and restrictions for each type of constraint and joint element. For details, see the Assumptions and Restrictions section in the individual constraint and joint element descriptions.

MPC184 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The BETD material property is not allowed.
- No special features are allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

MPC184-Link/Beam

Multipoint Constraint Element: Rigid Link or Rigid Beam

MP ME ST PR PRN <> <> <> PP EME MFS
Product Restrictions

MPC184 Rigid Link/Beam Element Description

The MPC184 rigid link/beam element can be used to model a rigid constraint between two deformable bodies or as a rigid component used to transmit forces and moments in engineering applications. This element is well suited for linear, large rotation, and/or large strain nonlinear applications.

The kinematic constraints are imposed using one of the following two methods:

- The **direct elimination method**, wherein the kinematic constraints are imposed by internally generated MPC (multipoint constraint) equations. The degrees of freedom of a dependent node in the MPC equations are eliminated in favor of an independent node.
- The **Lagrange multiplier method**, wherein the kinematic constraints are imposed using Lagrange multipliers. In this case, all the participating degrees of freedom are retained.

Figure 184link.1: MPC184 Rigid Link/Beam Geometry

MPC184 Rigid Link/Beam Input Data

Figure 184link.1: MPC184 Rigid Link/Beam Geometry (p. 843) shows the geometry, node locations, and the coordinate system for this element. Two nodes define the element. The element x-axis is oriented from node I toward node J. The cross-sectional area of the element is assumed to be one unit. ANSYS selects the cross-section coordinate system automatically. The cross-section coordinate system is relevant only for the output of bending moments when the element is used as a rigid beam.

If KEYOPT(1) = 0 (default), the element is a rigid link with two nodes and three degrees of freedom at each node (UX, UY, UZ). If KEYOPT(1) = 1, the element is a rigid beam with two nodes and six degrees of freedom at each node (UX, UY, UZ, ROTX, ROTY, ROTZ).

If KEYOPT(2) = 0 (default), then the constraints are implemented using the direct elimination method. If KEYOPT(2) = 1, then the Lagrange multiplier method is used to impose the constraints.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The MPC184 rigid link/beam element with KEYOPT(2) = 1 can also be used in applications that call for thermal expansion on an otherwise rigid structure. The direct elimination method cannot be used for thermal expansion problems.

Because the element models a rigid constraint or a rigid component, material stiffness properties are not required. When thermal expansion effects are desired, the coefficient of thermal expansion must be specified. Density must be specified if the mass of the rigid element is to be accounted for in the analysis. If density is specified, ANSYS calculates a lumped mass matrix for the element.

The element supports the birth and death options using **EALIVE** and **EKILL**.

Nodal Loading (p. 49) describes element loads. You can input temperatures as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. The node J temperature defaults to T(I).

MPC184 Rigid Link/Beam Input Summary

This input summary applies to the rigid link and rigid beam options of MPC184: KEYOPT(1) = 0 and 1.

Nodes
I, J

Degrees of Freedom
UX, UY, UZ if KEYOPT(1) = 0

UX, UY, UZ, ROTX, ROTY, ROTZ if KEYOPT(1) = 1

Real Constants
None

Material Properties
ALPX (or CTEX *or* THSX), DENS

Surface Loads
None

Body Loads

Temperatures --
T(I), T(J)

Element Loads
None

Special Features

Birth and death
Large deflection
Linear perturbation

KEYOPT(1)
Element behavior:

0 --
Rigid link (default)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
Rigid beam

KEYOPT(2)
Reduction method:

0 --
Direct elimination method (default)

1 --
Lagrange multiplier method

MPC184 Rigid Link/Beam Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 184link.1: MPC184 Rigid Link/Beam Element Output Definitions (p. 845).

Table 184link.1: MPC184 Rigid Link/Beam Element Output Definitions (p. 845) uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 184link.1: MPC184 Rigid Link/Beam Element Output Definitions

Name	Definition	O	R
Link/Beam Elements (KEYOPT(1) = 0 or 1, and KEYOPT(2) = 0 or 1)			
EL	Element number	-	Y
NODES	Element node numbers (I and J)	-	Y
Link/Beam Elements (KEYOPT(1) = 0 or 1, and KEYOPT(2) = 1)			
MAT	Material number for the element	-	Y
TEMP	Temperature at nodes I and J	-	Y
FX	Axial force	-	Y
MY, MZ	Bending moments	-	Y
SF:Y, Z	Section shear forces	-	Y
MX	Torsional moment	-	Y

Table 184link.2: MPC184 Rigid Link/Beam Item and Sequence Numbers (p. 846) lists output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) for further information. The table uses the following notation:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name
output quantity as defined in the Element Output Definitions table.

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 184link.2: MPC184 Rigid Link/Beam Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
Link/Beam Constraint (with KEYOPT(2) = 1)		
FX	SMISC	1
MY	SMISC	2
MZ	SMISC	3
MX	SMISC	4
SFZ	SMISC	5
SFY	SMISC	6

MPC184 Rigid Link/Beam Assumptions and Restrictions

The following restrictions apply to both the direct elimination method and the Lagrange multiplier method (KEYOPT(2) = 0 and 1):

- A finite element model cannot be made up of only rigid elements in a static analysis. At a minimum, a deformable element (or elements) must be connected to one of the end nodes of a rigid element.
- The cross-sectional area of the element is assumed to be unity.
- The element coordinate system (**/PSYMB**,ESYS) is not relevant.

Direct Elimination Method (KEYOPT(2) = 0)

These additional restrictions apply to the direct elimination method:

- The MPC184 rigid link/beam using the direct elimination method can be used in static, transient, modal, and buckling analyses.
- This element cannot be used with Distributed ANSYS when the direct elimination method is used.
- Displacement boundary conditions on the nodes of rigid link/beams must be applied prudently. In a rigid linkage (structure) made of a number of rigid link/beam elements, if displacement boundary conditions are applied at more than one location, ANSYS will use the first encountered displacement boundary condition to constrain the entire rigid linkage according to rigid kinematic conditions. In some cases where the applied displacements may be redundant or self-contradictory, ANSYS will issue warning or error messages.
- The direct elimination method cannot be used in problems involving thermal expansion. Use the Lagrange Multiplier method instead.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Reaction forces at the constrained nodes of a rigid link/beam may not always be available since the dependent and independent nodes are determined by ANSYS internally. We recommend that you check the interface nodes which connect rigid and deformable elements since reaction forces are available on these nodes.
- The nodes of a rigid link/beam using the direct elimination method should not be linked with a node of an element implemented via the Lagrange multiplier method. For example, a rigid beam implemented using the direct elimination method (KEYOPT(2) = 0) should not be linked to a rigid beam implemented via the Lagrange multiplier method (KEYOPT(2) = 1). Or, a rigid beam implemented via the direct elimination method should not be linked to a node of a contact element that is implemented via the Lagrange multiplier method (KEYOPT(2) = 2 on the contact element).
- Coupling constraints (**CP** command) cannot be applied to nodes of rigid links/beams using the direct elimination method.
- Nodes of rigid links/beams cannot be part of the retained nodes (nodes specified by the **M** command) in a substructure. However, the rigid links/beams can be entirely within the substructure.
- Rigid links/beams should be not used in cyclic symmetry analyses.

Lagrange Multiplier Method (KEYOPT(2) = 1)

These additional restrictions apply to the Lagrange Multiplier method:

- To employ this feature successfully, use as few of these elements as possible. For example, it may be sufficient to overlay rigid line elements on a perimeter of a rigid region modeled with shell elements, as opposed to overlaying rigid line elements along each element boundary of the interior.
- Modeling that avoids overconstraining the problem is necessary. Overconstrained models may result in trivial solutions, zero pivot messages (in a properly restrained system), or nonlinear convergence difficulties.
- The temperature is assumed to vary linearly along the spar of the rigid link or rigid beam element.
- If constraint equations are specified for the DOFs of a rigid element, it may be an overconstrained system. Similarly, prescribed displacements on both ends of the element is an indication of overconstraint.
- When used as a link element, exercise the same precautions that you would when using a truss element (for example, LINK180).
- The equation solver (**EQSLV**) must be the sparse solver or the PCG solver. The command **PCGOPT**,,,,,,ON is also required in order to use the PCG solver.
- The element is valid for static and transient analyses (linear and nonlinear), and rigid beam is valid for harmonic analyses. The element is not supported for buckling analyses.

MPC184 Rigid Link/Beam Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- No special features are allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

MPC184-Slider

Multipoint Constraint Element: Slider

MP ME ST PR PRN <> <> <> PP EME MFS
Product Restrictions

MPC184 Slider Element Description

The MPC184 slider element can be used to model a slider constraint. The element has three nodes with three degrees of freedom (translations in X, Y, and Z) at each node. The slider element imposes a kinematic constraint such that a "dependent" node (I) must always lie on a line joining two other "independent" nodes (J and K). The I node is allowed to slide on the line joining J and K nodes. The kinematic constraints are imposed using the Lagrange multiplier method.

Figure 184slid.1: MPC184 Slider Geometry

MPC184 Slider Input Data

Set KEYOPT(1) = 3 to define a three-node slider element.

Figure 184slid.1: MPC184 Slider Geometry (p. 849) shows the geometry and node locations for this element. Three nodes (I, J, and K) define the element. The node I is expected to lie initially on the line joining the nodes J and K.

Material stiffness properties are not required for this element. The element currently does not support birth or death options.

MPC184 Slider Input Summary

This input summary applies to the slider element option of MPC184: KEYOPT(1) = 3.

Nodes
I, J, K

Degrees of Freedom
UX, UY, UZ

Real Constants
None

Material Properties
None

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface Loads
None

Body Loads
None

Element Loads
None

Special Features

Large deflection
Linear perturbation

KEYOPT(1)
Element behavior:

3 --
Slider element

MPC184 Slider Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 184slid.1: MPC184 Slider Element Output Definitions (p. 850).

Table 184slid.1: MPC184 Slider Element Output Definitions (p. 850) uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 184slid.1: MPC184 Slider Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
NODES	Element node numbers (I, J, K)	-	Y
FY	Constraint Force 1	-	Y
FZ	Constraint Force 2	-	Y

Table 184slid.2: MPC184 Slider Item and Sequence Numbers (p. 851) lists output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) for further information. The table uses the following notation:

Name
output quantity as defined in the Element Output Definitions table.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Item

predetermined Item label for **ETABLE** command

E

sequence number for single-valued or constant element data

Table 184slid.2: MPC184 Slider Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FY	SMISC	1
FZ	SMISC	2

MPC184 Slider Assumptions and Restrictions

- The distance between nodes I and J must be greater than zero.
- Node I must initially lie between the J and K nodes.
- Displacement boundary conditions cannot be applied on the nodes forming the slider element.
- The element currently does not support birth or death options.
- The equation solver (**EQSLV**) must be the sparse solver or the PCG solver. The command **PCGOPT**,,,,,,ON is also required in order to use the PCG solver.
- The element coordinate system (**/PSYMB**,ESYS) is not relevant.

MPC184 Slider Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- No special features are allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

MPC184-Revolute

Multipoint Constraint Element: Revolute Joint

MP ME ST PR PRN <> <> <> PP EME MFS
Product Restrictions

MPC184 Revolute Joint Element Description

The MPC184 revolute joint is a two-node element that has only one primary degree of freedom, the relative rotation about the revolute (or hinge) axis. This element imposes kinematic constraints such that the nodes forming the element have the same displacements. Additionally, only a relative rotation is allowed about the revolute axis, while the rotations about the other two directions are fixed.

Figure 184revo.1: MPC184 Revolute Joint Geometry

MPC184 Revolute Joint Input Data

Set KEYOPT(1) = 6 to define a two-node revolute joint element.

Figure 184revo.1: MPC184 Revolute Joint Geometry (p. 853) shows the geometry and node locations for this element. Two nodes (I and J) define the element. The two nodes are expected to have identical spatial coordinates initially.

If KEYOPT(4) = 0, then element is an x-axis revolute joint with the local e_1 axis as the revolute axis.

If KEYOPT(4) = 1, then element is a z-axis revolute joint with the local e_3 axis as the revolute axis.

A local Cartesian coordinate system must be specified at the first node, I, of the element. The specification of the second local coordinate system at node J is optional. If the local coordinate system is not specified at node J, then the local coordinate system at node J is assumed to be the same as that at node I.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Either the local e_1 or local e_3 direction may be specified as the axis of rotation at the nodes. The specification of the other two local directions is not critical, but it will be used to determine the relative rotation between the two nodes during the course of deformation. The orientation of local directions must follow the convention specified in Figure 184revo.1: MPC184 Revolute Joint Geometry (p. 853). These local coordinate systems evolve with the rotations at the respective nodes (if any). Use the **SECJOINT** command to specify the identifiers of the local coordinate systems.

The constraints imposed in a revolute joint element with the local e_1 axis as the revolute axis are described below. Similar constraint conditions are set up when the local e_3 axis is the revolute axis.

Consider the two local coordinate systems (Cartesian) attached to node I and node J (see Figure 184revo.1: MPC184 Revolute Joint Geometry (p. 853)). At any given instant of time, the constraints imposed in a revolute joint are as described below.

Displacement constraints:

$$u^I = u^J$$

Where, u^I is the displacement vector at node I and u^J is the displacement vector at node J.

Rotation constraints:

$$e_1^I \cdot e_2^J = 0$$

$$e_1^I \cdot e_3^J = 0$$

If the revolute axes e_1^I and e_1^J are not aligned at the start of the analysis, then the angle between the two is held fixed at the starting value.

The relative position of the local coordinate system at node I with respect to node J is characterized by the first Cardan (or Bryant) angle given by:

$$\phi = -\tan^{-1}\left(\frac{e_2^I \cdot e_3^J}{e_3^I \cdot e_3^J}\right)$$

The change in the relative angular position between the two local coordinate system is given by:

$$u_r = \phi - \phi_0 + m\pi$$

Where, φ_0 is the initial angular offset (the first Cardan (or Bryant) angle measured in the reference configuration) between the two coordinate systems and m is an integer accounting for multiple rotations about the revolute axis.

The constitutive calculations use the following definition of the joint rotation:

$$u_r^c = \phi + m\pi - \phi_1^{ref}$$

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Where ϕ_1^{ref} is the reference angle, angle1, specified on the **SECDATA** command. If this value is not specified, then φ_0 is used in place of ϕ_1^{ref}.

Other input data that are common to all joint elements (material behavior, stops and limits, locks, etc.) are described in "Joint Input Data" (p. 839) in the MPC184 element description.

MPC184 Revolute Joint Input Summary

This input summary applies to the revolute joint element option of MPC184: KEYOPT(1) = 6.

Nodes

I, J

Note

For a grounded joint element, specify either node I or node J in the element definition and leave the other node (the grounded node) blank.

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants

None

Material Properties

Use the JOIN label on the **TB** command to define stiffness, damping, and Coulomb friction behavior. (See MPC184 Joint in the *Material Reference* for detailed information on defining joint materials.)

Surface Loads

None

Body Loads

Temperatures --

T(I), T(J)

Element Loads

Rotations --

ROTX (KEYOPT(4) = 0)
ROTZ (KEYOPT(4) = 1)

Angular velocities --

OMGX (KEYOPT(4) = 0)
OMGZ (KEYOPT(4) = 1)

Angular accelerations --

DMGX (KEYOPT(4) = 0)
DMGZ (KEYOPT(4) = 1)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Moments --

MX (KEYOPT(4) = 0)
MZ (KEYOPT(4) = 1)

Special Features

Large deflection
Linear perturbation

KEYOPT(1)
Element behavior:

6 --
Revolute joint element

KEYOPT(4)
Element configuration:

0 --
x-axis revolute joint with local 1 direction as the revolute axis.

1 --
z-axis revolute joint with local 3 direction as the revolute axis.

MPC184 Revolute Joint Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 184revo.1: MPC184 Revolute Joint Element Output Definitions (p. 856) and Table 184revo.2: MPC184 Revolute Joint Element - NMISC Output (p. 858).

These tables use the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 184revo.1: MPC184 Revolute Joint Element Output Definitions

Name	Definition	O	R
x-axis Revolute Joint (KEYOPT(4) = 0)			
EL	Element Number	-	Y
NODES	Element node numbers (I, J)	-	Y
FX	Constraint Force in X direction	-	Y
FY	Constraint Force in Y direction	-	Y
FZ	Constraint Force in Z direction	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
MY	Constraint Moment in Y direction	-	Y
MZ	Constraint Moment in Z direction	-	Y
CSTOP4	Constraint moment if stop is specified on DOF 4	-	Y
CLOCK4	Constraint moment if lock is specified on DOF 4	-	Y
CSST4	Constraint stop status[1]	-	Y
CLST4	Constraint lock status[2]	-	Y
JRP4	Joint relative position	-	Y
JCD4	Joint constitutive rotation	-	Y
JEF4	Joint elastic moment	-	Y
JDF4	Joint damping moment	-	Y
JFF4	Joint friction moment	-	Y
JRU4	Joint relative rotation	-	Y
JRV4	Joint relative velocity	-	Y
JRA4	Joint relative acceleration	-	Y
JTEMP	Average temperature in the element[3]	-	Y
JFST4	Stick/slip status when friction is specified[4]	-	Y
JFNF4	Normal moment in friction calculations	-	Y
z-axis Revolute Joint (KEYOPT(4) = 1)			
EL	Element Number	-	Y
NODES	Element node numbers (I, J)	-	Y
FX	Constraint Force in X direction	-	Y
FY	Constraint Force in Y direction	-	Y
FZ	Constraint Force in Z direction	-	Y
MX	Constraint Moment in X direction	-	Y
MY	Constraint Moment in Y direction	-	Y
CSTOP6	Constraint moment if stop is specified on DOF 6	-	Y
CLOCK6	Constraint moment if lock is specified on DOF 6	-	Y
CSST6	Constraint stop status[1]	-	Y
CLST6	Constraint lock status[2]	-	Y
JRP6	Joint relative position	-	Y
JCD6	Joint constitutive rotation	-	Y
JEF6	Joint elastic moment	-	Y
JDF6	Joint damping moment	-	Y
JFF6	Joint friction moment	-	Y
JRU6	Joint relative rotation	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
JRV6	Joint relative velocity	-	Y
JRA6	Joint relative acceleration	-	Y
JTEMP	Average temperature in the element[3]	-	Y
JFST6	Slip/stick status when friction is specified[4]	-	Y
JFNF6	Normal moment in friction calculations	-	Y

1. Constraint stop status:

   0 = stop not active, or deactivated
   1 = stopped at minimum value
   2 = stopped at maximum value

2. Constraint lock status:

   0 = lock not active
   1 = locked at minimum value
   2 = locked at maximum value

3. Average temperature in the element when temperatures are applied on the nodes of the element using the **BF** command, or when temperature are applied on the element using the **BFE** command.

4. Stick/slip status when friction is active:

   0 = friction is not activated
   1 = sticking
   2 = slipping or sliding

The following table shows additional non-summable miscellaneous (NMISC) output available for all forms of the revolute joint element.

Note

This output is intended for use in the ANSYS Workbench program to track the evolution of local coordinate systems specified at the nodes of joint elements.

Table 184revo.2: MPC184 Revolute Joint Element - NMISC Output

Name	Definition	O	R
The following output is available for all revolute joint elements (KEYOPT(4) = 0 and 1)			
E1X-I, E1Y-I, E1Z-I	X, Y, Z components of the evolved e_1 axis at node I	-	Y
E2X-I, E2Y-I, E2Z-I	X, Y, Z components of the evolved e_2 axis at node I	-	Y
E3X-I, E3Y-I, E3Z-I	X, Y, Z components of the evolved e_3 axis at node I	-	Y
E1X-J, E1Y-J, E1Z-J	X, Y, Z components of the evolved e_1 axis at node J	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
E2X-J, E2Y-J, E2Z-J	X, Y, Z components of the evolved e_2 axis at node J	-	Y
E3X-J, E3Y-J, E3Z-J	X, Y, Z components of the evolved e_3 axis at node J	-	Y
JFX, JFY, JFZ	Constraint forces expressed in the evolved co-ordinate system specified at node I	-	Y
JMX, JMY, JMZ	Constraint moments expressed in the evolved coordinate system specified at node I	-	Y

Table 184revo.3: MPC184 Revolute Joint Item and Sequence Numbers - SMISC Items (p. 859) and Table 184revo.4: MPC184 Revolute Joint Item and Sequence Numbers - NMISC Items (p. 860) list output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) for further information. The table uses the following notation:

Name
output quantity as defined in the Element Output Definitions table.

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 184revo.3: MPC184 Revolute Joint Item and Sequence Numbers - SMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
x-axis Revolute Joint (KEYOPT(4) = 0)		
FX	SMISC	1
FY	SMISC	2
FZ	SMISC	3
MY	SMISC	5
MZ	SMISC	6
CSTOP4	SMISC	10
CLOCK4	SMISC	16
CSST4	SMISC	22
CLST4	SMISC	28
JRP4	SMISC	34
JCD4	SMISC	40
JEF4	SMISC	46
JDF4	SMISC	52
JFF4	SMISC	58
JRU4	SMISC	64
JRV4	SMISC	70

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
JRA4	SMISC	76
JTEMP	SMISC	79
JFST4	SMISC	80
JFNF4	SMISC	84
z-axis Revolute Joint (KEYOPT(4) = 1)		
FX	SMISC	1
FY	SMISC	2
FZ	SMISC	3
MX	SMISC	4
MY	SMISC	5
CSTOP6	SMISC	12
CLOCK6	SMISC	18
CSST6	SMISC	24
CLST6	SMISC	30
JRP6	SMISC	36
JCD6	SMISC	42
JEF6	SMISC	48
JDF6	SMISC	54
JFF6	SMISC	60
JRU6	SMISC	66
JRV6	SMISC	72
JRA6	SMISC	78
JTEMP	SMISC	79
JFST6	SMISC	82
JFNF6	SMISC	86

Table 184revo.4: MPC184 Revolute Joint Item and Sequence Numbers - NMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
The following output is available for all revolute joint elements (KEYOPT(4) = 0 and 1)		
E1X-I	NMISC	1
E1Y-I	NMISC	2
E1Z-I	NMISC	3
E2X-I	NMISC	4
E2Y-I	NMISC	5
E2Z-I	NMISC	6
E3X-I	NMISC	7

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
E3Y-I	NMISC	8
E3Z-I	NMISC	9
E1X-J	NMISC	10
E1Y-J	NMISC	11
E1Z-J	NMISC	12
E2X-J	NMISC	13
E2Y-J	NMISC	14
E2Z-J	NMISC	15
E3X-J	NMISC	16
E3Y-J	NMISC	17
E3Z-J	NMISC	18
JFX	NMISC	19
JFY	NMISC	20
JFZ	NMISC	21
JMX	NMISC	22
JMY	NMISC	23
JMZ	NMISC	24

MPC184 Revolute Joint Assumptions and Restrictions

- The nodes I and J must be coincident.
- The local coordinate systems at the nodes must be specified such that the revolute axis is well defined. Otherwise, it is possible that the rotational motion might not be what is expected.
- Boundary conditions cannot be applied on the nodes forming the revolute joint.
- Rotational degrees of freedom are activated at the nodes forming the element. When these elements are used in conjunction with solid elements, the rotational degrees of freedom must be suitably constrained. Since boundary conditions cannot be applied to the nodes of the Revolute Joint, a beam or shell element with very weak stiffness may be used with the underlying solid elements at the nodes forming the joint element to avoid any rigid body modes.
- If both stops and locks are specified, then lock specification takes precedence. That is, if the degree of freedom is locked at a given value, then it will remain locked for the rest of the analysis.
- In a nonlinear analysis, the component of relative motion (rotation around the revolute axis) is accumulated over all the substeps. It is essential that the substep size be restricted such that this rotation in a given substep is less than π for the values to be accumulated correctly.
- The element currently does not support birth or death options.
- The equation solver (**EQSLV**) must be the sparse solver or the PCG solver. The command **PCGOPT**,,,,,,ON is also required in order to use the PCG solver.
- The element coordinate system (**/PSYMB**,ESYS) is not relevant.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MPC184 Revolute Joint Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- No special features are allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MPC184-Universal

Multipoint Constraint Element: Universal Joint

MP ME ST PR PRN <> <> <> PP EME MFS
Product Restrictions

MPC184 Universal Joint Element Description

The MPC184 universal joint element is a two-node element that has two free relative rotational degrees of freedom. The two nodes forming the element must have identical spatial coordinates.

Figure 184univ.1: MPC184 Universal Joint Geometry

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MPC184 Universal Joint Input Data

Set KEYOPT(1) = 7 to define a two-node universal joint element.

Figure 184univ.1: MPC184 Universal Joint Geometry (p. 863) shows the geometry and node locations for this element. Two nodes (I and J) define the element. The two nodes are expected to have identical spatial coordinates.

A local Cartesian coordinate system must be specified at the first node, I, of the element. The specification of the second local coordinate system at node J is optional. If the local coordinate system is not specified at node J, then the local coordinate system at node J is assumed to be the same as that at node I. The local 2 direction is usually aligned along the shaft axes of the universal joint. The orientation of local directions must follow the convention specified in Figure 184univ.1: MPC184 Universal Joint Geometry (p. 863). These local coordinate systems evolve with the rotations at the respective nodes (if any). Use the **SECJOINT** command to specify the identifiers of the local coordinate systems.

The constraints imposed in a universal joint element are easily described by considering the two local coordinate systems (Cartesian) attached to node I and node J (Figure 184univ.1: MPC184 Universal Joint Geometry (p. 863)). At any given instant of time, the constraints imposed in a universal joint are as described below.

Displacement constraints:

$$u^I = u^J$$

Where, u^I is the displacement vector at node I, and u^J is the displacement vector at node J.

Rotation constraints:

$$e_1^I \cdot e_3^J = 0$$

If the axes e_2^I and e_2^J are not aligned at the start of the analysis, then the angle between the two is held fixed at the initial value.

The relative position of the local coordinate system at node I with respect to node J is characterized by the first and the third Cardan (or Bryant) angles as:

$$\phi = -\tan^{-1}\left(\frac{e_2^I \cdot e_3^J}{e_3^I \cdot e_3^J}\right)$$

$$\psi = -\tan^{-1}\left(\frac{e_1^I \cdot e_2^J}{e_1^I \cdot e_1^J}\right)$$

The change in the relative angular position between the two local coordinate system is given by

$$u_{r4} = \varphi - \varphi_0$$

$$u_{r6} = \psi - \psi_0$$

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Where, φ_0 and ψ_0 are the initial angular offsets between the two coordinate systems (that is, the first and third Cardan (or Bryant) angles measured in the reference configuration).

The constitutive calculations use the following definition of the joint rotation:

$$u_{r4}^{c} = \phi - \phi_1^{ref}$$

$$u_{r6}^{c} = \psi - \phi_3^{ref}$$

Where, ϕ_1^{ref}, ϕ_3^{ref} are the reference angles, angle1 and angle3, specified on the **SECDATA** command. If these values are not specified, then φ_0 and ψ_0 are used in place of ϕ_1^{ref} and ϕ_3^{ref}, respectively.

Other input data that are common to all joint elements (material behavior, stops and limits, locks, etc.) are described in "Joint Input Data" (p. 839) in the MPC184 element description.

MPC184 Universal Joint Input Summary

This input summary applies to the universal joint element option of MPC184: KEYOPT(1) = 7.

Nodes

I, J

Note

For a grounded joint element, specify either node I or node J in the element definition and leave the other node (the grounded node) blank.

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants

None

Material Properties

Use the JOIN label on the **TB** command to define stiffness and damping behavior. (See MPC184 Joint in the *Material Reference* for detailed information on defining joint materials.)

Surface Loads

None

Body Loads

Temperatures --

T(I), T(J)

Element Loads

Rotations --

ROTX, ROTZ

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Moments --
MX, MZ

Special Features

Large deflection
Linear perturbation

KEYOPT(1)
Element behavior:

7 --
Universal joint element

MPC184 Universal Joint Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 184univ.1: MPC184 Universal Joint Element Output Definitions (p. 866) and Table 184univ.2: MPC184 Universal Joint Element - NMISC Output (p. 868).

These tables use the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 184univ.1: MPC184 Universal Joint Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
NODES	Element node numbers (I, J)	-	Y
FX	Constraint force in X direction	-	Y
FY	Constraint force in Y direction	-	Y
FZ	Constraint force in Z direction	-	Y
MY	Constraint moment in Y direction	-	Y
CSTOP4	Constraint moment if stop is specified on DOF 4	-	Y
CSTOP6	Constraint moment if stop is specified on DOF 6	-	Y
CLOCK4	Constraint moment if lock is specified on DOF 4	-	Y
CLOCK6	Constraint moment if lock is specified on DOF 6	-	Y
CSST4	Constraint stop status on DOF 4[1]	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
CLST4	Constraint lock status on DOF 4[2]	-	Y
CSST6	Constraint stop status on DOF 6[1]	-	Y
CLST6	Constraint lock status on DOF 6[2]	-	Y
JRP4	Joint relative position of DOF4	-	Y
JRP6	Joint relative position of DOF6	-	Y
JCD4	Joint constitutive rotation on DOF4	-	Y
JCD6	Joint constitutive rotation on DOF6	-	Y
JEF4	Joint elastic moment in direction -4	-	Y
JEF6	Joint elastic moment in direction -6	-	Y
JDF4	Joint damping moment in direction -4	-	Y
JDF6	Joint damping moment in direction -6	-	Y
JRU4	Joint relative rotation of DOF4	-	Y
JRU6	Joint relative rotation of DOF6	-	Y
JRV4	Joint relative rotational velocity of DOF4	-	Y
JRV6	Joint relative rotational velocity of DOF6	-	Y
JRA4	Joint relative rotational acceleration of DOF4	-	Y
JRA6	Joint relative rotational acceleration of DOF6	-	Y
JTEMP	Average temperature in the element[3]	-	Y

1. Constraint stop status:

   0 = stop not active, or deactivated
   1 = stopped at minimum value
   2 = stopped at maximum value

2. Constraint lock status:

   0 = lock not active
   1 = locked at minimum value
   2 = locked at maximum value

3. Average temperature in the element when temperatures are applied on the nodes of the element using the **BF** command, or when temperature are applied on the element using the **BFE** command.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following table shows additional non-summable miscellaneous (NMISC) output available for the universal joint element.

Note

This output is intended for use in the ANSYS Workbench program to track the evolution of local coordinate systems specified at the nodes of joint elements.

Table 184univ.2: MPC184 Universal Joint Element - NMISC Output

Name	Definition	O	R
E1X-I, E1Y-I, E1Z-I	X, Y, Z components of the evolved e_1 axis at node I	-	Y
E2X-I, E2Y-I, E2Z-I	X, Y, Z components of the evolved e_2 axis at node I	-	Y
E3X-I, E3Y-I, E3Z-I	X, Y, Z components of the evolved e_3 axis at node I	-	Y
E1X-J, E1Y-J, E1Z-J	X, Y, Z components of the evolved e_1 axis at node J	-	Y
E2X-J, E2Y-J, E2Z-J	X, Y, Z components of the evolved e_2 axis at node J	-	Y
E3X-J, E3Y-J, E3Z-J	X, Y, Z components of the evolved e_3 axis at node J	-	Y
JFX, JFY, JFZ	Constraint forces expressed in the evolved co-ordinate system specified at node I	-	Y
JMX, JMY, JMZ	Constraint moments expressed in the evolved coordinate system specified at node I	-	Y

Table 184univ.3: MPC184 Universal Joint Item and Sequence Numbers - SMISC Items (p. 868) and Table 184univ.4: MPC184 Universal Joint Item and Sequence Numbers - NMISC Items (p. 869) list output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) for further information. The table uses the following notation:

Name
output quantity as defined in the Element Output Definitions table.

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 184univ.3: MPC184 Universal Joint Item and Sequence Numbers - SMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FX	SMISC	1
FY	SMISC	2

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FZ	SMISC	3
MY	SMISC	5
MZ	SMISC	6
CSTOP4	SMISC	10
CSTOP6	SMISC	12
CLOCK4	SMISC	16
CLOCK6	SMISC	18
CSST4	SMISC	22
CLST4	SMISC	28
CSST6	SMISC	24
CLST6	SMISC	30
JRP4	SMISC	34
JRP6	SMISC	36
JCD4	SMISC	40
JCD6	SMISC	42
JEF4	SMISC	46
JEF6	SMISC	48
JDF4	SMISC	52
JDF6	SMISC	54
JRU4	SMISC	64
JRU6	SMISC	66
JRV4	SMISC	70
JRV6	SMISC	72
JRA4	SMISC	76
JRA6	SMISC	78
JTEMP	SMISC	79

Table 184univ.4: MPC184 Universal Joint Item and Sequence Numbers - NMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
E1X-I	NMISC	1
E1Y-I	NMISC	2
E1Z-I	NMISC	3
E2X-I	NMISC	4
E2Y-I	NMISC	5
E2Z-I	NMISC	6
E3X-I	NMISC	7
E3Y-I	NMISC	8

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
E3Z-I	NMISC	9
E1X-J	NMISC	10
E1Y-J	NMISC	11
E1Z-J	NMISC	12
E2X-J	NMISC	13
E2Y-J	NMISC	14
E2Z-J	NMISC	15
E3X-J	NMISC	16
E3Y-J	NMISC	17
E3Z-J	NMISC	18
JFX	NMISC	19
JFY	NMISC	20
JFZ	NMISC	21
JMX	NMISC	22
JMY	NMISC	23
JMZ	NMISC	24

MPC184 Universal Joint Assumptions and Restrictions

- The nodes I and J must be coincident.
- The local coordinate systems at the nodes must be specified such that the axes of rotation are well defined. Otherwise, it is possible that the rotational motion might not be what is expected.
- Boundary conditions cannot be applied on the nodes forming the universal joint.
- Rotational degrees of freedom are activated at the nodes forming the element. When these elements are used in conjunction with solid elements, the rotational degrees of freedom must be suitably constrained. Since boundary conditions cannot be applied to the nodes of the universal joint, a beam or shell element with very weak stiffness may be used with the underlying solid elements at the nodes forming the joint element to avoid any rigid body modes.
- If both stops and locks are specified, then lock specification takes precedence. That is, if the degree of freedom is locked at a given value, then it will remain locked for the rest of the analysis.
- In a nonlinear analysis, the components of relative motion are accumulated over all the substeps. It is essential that the substep size be restricted such that these rotations in a given substep are less than π for the values to be accumulated correctly.
- The element currently does not support birth or death options.
- The equation solver (**EQSLV**) must be the sparse solver or the PCG solver. The command **PCGOPT**,,,,,,ON is also required in order to use the PCG solver.
- The element coordinate system (**/PSYMB**,ESYS) is not relevant.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MPC184 Universal Joint Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- No special features are allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

MPC184-Slot

Multipoint Constraint Element: Slot Joint

MP ME ST PR PRN <> <> <> PP EME MFS
Product Restrictions

MPC184 Slot Joint Element Description

The MPC184 slot joint element is a two-node element that has one relative displacement degree of freedom. The rotational degrees of freedom at nodes I and J are left free.

Figure 184slot.1: MPC184 Slot Joint Geometry

MPC184 Slot Joint Input Data

Set KEYOPT(1) = 8 to define a two-node slot joint element.

Figure 184slot.1: MPC184 Slot Joint Geometry (p. 873) shows the geometry and node locations for this element. Two nodes (I and J) define the element.

A local Cartesian coordinate system must be specified at the first node, I, of the element. The second node, J, is constrained to move on the local $\mathbf{e}_1$ axis specified at node I. The local coordinate system specified at node I evolves with the rotations at node I. Use the **SECJOINT** command to specify the identifiers of the local coordinate systems.

The constraints imposed on a slot joint element are easily described by referring to Figure 184slot.1: MPC184 Slot Joint Geometry (p. 873). At any given instant of time, the constraints imposed in a 3-D slot joint are as follows:

$$\mathbf{e}_2^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - \mathbf{E}_2^I \cdot (\mathbf{X}^J - \mathbf{X}^I) = 0$$
$$\mathbf{e}_3^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - \mathbf{E}_3^I \cdot (\mathbf{X}^J - \mathbf{X}^I) = 0$$

Where, $\mathbf{x}^I$ and $\mathbf{x}^J$ are the position vectors of nodes I and J in the current configuration, and $\mathbf{X}^I$ and $\mathbf{X}^J$ are the position vectors of nodes I and J in the reference configuration. Essentially these constraints

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

force the node J to move along the $\mathbf{e}_1$ axis of the local coordinate system specified at node I. $\mathbf{e}^I$ are in the current configuration, while $\mathbf{E}^I$ are specified in the initial configuration.

The change in the relative position of the nodes I and J is given by:

$$u_1 = \ell - \ell_0$$

Where u is the initial offset computed based on the initial configuration and the local coordinate system associated with node I, and

$$\ell = \mathbf{e}_1^I \cdot (\mathbf{x}^J - \mathbf{x}^I) \text{ and } \ell_0 = \mathbf{E}_1^I \cdot (\mathbf{X}^J - \mathbf{X}^I)$$

The constitutive calculations use the following definition of the joint displacement:

$$u_1^c = \ell - \ell_1^{ref}$$

where:

ℓ_1^{ref} = reference length, length1, specified on **SECDATA** command.

If the reference length is not specified, the initial offset is used.

Other input data that are common to all joint elements (material behavior, stops and limits, locks, etc.) are described in "Joint Input Data" (p. 839) in the MPC184 element description.

MPC184 Slot Joint Input Summary

This input summary applies to the slot joint element option of MPC184: KEYOPT(1) = 8.

Nodes

I, J

Note

For a grounded joint element, specify either node I or node J in the element definition and leave the other node (the grounded node) blank.

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants

None

Material Properties

Use the JOIN label on the **TB** command to define stiffness, damping, and Coulomb friction behavior. (See MPC184 Joint in the *Material Reference* for detailed information on defining joint materials.)

Surface Loads

None

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Body Loads

Temperatures --
T(I), T(J)

Element Loads

Displacements --
UX

Forces --
FX

Special Features

Large deflection
Linear perturbation

KEYOPT(1)
Element behavior:

8 --
Slot joint element

MPC184 Slot Joint Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 184slot.1: MPC184 Slot Joint Element Output Definitions (p. 875) and Table 184slot.2: MPC184 Slot Joint Element - NMISC Output (p. 876).

These tables use the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 184slot.1: MPC184 Slot Joint Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
NODES	Element node numbers (I, J)	-	Y
FY	Constraint force in Y direction	-	Y
FZ	Constraint force in Z direction	-	Y
CSTOP1	Constraint force if stop is specified on DOF 1	-	Y
CLOCK1	Constraint force if lock is specified on DOF 1	-	Y
CSST1	Constraint stop status[1]	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
CLST1	Constraint lock status[2]	-	Y
JRP1	Joint relative position	-	Y
JCD1	Joint constitutive displacement	-	Y
JEF1	Joint elastic force	-	Y
JDF1	Joint damping force	-	Y
JFF1	Joint friction force	-	Y
JRU1	Joint relative displacement	-	Y
JRA1	Joint relative acceleration	-	Y
JRV1	Joint relative velocity	-	Y
JTEMP	Average temperature in the element[3]	-	Y
JFST1	Slip/stick status when friction is specified[4]	-	Y
JFNF1	Normal moment in friction calculations	-	Y

1. Constraint stop status:

   0 = stop not active, or deactivated
   1 = stopped at minimum value
   2 = stopped at maximum value

2. Constraint lock status:

   0 = lock not active
   1 = locked at minimum value
   2 = locked at maximum value

3. Average temperature in the element when temperatures are applied on the nodes of the element using the **BF** command, or when temperature are applied on the element using the **BFE** command.

4. Stick/slip status when friction is active:

   0 = friction is not activated
   1 = sticking
   2 = slipping or sliding

The following table shows additional non-summable miscellaneous (NMISC) output available for the slot joint element.

Note

This output is intended for use in the ANSYS Workbench program to track the evolution of local coordinate systems specified at the nodes of joint elements.

Table 184slot.2: MPC184 Slot Joint Element - NMISC Output

Name	Definition	O	R
E1X-I, E1Y-I, E1Z-I	X, Y, Z components of the evolved e_1 axis at node I	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
E2X-I, E2Y-I, E2Z-I	X, Y, Z components of the evolved e_2 axis at node I	-	Y
E3X-I, E3Y-I, E3Z-I	X, Y, Z components of the evolved e_3 axis at node I	-	Y
E1X-J, E1Y-J, E1Z-J	X, Y, Z components of the evolved e_1 axis at node J	-	Y
E2X-J, E2Y-J, E2Z-J	X, Y, Z components of the evolved e_2 axis at node J	-	Y
E3X-J, E3Y-J, E3Z-J	X, Y, Z components of the evolved e_3 axis at node J	-	Y
JFX, JFY, JFZ	Constraint forces expressed in the evolved co-ordinate system specified at node I	-	Y
JMX, JMY, JMZ	Constraint moments expressed in the evolved coordinate system specified at node I	-	Y

Table 184slot.3: MPC184 Slot Joint Item and Sequence Numbers - SMISC Items (p. 877) and Table 184slot.4: MPC184 Slot Joint Item and Sequence Numbers - NMISC Items (p. 878) list output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) for further information. The table uses the following notation:

Name
output quantity as defined in the Element Output Definitions table.

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 184slot.3: MPC184 Slot Joint Item and Sequence Numbers - SMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FY	SMISC	2
FZ	SMISC	3
CSTOP1	SMISC	7
CLOCK1	SMISC	13
CSST1	SMISC	19
CLST1	SMISC	25
JRP1	SMISC	31
JCD1	SMISC	37
JEF1	SMISC	43
JDF1	SMISC	49
JFF1	SMISC	55
JRU1	SMISC	61

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
JRV1	SMISC	67
JRA1	SMISC	73
JTEMP	SMISC	79
JFST1	SMISC	80
JFNF1	SMISC	81

Table 184slot.4: MPC184 Slot Joint Item and Sequence Numbers - NMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
E1X-I	NMISC	1
E1Y-I	NMISC	2
E1Z-I	NMISC	3
E2X-I	NMISC	4
E2Y-I	NMISC	5
E2Z-I	NMISC	6
E3X-I	NMISC	7
E3Y-I	NMISC	8
E3Z-I	NMISC	9
E1X-J	NMISC	10
E1Y-J	NMISC	11
E1Z-J	NMISC	12
E2X-J	NMISC	13
E2Y-J	NMISC	14
E2Z-J	NMISC	15
E3X-J	NMISC	16
E3Y-J	NMISC	17
E3Z-J	NMISC	18
JFX	NMISC	19
JFY	NMISC	20
JFZ	NMISC	21
JMX	NMISC	22
JMY	NMISC	23
JMZ	NMISC	24

MPC184 Slot Joint Assumptions and Restrictions

- Boundary conditions cannot be applied on the nodes forming the slot joint.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Rotational degrees of freedom are activated at the nodes forming the element. When these elements are used in conjunction with solid elements, the rotational degrees of freedom must be suitably constrained. Since boundary conditions cannot be applied to the nodes of the slot joint, a beam or shell element with very weak stiffness may be used with the underlying solid elements at the nodes forming the joint element to avoid any rigid body modes.
- Stops (**SECSTOP**) and locks (**SECLOCK**) can only be applied on the relative x-direction. These are not applicable to the rotational degrees of freedom.
- If both stops and locks are specified, then lock specification takes precedence. That is, if the degree of freedom is locked at a given value, then it will remain locked for the rest of the analysis.
- In a nonlinear analysis, the components of relative motion are accumulated over all the substeps. It is essential that the substep size be restricted such that these rotations in a given substep are less than π for the values to be accumulated correctly.
- The element currently does not support birth or death options.
- The equation solver (**EQSLV**) must be the sparse solver or the PCG solver. The command **PCGOPT**,,,,,,ON is also required in order to use the PCG solver.
- The element coordinate system (**/PSYMB**,ESYS) is not relevant.

MPC184 Slot Joint Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- No special features are allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

MPC184-Point

Multipoint Constraint Element: Point-in-plane Joint

MP ME ST PR PRN <> <> <> PP EME MFS
Product Restrictions

MPC184 Point-in-plane Joint Element Description

The MPC184 point-in-plane joint element is a two-node element that has two relative displacement degrees of freedom. The relative rotational degrees of freedom are not considered and cannot be controlled.

Figure 184poin.1: MPC184 Point-in-plane Joint Geometry

MPC184 Point-in-plane Joint Input Data

Set KEYOPT(1) = 9 to define a two-node point-in-plane joint element.

Figure 184poin.1: MPC184 Point-in-plane Joint Geometry (p. 881) shows the geometry and node locations for this element. Two nodes (I and J) define the element.

A local Cartesian coordinate system must be specified at the first node, I, of the element. The second node, J, is constrained such that it remains on a plane defined by the local e_2^I and e_3^I axes. The normal distance from this plane containing node J to node I is held fixed. The local coordinate system specified at node I evolves with the rotations at node I. Use the **SECJOINT** command to specify the identifiers of the local coordinate systems.

The constraints imposed on a point-in-plane joint element are easily described by referring to Figure 184poin.1: MPC184 Point-in-plane Joint Geometry (p. 881). At any given instant of time, the constraint imposed is as follows:

$$\mathbf{e}_1^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - E_1^I \cdot (\mathbf{X}^J - \mathbf{X}^I) = 0$$

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Where, x^I and x^J are the positional vectors of nodes I and J in the current configuration, and X^I and X^J are the position vectors of nodes I and J in the reference configuration. e^I are in the current configuration, while E^I are specified in the initial configuration.

The changes in the relative position of the nodes I and J are given by:

$$u_2 = \mathbf{e}_2^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - E_2^I \cdot (\mathbf{X}^J - \mathbf{X}^I)$$

$$u_3 = \mathbf{e}_3^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - E_3^I \cdot (\mathbf{X}^J - \mathbf{X}^I)$$

The constitutive calculations use the following definition of the joint displacement:

$$u_2^c = \mathbf{e}_2^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - \ell_2^{ref}$$

$$u_3^c = \mathbf{e}_3^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - \ell_3^{ref}$$

where:

ℓ_2^{ref} = reference length specified on the **SECDATA** command

ℓ_3^{ref} = reference length specified on the **SECDATA** command

If the reference lengths are not specified, the initial offsets are used.

Other input data that are common to all joint elements (material behavior, stops and limits, locks, etc.) are described in "Joint Input Data" (p. 839) in the MPC184 element description.

MPC184 Point-in-plane Joint Input Summary

This input summary applies to the point-in-plane joint element option of MPC184: KEYOPT(1) = 9.

Nodes

I, J

Note

For a grounded joint element, specify either node I or node J in the element definition and leave the other node (the grounded node) blank.

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants

None

Material Properties

Use the JOIN label on the **TB** command to define stiffness and damping behavior. (See MPC184 Joint in the *Material Reference* for detailed information on defining joint materials.)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface Loads
None

Body Loads

Temperatures --
T(I), T(J)

Element Loads
None

Special Features

Large deflection
Linear perturbation

KEYOPT(1)
Element behavior:

9 --
Point-in-plane joint element

MPC184 Point-in-plane Joint Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 184poin.1: MPC184 Point-in-plane Joint Element Output Definitions (p. 883) and Table 184poin.2: MPC184 Point-in-plane Joint Element - NMISC Output (p. 885).

These tables use the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 184poin.1: MPC184 Point-in-plane Joint Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
NODES	Element node numbers (I, J)	-	Y
FX	Constraint Force in X direction	-	Y
CSTOP2	Constraint force if stop is specified on DOF 2	-	Y
CSTOP3	Constraint force if stop is specified on DOF 3	-	Y
CLOCK2	Constraint force if lock is specified on DOF 2	-	Y
CLOCK3	Constraint force if lock is specified on DOF 3	-	Y
CSST2	Constraint stop status on DOF 2[1]	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
CLST2	Constraint lock status on DOF 2[2]	-	Y
CSST3	Constraint stop status on DOF 3[1]	-	Y
CLST3	Constraint lock status on DOF 3[2]	-	Y
JRP2	Joint relative position of DOF2	-	Y
JRP3	Joint relative position of DOF3	-	Y
JCD2	Joint constitutive displacement on DOF2	-	Y
JCD3	Joint constitutive displacement on DOF3	-	Y
JEF2	Joint elastic force in direction -2	-	Y
JEF3	Joint elastic force in direction -3	-	Y
JDF2	Joint damping force in direction -2	-	Y
JDF3	Joint damping force in direction -3	-	Y
JRU2	Joint relative displacement in direction -2	-	Y
JRU3	Joint relative displacement in direction -3	-	Y
JRV2	Joint relative velocity in direction -2	-	Y
JRV3	Joint relative velocity in direction -3	-	Y
JRA2	Joint relative acceleration in direction -2	-	Y
JRA3	Joint relative acceleration in direction -3	-	Y
JTEMP	Average temperature in the element[3]	-	Y

1. Constraint stop status:

   0 = stop not active, or deactivated
   1 = stopped at minimum value
   2 = stopped at maximum value

2. Constraint lock status:

   0 = lock not active
   1 = locked at minimum value
   2 = locked at maximum value

3. Average temperature in the element when temperatures are applied on the nodes of the element using the **BF** command, or when temperature are applied on the element using the **BFE** command.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following table shows additional non-summable miscellaneous (NMISC) output available for the point-in-plane joint element.

Note

This output is intended for use in the ANSYS Workbench program to track the evolution of local coordinate systems specified at the nodes of joint elements.

Table 184poin.2: MPC184 Point-in-plane Joint Element - NMISC Output

Name	Definition	O	R
E1X-I, E1Y-I, E1Z-I	X, Y, Z components of the evolved e_1 axis at node I	-	Y
E2X-I, E2Y-I, E2Z-I	X, Y, Z components of the evolved e_2 axis at node I	-	Y
E3X-I, E3Y-I, E3Z-I	X, Y, Z components of the evolved e_3 axis at node I	-	Y
E1X-J, E1Y-J, E1Z-J	X, Y, Z components of the evolved e_1 axis at node J	-	Y
E2X-J, E2Y-J, E2Z-J	X, Y, Z components of the evolved e_2 axis at node J	-	Y
E3X-J, E3Y-J, E3Z-J	X, Y, Z components of the evolved e_3 axis at node J	-	Y
JFX, JFY, JFZ	Constraint forces expressed in the evolved co-ordinate system specified at node I	-	Y
JMX, JMY, JMZ	Constraint moments expressed in the evolved coordinate system specified at node I	-	Y

Table 184poin.3: MPC184 Point-in-plane Joint Item and Sequence Numbers - SMISC Items (p. 885) and Table 184poin.4: MPC184 Point-in-plane Joint Item and Sequence Numbers - NMISC Items (p. 886) list output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) for further information. The table uses the following notation:

Name
output quantity as defined in the Element Output Definitions table.

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 184poin.3: MPC184 Point-in-plane Joint Item and Sequence Numbers - SMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FX	SMISC	1
CSTOP2	SMISC	8

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
CSTOP3	SMISC	9
CLOCK2	SMISC	14
CLOCK3	SMISC	15
CSST2	SMISC	20
CLST2	SMISC	26
CSST3	SMISC	21
CLST3	SMISC	27
JRP2	SMISC	32
JRP3	SMISC	33
JCD2	SMISC	38
JCD3	SMISC	39
JEF2	SMISC	44
JEF3	SMISC	45
JDF2	SMISC	50
JDF3	SMISC	51
JRU2	SMISC	62
JRU3	SMISC	63
JRV2	SMISC	68
JRV3	SMISC	69
JRA2	SMISC	74
JRA3	SMISC	75
JTEMP	SMISC	79

Table 184poin.4: MPC184 Point-in-plane Joint Item and Sequence Numbers - NMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
E1X-I	NMISC	1
E1Y-I	NMISC	2
E1Z-I	NMISC	3
E2X-I	NMISC	4
E2Y-I	NMISC	5
E2Z-I	NMISC	6
E3X-I	NMISC	7
E3Y-I	NMISC	8
E3Z-I	NMISC	9
E1X-J	NMISC	10
E1Y-J	NMISC	11
E1Z-J	NMISC	12

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
E2X-J	NMISC	13
E2Y-J	NMISC	14
E2Z-J	NMISC	15
E3X-J	NMISC	16
E3Y-J	NMISC	17
E3Z-J	NMISC	18
JFX	NMISC	19
JFY	NMISC	20
JFZ	NMISC	21
JMX	NMISC	22
JMY	NMISC	23
JMZ	NMISC	24

MPC184 Point-in-plane Joint Assumptions and Restrictions

- Boundary conditions cannot be applied on the nodes forming the point-in-plane joint.
- Rotational degrees of freedom are activated at the nodes forming the element. When these elements are used in conjunction with solid elements, the rotational degrees of freedom must be suitably constrained. Since boundary conditions cannot be applied to the nodes of the point-in-plane joint, a beam or shell element with very weak stiffness may be used with the underlying solid elements at the nodes forming the joint element to avoid any rigid body modes.
- If both stops and locks are specified, then lock specification takes precedence. That is, if the degree of freedom is locked at a given value, then it will remain locked for the rest of the analysis.
- In a nonlinear analysis, the components of relative motion are accumulated over all the substeps. It is essential that the substep size be restricted such that these rotations in a given substep are less than π for the values to be accumulated correctly.
- The element currently does not support birth or death options.
- The equation solver (**EQSLV**) must be the sparse solver or the PCG solver. The command **PCGOPT**,,,,,,ON is also required in order to use the PCG solver.
- The element coordinate system (**/PSYMB**,ESYS) is not relevant.

MPC184 Point-in-plane Joint Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- No special features are allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

MPC184-Trans

Multipoint Constraint Element: Translational Joint

MP ME ST PR PRN <> <> <> PP EME MFS
Product Restrictions

MPC184 Translational Joint Element Description

The MPC184 translational joint element is a two-node element that has one relative displacement degree of freedom. All other relative degrees of freedom are fixed.

Figure 184tran.1: MPC184 Translational Joint Geometry

MPC184 Translational Joint Input Data

Set KEYOPT(1) = 10 to define a two-node translational joint element.

Figure 184tran.1: MPC184 Translational Joint Geometry (p. 889) shows the geometry and node locations for this element. Two nodes (I and J) define the element.

A local Cartesian coordinate system must be specified at the first node, I, of the element. The local coordinate system specification at the second node is optional. The second node J is constrained to move along the e_1 axis specified at node I. The local coordinate system specified at node I evolves with the rotations at node I. Use the **SECJOINT** command to specify the identifiers of the local coordinate systems.

The constraints imposed on a translational joint element are easily described by referring to Figure 184tran.1: MPC184 Translational Joint Geometry (p. 889). At any given instant of time, the constraints imposed are as follows:

$$\mathbf{e}_2^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - E_2^I \cdot (\mathbf{X}^J - \mathbf{X}^I) = 0$$

$$\mathbf{e}_3^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - E_3^I \cdot (\mathbf{X}^J - \mathbf{X}^I) = 0$$

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\mathbf{e}_1^I \cdot \mathbf{e}_2^J - \mathbf{E}_1^I \cdot \mathbf{E}_2^J = 0$$

$$\mathbf{e}_2^I \cdot \mathbf{e}_3^J - \mathbf{E}_2^I \cdot \mathbf{E}_3^J = 0$$

$$\mathbf{e}_1^I \cdot \mathbf{e}_3^J - \mathbf{E}_1^I \cdot \mathbf{E}_3^J = 0$$

Where, x^I and x^J are the positional vectors of nodes I and J in the current configuration, and X^I and X^J are the position vectors of nodes I and J in the reference configuration. Essentially these constraints force the node J to move along the e_1 axis of the local coordinate system specified at node I. e^I are in the current configuration, while E^I are specified in the initial configuration.

The change in the relative position of the nodes I and J is given by:

$$u_1 = \mathbf{e}_1^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - \mathbf{E}_1^I \cdot (\mathbf{X}^J - \mathbf{X}^I)$$

The constitutive calculations use the following definition of the joint displacement:

$$u_1^c = \mathbf{e}_1^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - \ell_1^{ref}$$

where:

ℓ_1^{ref} = reference length, length1, specified on **SECDATA** command

If the reference length is not specified, the initial offset is used.

Other input data that are common to all joint elements (material behavior, stops and limits, locks, etc.) are described in "Joint Input Data" (p. 839) in the MPC184 element description.

MPC184 Translational Joint Input Summary

This input summary applies to the translational joint element option of MPC184: KEYOPT(1) = 10.

Nodes

I, J

Note

For a grounded joint element, specify either node I or node J in the element definition and leave the other node (the grounded node) blank.

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants

None

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Material Properties

Use the JOIN label on the **TB** command to define stiffness, damping, and Coulomb friction behavior. (See MPC184 Joint in the *Material Reference* for detailed information on defining joint materials.)

Surface Loads

None

Body Loads

Temperatures --

T(I), T(J)

Element Loads

None

Special Features

Large deflection
Linear perturbation

KEYOPT(1)

Element behavior:

10 --

Translational joint element

MPC184 Translational Joint Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 184tran.1: MPC184 Translational Joint Element Output Definitions (p. 891) and Table 184tran.2: MPC184 Translational Joint Element - NMISC Output (p. 893).

These tables use the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 184tran.1: MPC184 Translational Joint Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Element node numbers (I, J)	-	Y
FY	Constraint force in Y direction	-	Y
FZ	Constraint force in Z direction	-	Y
MX	Constraint moment in X direction	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
MY	Constraint moment in Y direction	-	Y
MZ	Constraint moment in Z direction	-	Y
CSTOP1	Constraint force if stop is specified on DOF 1	-	Y
CLOCK1	Constraint force if lock is specified on DOF 1	-	Y
CSST1	Constraint stop status[1]	-	Y
CLST1	Constraint lock status[2]	-	Y
JRP1	Joint relative position	-	Y
JCD1	Joint constitutive displacement	-	Y
JEF1	Joint elastic force	-	Y
JDF1	Joint damping force	-	Y
JFF1	Joint friction force	-	Y
JRU1	Joint relative displacement	-	Y
JRV1	Joint relative velocity	-	Y
JRA1	Joint relative acceleration	-	Y
JTEMP	Average temperature in the element[3]	-	Y
JFST1	Slip/stick status when friction is specified[4]	-	Y
JFNF1	Normal moment in friction calculations	-	Y

1. Constraint stop status:

   0 = stop not active, or deactivated
   1 = stopped at minimum value
   2 = stopped at maximum value

2. Constraint lock status:

   0 = lock not active
   1 = locked at minimum value
   2 = locked at maximum value

3. Average temperature in the element when temperatures are applied on the nodes of the element using the **BF** command, or when temperature are applied on the element using the **BFE** command.

4. Stick/slip status when friction is active:

   0 = friction is not activated
   1 = sticking
   2 = slipping or sliding

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following table shows additional non-summable miscellaneous (NMISC) output available for the translational joint element.

Note

This output is intended for use in the ANSYS Workbench program to track the evolution of local coordinate systems specified at the nodes of joint elements.

Table 184tran.2: MPC184 Translational Joint Element - NMISC Output

Name	Definition	O	R
E1X-I, E1Y-I, E1Z-I	X, Y, Z components of the evolved e_1 axis at node I	-	Y
E2X-I, E2Y-I, E2Z-I	X, Y, Z components of the evolved e_2 axis at node I	-	Y
E3X-I, E3Y-I, E3Z-I	X, Y, Z components of the evolved e_3 axis at node I	-	Y
E1X-J, E1Y-J, E1Z-J	X, Y, Z components of the evolved e_1 axis at node J	-	Y
E2X-J, E2Y-J, E2Z-J	X, Y, Z components of the evolved e_2 axis at node J	-	Y
E3X-J, E3Y-J, E3Z-J	X, Y, Z components of the evolved e_3 axis at node J	-	Y
JFX, JFY, JFZ	Constraint forces expressed in the evolved co-ordinate system specified at node I	-	Y
JMX, JMY, JMZ	Constraint moments expressed in the evolved coordinate system specified at node I	-	Y

Table 184tran.3: MPC184 Translational Joint Item and Sequence Numbers - SMISC Items (p. 893) and Table 184tran.4: MPC184 Translational Joint Item and Sequence Numbers - NMISC Items (p. 894) list output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) for further information. The table uses the following notation:

Name
output quantity as defined in the Element Output Definitions table.

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 184tran.3: MPC184 Translational Joint Item and Sequence Numbers - SMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FY	SMISC	2
FZ	SMISC	3

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
MX	SMISC	4
MY	SMISC	5
MZ	SMISC	6
CSTOP1	SMISC	7
CLOCK1	SMISC	13
CSST1	SMISC	19
CLST1	SMISC	25
JRP1	SMISC	31
JCD1	SMISC	37
JEF1	SMISC	43
JDF1	SMISC	49
JFF1	SMISC	55
JRU1	SMISC	61
JRV1	SMISC	67
JRA1	SMISC	73
JTEMP	SMISC	79
JFST1	SMISC	80
JFNF1	SMISC	81

Table 184tran.4: MPC184 Translational Joint Item and Sequence Numbers - NMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
E1X-I	NMISC	1
E1Y-I	NMISC	2
E1Z-I	NMISC	3
E2X-I	NMISC	4
E2Y-I	NMISC	5
E2Z-I	NMISC	6
E3X-I	NMISC	7
E3Y-I	NMISC	8
E3Z-I	NMISC	9
E1X-J	NMISC	10
E1Y-J	NMISC	11
E1Z-J	NMISC	12
E2X-J	NMISC	13
E2Y-J	NMISC	14
E2Z-J	NMISC	15
E3X-J	NMISC	16

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
E3Y-J	NMISC	17
E3Z-J	NMISC	18
JFX	NMISC	19
JFY	NMISC	20
JFZ	NMISC	21
JMX	NMISC	22
JMY	NMISC	23
JMZ	NMISC	24

MPC184 Translational Joint Assumptions and Restrictions

- Boundary conditions cannot be applied on the nodes forming the translational joint.
- Rotational degrees of freedom are activated at the nodes forming the element. When these elements are used in conjunction with solid elements, the rotational degrees of freedom must be suitably constrained. Since boundary conditions cannot be applied to the nodes of the translational joint, a beam or shell element with very weak stiffness may be used with the underlying solid elements at the nodes forming the joint element to avoid any rigid body modes.
- If both stops and locks are specified, then lock specification takes precedence. That is, if the degree of freedom is locked at a given value, then it will remain locked for the rest of the analysis.
- In a nonlinear analysis, the components of relative motion are accumulated over all the substeps. It is essential that the substep size be restricted such that these rotations in a given substep are less than π for the values to be accumulated correctly.
- The element currently does not support birth or death options.
- The equation solver (**EQSLV**) must be the sparse solver or the PCG solver. The command **PCGOPT**,,,,,,ON is also required in order to use the PCG solver.
- The element coordinate system (**/PSYMB**,ESYS) is not relevant.

MPC184 Translational Joint Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- No special features are allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

MPC184-Cylin

Multipoint Constraint Element: Cylindrical Joint

MP ME ST PR PRN <> <> <> PP EME MFS
Product Restrictions

MPC184 Cylindrical Joint Element Description

The MPC184 cylindrical joint element is a two-node element that has one free relative displacement degree of freedom and one free relative rotational degree of freedom (around the cylindrical or revolute axis). All other relative degrees of freedom are fixed.

Figure 184cyl.1: MPC184 Cylindrical Joint Geometry

MPC184 Cylindrical Joint Input Data

Set KEYOPT(1) = 11 to define a two-node cylindrical joint element.

Figure 184cyl.1: MPC184 Cylindrical Joint Geometry (p. 897) shows the geometry and node locations for this element. Two nodes (I and J) define the element.

If KEYOPT(4) = 0, then the element is an x-axis cylindrical joint element with the local e_1 axis as the cylindrical or revolute axis. Translational motion along this axis is also allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If KEYOPT(4) = 1, then the element is a z-axis cylindrical joint element with the local e_3 axis as the cylindrical or revolute axis. Translational motion along this axis is also allowed.

A local Cartesian coordinate system must be specified at the first node, I, of the element. The local coordinate system specification at the second node is optional. The local coordinate systems specified at node I and J evolve with the rotations at the respective nodes. Use the **SECJOINT** command to specify the identifiers of the local coordinate systems.

The constraints imposed in a cylindrical joint element with local e_1 axis as the cylindrical or revolute axis are described below. Similar constraint conditions are set up when the local e_3 axis is the cylindrical or revolute axis. Referring to Figure 184cyl.1: MPC184 Cylindrical Joint Geometry (p. 897), with local coordinate systems specified at nodes I and J, the constraints imposed at any given time are as follows:

$$\mathbf{e}_2^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - E_2^I \cdot (\mathbf{X}^J - \mathbf{X}^I) = 0$$

$$\mathbf{e}_3^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - E_3^I \cdot (\mathbf{X}^J - \mathbf{X}^I) = 0$$

$$\mathbf{e}_1^I \cdot \mathbf{e}_2^J - \mathbf{E}_1^I \cdot \mathbf{E}_2^J = 0$$

$$\mathbf{e}_1^I \cdot \mathbf{e}_3^J - \mathbf{E}_1^I \cdot \mathbf{E}_3^J = 0$$

The change in the relative position of the nodes I and J is given by:

$$u_1 = \ell - \ell_0$$

where:

$$\ell = \mathbf{e}_1^I \cdot (\mathbf{x}^J - \mathbf{x}^I) \text{ and } \ell_0 = \mathbf{E}_1^I \cdot (\mathbf{X}^J - \mathbf{X}^I)$$

The relative rotation between nodes I and J is given by:

$$\phi = -\tan^{-1}\left(\frac{e_2^I \cdot e_3^J}{e_3^I \cdot e_3^J}\right)$$

The change in the relative angular position between the two local coordinate systems is given by

$$u_r = \varphi - \varphi_0 + m\pi$$

where φ_0 is the initial angular offset between the two coordinate systems and m is an integer accounting for multiple rotations about the cylindrical axis.

The constitutive calculations use the following definition of the joint displacement:

$$u_1^c = \mathbf{e}_1^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - \ell_1^{ref}$$

where:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ℓ_1^{ref} = reference length specified on **SECDATA** command.

The constitutive calculations use the following definition of the joint rotation:

$$u_{r4}^{c} = \phi + m\pi - \phi_1^{ref}$$

where:

ϕ_1^{ref} = reference angle, angle1, specified on the **SECDATA** command. If this value is not specified, then Φ_0 is used in place of ϕ_1^{ref}

Other input data that are common to all joint elements (material behavior, stops and limits, locks, etc.) are described in "Joint Input Data" (p. 839) in the MPC184 element description.

MPC184 Cylindrical Joint Input Summary

This input summary applies to the cylindrical joint element option of MPC184: KEYOPT(1) = 11.

Nodes

I, J

Note

For a grounded joint element, specify either node I or node J in the element definition and leave the other node (the grounded node) blank.

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants

None

Material Properties

Use the JOIN label on the **TB** command to define stiffness, and damping behavior. (See MPC184 Joint in the *Material Reference* for detailed information on defining joint materials.)

Surface Loads

None

Body Loads

Temperatures --

T(I), T(J)

Element Loads when KEYOPT(4) = 0:

Displacements/Rotations --

UX, ROTX

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Velocities --
VELX, OMGX

Accelerations --
ACCX, DMGX

Force/Moments --
FX, MX

Element Loads when KEYOPT(4) = 1:

Displacements/Rotations --
UZ, ROTZ

Velocities --
VELZ, OMGZ

Accelerations --
ACCZ, DMGZ

Force/Moments --
FZ, MZ

Special Features

Large deflection
Linear perturbation

KEYOPT(1)
Element behavior:

11 --
Cylindrical joint element

KEYOPT(4)
Element configuration:

0 --
x-axis Cylindrical joint with local 1 direction as the cylindrical axis.

1 --
z-axis Cylindrical joint with local 3 direction as the cylindrical axis.

MPC184 Cylindrical Joint Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 184cyl.1: MPC184 Cylindrical Joint Element Output Definitions (p. 901) and Table 184cyl.2: MPC184 Cylindrical Joint Element - NMISC Output (p. 903).

These tables use the following notation:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 184cyl.1: MPC184 Cylindrical Joint Element Output Definitions

Name	Definition	O	R
x-axis Cylindrical Joint Element (KEYOPT(4) = 0)			
EL	Element Number	-	Y
NODES	Element node numbers (I, J)	-	Y
FY	Constraint Force in Y direction	-	Y
FZ	Constraint Force in Z direction	-	Y
MY	Constraint Moment in Y direction	-	Y
MZ	Constraint Moment in Z direction	-	Y
CSTOP1	Constraint force if stop is specified on DOF 1	-	Y
CSTOP4	Constraint moment if stop is specified on DOF 4	-	Y
CLOCK1	Constraint force if lock is specified on DOF 1	-	Y
CLOCK4	Constraint moment if lock is specified on DOF 4	-	Y
CSST1	Constraint stop status on DOF 1[1]	-	Y
CLST1	Constraint lock status on DOF 1[2]	-	Y
CSST4	Constraint stop status on DOF 4[1]	-	Y
CLST4	Constraint lock status on DOF 4[2]	-	Y
JRP1	Joint relative position of DOF 1	-	Y
JRP4	Joint relative position of DOF 4	-	Y
JCD1	Joint constitutive displacement on DOF 1	-	Y
JCD4	Joint constitutive rotation on DOF 4	-	Y
JEF1	Joint elastic force in direction -1	-	Y
JEF4	Joint elastic moment in direction -4	-	Y
JDF1	Joint damping force in direction -1	-	Y
JDF4	Joint damping moment in direction -4	-	Y
JRU1	Joint relative displacement of DOF 1	-	Y
JRU4	Joint relative rotation of DOF 4	-	Y
JRV1	Joint relative velocity of DOF 1	-	Y
JRV4	Joint relative rotational velocity of DOF 4	-	Y
JRA1	Joint relative acceleration of DOF 1	-	Y
JRA4	Joint relative rotational acceleration of DOF 4	-	Y
JTEMP	Average temperature in the element[3]	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
z-axis Cylindrical Joint Element (KEYOPT(4) = 1)			
EL	Element Number	-	Y
NODES	Element node numbers (I, J)	-	Y
FX	Constraint Force in X direction	-	Y
FY	Constraint Force in Y direction	-	Y
MX	Constraint Moment in X direction	-	Y
MY	Constraint Moment in Y direction	-	Y
CSTOP3	Constraint force if stop is specified on DOF 3	-	Y
CSTOP6	Constraint moment if stop is specified on DOF 6	-	Y
CLOCK3	Constraint force if lock is specified on DOF 3	-	Y
CLOCK6	Constraint moment if lock is specified on DOF 6	-	Y
CSST3	Constraint stop status on DOF 3[1]	-	Y
CLST3	Constraint lock status on DOF 3[2]	-	Y
CSST6	Constraint stop status on DOF 6[1]	-	Y
CLST6	Constraint lock status on DOF 6[2]	-	Y
JRP3	Joint relative position of DOF 3	-	Y
JRP6	Joint relative position of DOF 6	-	Y
JCD3	Joint constitutive displacement on DOF 3	-	Y
JCD6	Joint constitutive rotation on DOF 6	-	Y
JEF3	Joint elastic force in direction -3	-	Y
JEF6	Joint elastic moment in direction -6	-	Y
JDF3	Joint damping force in direction -3	-	Y
JDF6	Joint damping moment in direction -6	-	Y
JRU3	Joint relative displacement of DOF 3	-	Y
JRU6	Joint relative rotation of DOF 6	-	Y
JRV3	Joint relative velocity of DOF 3	-	Y
JRV6	Joint relative rotational velocity of DOF 6	-	Y
JRA3	Joint relative acceleration of DOF 3	-	Y
JRA6	Joint relative rotational acceleration of DOF 6	-	Y
JTEMP	Average temperature in the element[3]	-	Y

1. Constraint stop status:

   0 = stop not active, or deactivated
   1 = stopped at minimum value
   2 = stopped at maximum value

2. Constraint lock status:

   0 = lock not active

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 = locked at minimum value
2 = locked at maximum value

3. Average temperature in the element when temperatures are applied on the nodes of the element using the **BF** command, or when temperature are applied on the element using the **BFE** command.

The following table shows additional non-summable miscellaneous (NMISC) output available for all forms of the cylindrical joint element.

Note

This output is intended for use in the ANSYS Workbench program to track the evolution of local coordinate systems specified at the nodes of joint elements.

Table 184cyl.2: MPC184 Cylindrical Joint Element - NMISC Output

Name	Definition	O	R
The following output is available for all cylindrical joint elements (KEYOPT(4) = 0 and 1)			
E1X-I, E1Y-I, E1Z-I	X, Y, Z components of the evolved e_1 axis at node I	-	Y
E2X-I, E2Y-I, E2Z-I	X, Y, Z components of the evolved e_2 axis at node I	-	Y
E3X-I, E3Y-I, E3Z-I	X, Y, Z components of the evolved e_3 axis at node I	-	Y
E1X-J, E1Y-J, E1Z-J	X, Y, Z components of the evolved e_1 axis at node J	-	Y
E2X-J, E2Y-J, E2Z-J	X, Y, Z components of the evolved e_2 axis at node J	-	Y
E3X-J, E3Y-J, E3Z-J	X, Y, Z components of the evolved e_3 axis at node J	-	Y
JFX, JFY, JFZ	Constraint forces expressed in the evolved co-ordinate system specified at node I	-	Y
JMX, JMY, JMZ	Constraint moments expressed in the evolved coordinate system specified at node I	-	Y

Table 184cyl.3: MPC184 Cylindrical Joint Item and Sequence Numbers - SMISC Items (p. 904) and Table 184cyl.4: MPC184 Cylindrical Joint Item and Sequence Numbers - NMISC Items (p. 905) list output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) for further information. The table uses the following notation:

Name
output quantity as defined in the Element Output Definitions table.

Item
predetermined Item label for **ETABLE** command

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

E

sequence number for single-valued or constant element data

Table 184cyl.3: MPC184 Cylindrical Joint Item and Sequence Numbers - SMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
x-axis Cylindrical Joint Element (KEYOPT(4) = 0)		
FY	SMISC	2
FZ	SMISC	3
MY	SMISC	5
MZ	SMISC	6
CSTOP1	SMISC	7
CSTOP4	SMISC	10
CLOCK1	SMISC	13
CLOCK4	SMISC	16
CSST1	SMISC	19
CSST4	SMISC	22
CLST1	SMISC	25
CLST4	SMISC	28
JRP1	SMISC	31
JRP4	SMISC	34
JCD1	SMISC	37
JCD4	SMISC	40
JEF1	SMISC	43
JEF4	SMISC	46
JDF1	SMISC	49
JDF4	SMISC	52
JRU1	SMISC	61
JRU4	SMISC	64
JRV1	SMISC	67
JRV4	SMISC	70
JRA1	SMISC	73
JRA4	SMISC	76
JTEMP	SMISC	79
z-axis Cylindrical Joint Element (KEYOPT(4) = 1)		
FX	SMISC	1
FY	SMISC	2
MX	SMISC	4
MY	SMISC	5
CSTOP3	SMISC	9

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
CSTOP6	SMISC	12
CLOCK3	SMISC	15
CLOCK6	SMISC	18
CSST3	SMISC	21
CSST6	SMISC	24
CLST3	SMISC	27
CLST6	SMISC	30
JRP3	SMISC	33
JRP6	SMISC	36
JCD3	SMISC	39
JCD6	SMISC	42
JEF3	SMISC	45
JEF6	SMISC	48
JDF3	SMISC	51
JDF6	SMISC	54
JRU3	SMISC	63
JRU6	SMISC	66
JRV3	SMISC	69
JRV6	SMISC	72
JRA3	SMISC	75
JRA6	SMISC	78
JTEMP	SMISC	79

Table 184cyl.4: MPC184 Cylindrical Joint Item and Sequence Numbers - NMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
The following output is available for all cylindrical joint elements (KEYOPT(4) = 0 and 1)		
E1X-I	NMISC	1
E1Y-I	NMISC	2
E1Z-I	NMISC	3
E2X-I	NMISC	4
E2Y-I	NMISC	5
E2Z-I	NMISC	6
E3X-I	NMISC	7
E3Y-I	NMISC	8
E3Z-I	NMISC	9
E1X-J	NMISC	10

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
E1Y-J	NMISC	11
E1Z-J	NMISC	12
E2X-J	NMISC	13
E2Y-J	NMISC	14
E2Z-J	NMISC	15
E3X-J	NMISC	16
E3Y-J	NMISC	17
E3Z-J	NMISC	18
JFX	NMISC	19
JFY	NMISC	20
JFZ	NMISC	21
JMX	NMISC	22
JMY	NMISC	23
JMZ	NMISC	24

MPC184 Cylindrical Joint Assumptions and Restrictions

- Boundary conditions cannot be applied on the nodes forming the cylindrical joint.
- Rotational degrees of freedom are activated at the nodes forming the element. When these elements are used in conjunction with solid elements, the rotational degrees of freedom must be suitably constrained. Since boundary conditions cannot be applied to the nodes of the cylindrical joint, a beam or shell element with very weak stiffness may be used with the underlying solid elements at the nodes forming the joint element to avoid any rigid body modes.
- If both stops and locks are specified, then lock specification takes precedence. That is, if the degree of freedom is locked at a given value, then it will remain locked for the rest of the analysis.
- In a nonlinear analysis, the components of relative motion are accumulated over all the substeps. It is essential that the substep size be restricted such that these rotations in a given substep are less than π for the values to be accumulated correctly.
- The element currently does not support birth or death options.
- The equation solver (**EQSLV**) must be the sparse solver or the PCG solver. The command **PCGOPT**,,,,,,ON is also required in order to use the PCG solver.
- The element coordinate system (**/PSYMB**,ESYS) is not relevant.

MPC184 Cylindrical Joint Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- No special features are allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

MPC184-Planar

Multipoint Constraint Element: Planar Joint

MP ME ST PR PRN <> <> <> PP EME MFS
Product Restrictions

MPC184 Planar Joint Element Description

The MPC184 planar joint element is a two-node element that has two relative displacement degrees of freedom and one relative rotational degree of freedom. All other relative degrees of freedom are fixed.

Figure 184plan.1: MPC184 Planar Joint Geometry

MPC184 Planar Joint Input Data

Set KEYOPT(1) = 12 to define a two-node planar joint element.

Figure 184plan.1: MPC184 Planar Joint Geometry (p. 909) shows the geometry and node locations for this element. Two nodes (I and J) define the element.

If KEYOPT(4) = 0, the element is an x-axis planar joint element with the local e_1 axis as the rotation axis and also the axis along which the distance is fixed.

If KEYOPT(4) = 1, the element is a z-axis planar joint element with the local e_3 axis as the rotation axis and also the axis along which the distance is fixed.

A local Cartesian coordinate system must be specified at the first node, I, of the element. The local coordinate system specification at the second node is optional. The local coordinate systems specified at node I and J evolve with the rotations at the respective nodes. Use the **SECJOINT** command to specify the identifiers of the local coordinate systems.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The constraints imposed on a planar joint element with local e_1 axis as the axis of rotation are described below. Similar constraint conditions are set up when the local e_3 axis is the axis of rotation. Referring to Figure 184plan.1: MPC184 Planar Joint Geometry (p. 909), the constraints imposed at any given time are as follows:

$$\mathbf{e}_1^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - E_1^I \cdot (\mathbf{X}^J - \mathbf{X}^I) = 0$$

$$\mathbf{e}_1^I \cdot \mathbf{e}_2^J - E_1^I \cdot E_2^J = 0$$

$$\mathbf{e}_1^I \cdot \mathbf{e}_3^J - E_1^I \cdot E_3^J = 0$$

The changes in the relative position of the nodes I and J are given by:

$$u_2 = \mathbf{e}_2 \cdot (\mathbf{x}^J - \mathbf{x}^I) - \mathbf{E}_2 \cdot (\mathbf{X}^J - \mathbf{X}^I)$$

$$u_3 = \mathbf{e}_3 \cdot (\mathbf{x}^J - \mathbf{x}^I) - \mathbf{E}_3 \cdot (\mathbf{X}^J - \mathbf{X}^I)$$

The change in the relative angular position between the two local coordinate systems is given by:

$$u_r = \varphi - \varphi_0 + m\pi$$

The constitutive calculations use the following definition of the joint displacement:

$$u_2 = \mathbf{e}_2 \cdot (\mathbf{x}^J - \mathbf{x}^I) - \ell_2^{ref}$$

$$u_3 = \mathbf{e}_3 \cdot (\mathbf{x}^J - \mathbf{x}^I) - \ell_3^{ref}$$

where:

ℓ_2^{ref} and ℓ_3^{ref} = reference lengths, length2 and length3, specified on the **SECDATA** command.

The constitutive calculations use the following definition of the joint rotation:

$$u_{r4}^c = \phi + m\pi - \phi_1^{ref}$$

where:

ϕ_1^{ref} = reference angle, angle1, specified on the **SECDATA** command. If this value is not specified, then φ_0 is used in place of ϕ_1^{ref}.

Other input data that are common to all joint elements (material behavior, stops and limits, locks, etc.) are described in "Joint Input Data" (p. 839) in the MPC184 element description.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MPC184 Planar Joint Input Summary

This input summary applies to the planar joint element option of MPC184: KEYOPT(1) = 12.

Nodes

I, J

Note

For a grounded joint element, specify either node I or node J in the element definition and leave the other node (the grounded node) blank.

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants

None

Material Properties

Use the JOIN label on the **TB** command to define stiffness and damping behavior. (See MPC184 Joint in the *Material Reference* for detailed information on defining joint materials.)

Surface Loads

None

Body Loads

Temperatures --

T(I), T(J)

Element Loads when KEYOPT(4) = 0

Displacements/Rotations --

UY, UZ, ROTX

Velocities --

VELY, VELZ, OMGX

Accelerations --

ACCY, ACCZ, DMGX

Element Loads when KEYOPT(4) = 1

Displacements/Rotations --

UX, UY, ROTZ

Velocities --

VELX, VELY, OMGZ

Accelerations --

ACCX, ACCY, DMGZ

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Special Features

Large deflection
Linear perturbation

KEYOPT(1)
Element behavior:

12 --
Planar joint element

KEYOPT(4)
Element configuration:

0 --
x-axis Planar joint with local 1 direction as the rotation axis.

1 --
z-axis Planar joint with local 3 direction as the rotation axis.

MPC184 Planar Joint Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 184plan.1: MPC184 Planar Joint Element Output Definitions (p. 912) and Table 184plan.2: MPC184 Planar Joint Element - NMISC Output (p. 915).

These tables use the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 184plan.1: MPC184 Planar Joint Element Output Definitions

Name	Definition	O	R
x-axis Planar Joint Element (KEYOPT(4) = 0)			
EL	Element number	-	Y
NODES	Element node numbers (I, J)	-	Y
FX	Constraint force in X direction	-	Y
MY	Constraint moment in Y direction	-	Y
MZ	Constraint moment in Z direction	-	Y
CSTOP2	Constraint force if stop is specified on DOF 2	-	Y
CSTOP3	Constraint force if stop is specified on DOF 3	-	Y
CSTOP4	Constraint moment if stop is specified on DOF 4	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
CLOCK2	Constraint force if lock is specified on DOF 2	-	Y
CLOCK3	Constraint force if lock is specified on DOF 3	-	Y
CLOCK4	Constraint moment if lock is specified on DOF 4	-	Y
CSST2	Constraint stop status on DOF 2[1]	-	Y
CLST2	Constraint lock status on DOF 2[2]	-	Y
CSST3	Constraint stop status on DOF 3[1]	-	Y
CLST3	Constraint lock status on DOF 3[2]	-	Y
CSST4	Constraint stop status on DOF 4[1]	-	Y
CLST4	Constraint lock status on DOF 4[2]	-	Y
JRP2	Joint relative position of DOF 2	-	Y
JRP3	Joint relative position of DOF 3	-	Y
JRP4	Joint relative position of DOF 4	-	Y
JCD2	Joint constitutive displacement on DOF 2	-	Y
JCD3	Joint constitutive displacement on DOF 3	-	Y
JCD4	Joint constitutive rotation on DOF 4	-	Y
JEF2	Joint elastic force in direction -2	-	Y
JEF3	Joint elastic force in direction -3	-	Y
JEF4	Joint elastic moment in direction -4	-	Y
JDF2	Joint damping force in direction -2	-	Y
JDF3	Joint damping force in direction -3	-	Y
JDF4	Joint damping moment in direction -4	-	Y
JRU2	Joint relative displacement of DOF 2	-	Y
JRU3	Joint relative displacement of DOF 3	-	Y
JRU4	Joint relative rotation of DOF 4	-	Y
JRV2	Joint relative velocity of DOF 2	-	Y
JRV3	Joint relative velocity of DOF 3	-	Y
JRV4	Joint relative rotational velocity of DOF 4	-	Y
JRA2	Joint relative acceleration of DOF 2	-	Y
JRA3	Joint relative acceleration of DOF 3	-	Y
JRA4	Joint relative rotational acceleration of DOF 4	-	Y
JTEMP	Average temperature in the element[3]	-	Y
z-axis Planar Joint Element (KEYOPT(4) = 1)			
EL	Element Number	-	Y
NODES	Element node numbers (I, J)	-	Y
FZ	Constraint Force in Z direction	-	Y
MX	Constraint Moment in X direction	-	Y
MY	Constraint Moment in Y direction	-	Y
CSTOP1	Constraint force if stop is specified on DOF 1	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
CSTOP2	Constraint force if stop is specified on DOF 2	-	Y
CSTOP6	Constraint moment if stop is specified on DOF 6	-	Y
CLOCK1	Constraint force if lock is specified on DOF 1	-	Y
CLOCK2	Constraint force if lock is specified on DOF 2	-	Y
CLOCK6	Constraint moment if lock is specified on DOF 6	-	Y
CSST1	Constraint stop status on DOF 1[1]	-	Y
CLST1	Constraint lock status on DOF 1[2]	-	Y
CSST2	Constraint stop status on DOF 2[1]	-	Y
CLST2	Constraint lock status on DOF 2[2]	-	Y
CSST6	Constraint stop status on DOF 6[1]	-	Y
CLST6	Constraint lock status on DOF 6[2]	-	Y
JRP1	Joint relative position of DOF 1	-	Y
JRP2	Joint relative position of DOF 2	-	Y
JRP6	Joint relative position of DOF 6	-	Y
JCD1	Joint constitutive displacement on DOF 1	-	Y
JCD2	Joint constitutive displacement on DOF 2	-	Y
JCD6	Joint constitutive rotation on DOF 6	-	Y
JEF1	Joint elastic force in direction -1	-	Y
JEF2	Joint elastic force in direction -2	-	Y
JEF6	Joint elastic moment in direction -6	-	Y
JDF1	Joint damping force in direction -1	-	Y
JDF2	Joint damping force in direction -2	-	Y
JDF6	Joint damping moment in direction -6	-	Y
JRU1	Joint relative displacement of DOF 1	-	Y
JRU2	Joint relative displacement of DOF 2	-	Y
JRU6	Joint relative rotation of DOF 6	-	Y
JRV1	Joint relative velocity of DOF 1	-	Y
JRV2	Joint relative velocity of DOF 2	-	Y
JRV6	Joint relative rotational velocity of DOF 6	-	Y
JRA1	Joint relative acceleration of DOF 1	-	Y
JRA2	Joint relative acceleration of DOF 2	-	Y
JRA6	Joint relative rotational acceleration of DOF 6	-	Y
JTEMP	Average temperature in the element[3]	-	Y

1. Constraint stop status:

   0 = stop not active, or deactivated
   1 = stopped at minimum value
   2 = stopped at maximum value

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Constraint lock status:

   0 = lock not active
   1 = locked at minimum value
   2 = locked at maximum value

3. Average temperature in the element when temperatures are applied on the nodes of the element using the **BF** command, or when temperature are applied on the element using the **BFE** command.

The following table shows additional non-summable miscellaneous (NMISC) output available for all forms of the planar joint element.

Note

This output is intended for use in the ANSYS Workbench program to track the evolution of local coordinate systems specified at the nodes of joint elements.

Table 184plan.2: MPC184 Planar Joint Element - NMISC Output

Name	Definition	O	R
The following output is available for all planar joint elements (KEYOPT(4) = 0 and 1)			
E1X-I, E1Y-I, E1Z-I	X, Y, Z components of the evolved e_1 axis at node I	-	Y
E2X-I, E2Y-I, E2Z-I	X, Y, Z components of the evolved e_2 axis at node I	-	Y
E3X-I, E3Y-I, E3Z-I	X, Y, Z components of the evolved e_3 axis at node I	-	Y
E1X-J, E1Y-J, E1Z-J	X, Y, Z components of the evolved e_1 axis at node J	-	Y
E2X-J, E2Y-J, E2Z-J	X, Y, Z components of the evolved e_2 axis at node J	-	Y
E3X-J, E3Y-J, E3Z-J	X, Y, Z components of the evolved e_3 axis at node J	-	Y
JFX, JFY, JFZ	Constraint forces expressed in the evolved co-ordinate system specified at node I	-	Y
JMX, JMY, JMZ	Constraint moments expressed in the evolved coordinate system specified at node I	-	Y

Table 184plan.3: MPC184 Planar Joint Item and Sequence Numbers - SMISC Items (p. 916) and Table 184plan.4: MPC184 Planar Joint Item and Sequence Numbers - NMISC Items (p. 918) list output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) for further information. The table uses the following notation:

Name
output quantity as defined in the Element Output Definitions table.

Item
predetermined Item label for **ETABLE** command

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

E

sequence number for single-valued or constant element data

Table 184plan.3: MPC184 Planar Joint Item and Sequence Numbers - SMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
x-axis Planar Joint Element (KEYOPT(4) = 0)		
FX	SMISC	1
MY	SMISC	5
MZ	SMISC	6
CSTOP2	SMISC	8
CSTOP3	SMISC	9
CSTOP4	SMISC	10
CLOCK2	SMISC	14
CLOCK3	SMISC	15
CLOCK4	SMISC	16
CSST2	SMISC	20
CSST3	SMISC	21
CSST4	SMISC	22
CLST2	SMISC	26
CLST3	SMISC	27
CLST4	SMISC	28
JRP2	SMISC	32
JRP3	SMISC	33
JRP4	SMISC	34
JCD2	SMISC	38
JCD3	SMISC	39
JCD4	SMISC	40
JEF2	SMISC	44
JEF3	SMISC	45
JEF4	SMISC	46
JDF2	SMISC	50
JDF3	SMISC	51
JDF4	SMISC	52
JRU2	SMISC	62
JRU3	SMISC	63
JRU4	SMISC	64
JRV2	SMISC	68
JRV3	SMISC	69
JRV4	SMISC	70

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
JRA2	SMISC	74
JRA3	SMISC	75
JRA4	SMISC	76
JTEMP	SMISC	79
z-axis Planar Joint Element (KEYOPT(4) = 1)		
FZ	SMISC	3
MX	SMISC	4
MY	SMISC	5
CSTOP1	SMISC	7
CSTOP2	SMISC	8
CSTOP6	SMISC	12
CLOCK1	SMISC	13
CLOCK2	SMISC	14
CLOCK6	SMISC	18
CSST1	SMISC	19
CSST2	SMISC	20
CSST6	SMISC	24
CLST1	SMISC	25
CLST2	SMISC	26
CLST6	SMISC	30
JRP1	SMISC	31
JRP2	SMISC	32
JRP6	SMISC	36
JCD1	SMISC	37
JCD2	SMISC	38
JCD6	SMISC	42
JEF1	SMISC	43
JEF2	SMISC	44
JEF6	SMISC	48
JDF1	SMISC	49
JDF2	SMISC	50
JDF6	SMISC	54
JRU1	SMISC	61
JRU2	SMISC	62
JRU6	SMISC	66
JRV1	SMISC	67
JRV2	SMISC	68

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
JRV6	SMISC	72
JRA1	SMISC	73
JRA2	SMISC	74
JRA6	SMISC	78
JTEMP	SMISC	79

Table 184plan.4: MPC184 Planar Joint Item and Sequence Numbers - NMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
The following output is available for all planar joint elements (KEYOPT(4) = 0 and 1)		
E1X-I	NMISC	1
E1Y-I	NMISC	2
E1Z-I	NMISC	3
E2X-I	NMISC	4
E2Y-I	NMISC	5
E2Z-I	NMISC	6
E3X-I	NMISC	7
E3Y-I	NMISC	8
E3Z-I	NMISC	9
E1X-J	NMISC	10
E1Y-J	NMISC	11
E1Z-J	NMISC	12
E2X-J	NMISC	13
E2Y-J	NMISC	14
E2Z-J	NMISC	15
E3X-J	NMISC	16
E3Y-J	NMISC	17
E3Z-J	NMISC	18
JFX	NMISC	19
JFY	NMISC	20
JFZ	NMISC	21
JMX	NMISC	22
JMY	NMISC	23
JMZ	NMISC	24

MPC184 Planar Joint Assumptions and Restrictions

- Boundary conditions cannot be applied on the nodes forming the planar joint.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Rotational degrees of freedom are activated at the nodes forming the element. When these elements are used in conjunction with solid elements, the rotational degrees of freedom must be suitably constrained. Since boundary conditions cannot be applied to the nodes of the planar joint, a beam or shell element with very weak stiffness may be used with the underlying solid elements at the nodes forming the joint element to avoid any rigid body modes.
- If both stops and locks are specified, then lock specification takes precedence. That is, if the degree of freedom is locked at a given value, then it will remain locked for the rest of the analysis.
- In a nonlinear analysis, the components of relative motion are accumulated over all the substeps. It is essential that the substep size be restricted such that these rotations in a given substep are less than π for the values to be accumulated correctly.
- The element currently does not support birth or death options.
- The equation solver (**EQSLV**) must be the sparse solver or the PCG solver. The command **PCGOPT**,,,,,,ON is also required in order to use the PCG solver.
- The element coordinate system (**/PSYMB**,ESYS) is not relevant.

MPC184 Planar Joint Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- No special features are allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

MPC184-Weld

Multipoint Constraint Element: Weld Joint

MP ME ST PR PRN <> <> <> PP EME MFS
Product Restrictions

MPC184 Weld Joint Element Description

The MPC184 weld joint element is a two-node element that has all relative degrees of freedom fixed.

Figure 184weld.1: MPC184 Weld Joint Geometry

MPC184 Weld Joint Input Data

Set KEYOPT(1) = 13 to define a two-node weld joint element.

Figure 184weld.1: MPC184 Weld Joint Geometry (p. 921) shows the geometry and node locations for this element. Two nodes (I and J) define the element.

A local Cartesian coordinate system must be specified at the first node, I, of the element. The local coordinate system specification at the second node is optional. The local coordinate systems specified at node I and J evolve with the rotations at the respective nodes. Use the **SECJOINT** command to specify the identifiers of the local coordinate systems.

Other input data that are common to all joint elements (material behavior, etc.) are described in "Joint Input Data" (p. 839) in the MPC184 element description.

Note

The weld joint may also be simulated by using the **CE** command. See the **CE** command description for additional details.

MPC184 Weld Joint Input Summary

This input summary applies to the weld joint element option of MPC184: KEYOPT(1) = 13.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Nodes
I, J

> **Note**
>
> For a grounded joint element, specify either node I or node J in the element definition and leave the other node (the grounded node) blank.

Degrees of Freedom
UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants
None

Material Properties
None

Surface Loads
None

Body Loads
None

Element Loads
None

Special Features

Large deflection
Linear perturbation

KEYOPT(1)
Element behavior:

13 --
Weld joint element

MPC184 Weld Joint Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 184weld.1: MPC184 Weld Joint Element Output Definitions (p. 923) and Table 184weld.2: MPC184 Weld Joint Element - NMISC Output (p. 923).

These tables use the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 184weld.1: MPC184 Weld Joint Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Element node numbers (I, J)	-	Y
FX	Constraint force in X direction	-	Y
FY	Constraint force in Y direction	-	Y
FZ	Constraint force in Z direction	-	Y
MX	Constraint moment in X direction	-	Y
MY	Constraint moment in Y direction	-	Y
MZ	Constraint moment in Z direction	-	Y

The following table shows additional non-summable miscellaneous (NMISC) output available for the weld joint element.

Note

This output is intended for use in the ANSYS Workbench program to track the evolution of local coordinate systems specified at the nodes of joint elements.

Table 184weld.2: MPC184 Weld Joint Element - NMISC Output

Name	Definition	O	R
E1X-I, E1Y-I, E1Z-I	X, Y, Z components of the evolved e_1 axis at node I	-	Y
E2X-I, E2Y-I, E2Z-I	X, Y, Z components of the evolved e_2 axis at node I	-	Y
E3X-I, E3Y-I, E3Z-I	X, Y, Z components of the evolved e_3 axis at node I	-	Y
E1X-J, E1Y-J, E1Z-J	X, Y, Z components of the evolved e_1 axis at node J	-	Y
E2X-J, E2Y-J, E2Z-J	X, Y, Z components of the evolved e_2 axis at node J	-	Y
E3X-J, E3Y-J, E3Z-J	X, Y, Z components of the evolved e_3 axis at node J	-	Y
JFX, JFY, JFZ	Constraint forces expressed in the evolved co-ordinate system specified at node I	-	Y
JMX, JMY, JMZ	Constraint moments expressed in the evolved coordinate system specified at node I	-	Y

Table 184weld.3: MPC184 Weld Joint Item and Sequence Numbers - SMISC Items (p. 924) and Table 184weld.4: MPC184 Weld Joint Item and Sequence Numbers - NMISC Items (p. 924) list output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) for further information. The table uses the following notation:

Name
output quantity as defined in the Element Output Definitions table.

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 184weld.3: MPC184 Weld Joint Item and Sequence Numbers - SMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FX	SMISC	1
FY	SMISC	2
FZ	SMISC	3
MX	SMISC	4
MY	SMISC	5
MZ	SMISC	6

Table 184weld.4: MPC184 Weld Joint Item and Sequence Numbers - NMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
E1X-I	NMISC	1
E1Y-I	NMISC	2
E1Z-I	NMISC	3
E2X-I	NMISC	4
E2Y-I	NMISC	5
E2Z-I	NMISC	6
E3X-I	NMISC	7
E3Y-I	NMISC	8
E3Z-I	NMISC	9
E1X-J	NMISC	10
E1Y-J	NMISC	11
E1Z-J	NMISC	12
E2X-J	NMISC	13
E2Y-J	NMISC	14
E2Z-J	NMISC	15
E3X-J	NMISC	16
E3Y-J	NMISC	17
E3Z-J	NMISC	18

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
JFX	NMISC	19
JFY	NMISC	20
JFZ	NMISC	21
JMX	NMISC	22
JMY	NMISC	23
JMZ	NMISC	24

MPC184 Weld Joint Assumptions and Restrictions

- Boundary conditions cannot be applied on the nodes forming the weld joint.
- Rotational degrees of freedom are activated at the nodes forming the element. When these elements are used in conjunction with solid elements, the rotational degrees of freedom must be suitably constrained. Since boundary conditions cannot be applied to the nodes of the weld joint, a beam or shell element with very weak stiffness may be used with the underlying solid elements at the nodes forming the joint element to avoid any rigid body modes.
- Stops (**SECSTOP**) and locks (**SECLOCK**) are not applicable to this element.
- In a nonlinear analysis, the components of relative motion are accumulated over all the substeps. It is essential that the substep size be restricted such that these rotations in a given substep are less than π for the values to be accumulated correctly.
- The element currently does not support birth or death options.
- The equation solver (**EQSLV**) must be the sparse solver or the PCG solver. The command **PCGOPT**,,,,,,ON is also required in order to use the PCG solver.
- The element coordinate system (**/PSYMB**,ESYS) is not relevant.

MPC184 Weld Joint Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- No special features are allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

MPC184-Orient

Multipoint Constraint Element: Orient Joint

MP ME ST PR PRN <> <> <> PP EME MFS
Product Restrictions

MPC184 Orient Joint Element Description

The MPC184 orient joint is a two-node element. In this joint, the relative rotational degrees of freedom are fixed while the displacement degrees of freedom are left free.

Figure 184orie.1: MPC184 Orient Joint Geometry

MPC184 Orient Joint Input Data

Set KEYOPT(1) = 14 to define a two-node orient joint element.

Figure 184orie.1: MPC184 Orient Joint Geometry (p. 927) shows the geometry and node locations for this element. Two nodes (I and J) define the element.

A local Cartesian coordinate system must be specified at the first node, I, of the element. The local coordinate system specification at the second node is optional. The local coordinate systems specified at node I and J evolve with the rotations at the respective nodes. Use the **SECJOINT** command to specify the identifiers of the local coordinate systems.

The constraints imposed on an orient joint element are easily described by referring to Figure 184orie.1: MPC184 Orient Joint Geometry (p. 927). At any given instant of time, the constraints imposed are as follows:

$$\mathbf{e}_2^I \cdot \mathbf{e}_3^J - \mathbf{E}_2^I \cdot \mathbf{E}_3^J = 0$$

$$\mathbf{e}_1^I \cdot \mathbf{e}_2^J - E_1^I \cdot E_2^J = 0$$

$$\mathbf{e}_1^I \cdot \mathbf{e}_3^J - E_1^I \cdot E_3^J = 0$$

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Other input data that are common to all joint elements (material behavior, etc.) are described in "Joint Input Data" (p. 839) in the MPC184 element description.

MPC184 Orient Joint Input Summary

This input summary applies to the orient joint element option of MPC184: KEYOPT(1) = 14.

Nodes
I, J

Note

For a grounded joint element, specify either node I or node J in the element definition and leave the other node (the grounded node) blank.

Degrees of Freedom
UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants
None

Material Properties
None

Surface Loads
None

Body Loads
None

Element Loads
None

Special Features

Large deflection
Linear perturbation

KEYOPT(1)
Element behavior:

14 --
Orient joint element

MPC184 Orient Joint Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 184orie.1: MPC184 Orient Joint Element Output Definitions (p. 929) and Table 184orie.2: MPC184 Orient Joint Element - NMISC Output (p. 929).

These tables use the following notation:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 184orie.1: MPC184 Orient Joint Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Element node numbers (I, J)	-	Y
MX	Constraint moment in X direction	-	Y
MY	Constraint moment in Y direction	-	Y
MZ	Constraint moment in Z direction	-	Y

The following table shows additional non-summable miscellaneous (NMISC) output available for the orient joint element.

Note

This output is intended for use in the ANSYS Workbench program to track the evolution of local coordinate systems specified at the nodes of joint elements.

Table 184orie.2: MPC184 Orient Joint Element - NMISC Output

Name	Definition	O	R
E1X-I, E1Y-I, E1Z-I	X, Y, Z components of the evolved e_1 axis at node I	-	Y
E2X-I, E2Y-I, E2Z-I	X, Y, Z components of the evolved e_2 axis at node I	-	Y
E3X-I, E3Y-I, E3Z-I	X, Y, Z components of the evolved e_3 axis at node I	-	Y
E1X-J, E1Y-J, E1Z-J	X, Y, Z components of the evolved e_1 axis at node J	-	Y
E2X-J, E2Y-J, E2Z-J	X, Y, Z components of the evolved e_2 axis at node J	-	Y
E3X-J, E3Y-J, E3Z-J	X, Y, Z components of the evolved e_3 axis at node J	-	Y
JFX, JFY, JFZ	Constraint forces expressed in the evolved co-ordinate system specified at node I	-	Y
JMX, JMY, JMZ	Constraint moments expressed in the evolved coordinate system specified at node I	-	Y

Table 184orie.3: MPC184 Orient Joint Item and Sequence Numbers - SMISC Items (p. 930) and Table 184orie.4: MPC184 Orient Joint Item and Sequence Numbers - NMISC Items (p. 930) list output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) for further information. The table uses the following notation:

Name
output quantity as defined in the Element Output Definitions table.

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 184orie.3: MPC184 Orient Joint Item and Sequence Numbers - SMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
MX	SMISC	4
MY	SMISC	5
MZ	SMISC	6

Table 184orie.4: MPC184 Orient Joint Item and Sequence Numbers - NMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
E1X-I	NMISC	1
E1Y-I	NMISC	2
E1Z-I	NMISC	3
E2X-I	NMISC	4
E2Y-I	NMISC	5
E2Z-I	NMISC	6
E3X-I	NMISC	7
E3Y-I	NMISC	8
E3Z-I	NMISC	9
E1X-J	NMISC	10
E1Y-J	NMISC	11
E1Z-J	NMISC	12
E2X-J	NMISC	13
E2Y-J	NMISC	14
E2Z-J	NMISC	15
E3X-J	NMISC	16
E3Y-J	NMISC	17
E3Z-J	NMISC	18
JFX	NMISC	19
JFY	NMISC	20
JFZ	NMISC	21

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
JMX	NMISC	22
JMY	NMISC	23
JMZ	NMISC	24

MPC184 Orient Joint Assumptions and Restrictions

- Boundary conditions cannot be applied on the nodes forming the orient joint.
- Rotational degrees of freedom are activated at the nodes forming the element. When these elements are used in conjunction with solid elements, the rotational degrees of freedom must be suitably constrained. Since boundary conditions cannot be applied to the nodes of the orient joint, a beam or shell element with very weak stiffness may be used with the underlying solid elements at the nodes forming the joint element to avoid any rigid body modes.
- Stops (**SECSTOP**) and locks (**SECLOCK**) are not applicable to this element.
- In a nonlinear analysis, the components of relative motion are accumulated over all the substeps. It is essential that the substep size be restricted such that these rotations in a given substep are less than π for the values to be accumulated correctly.
- The element currently does not support birth or death options.
- The equation solver (**EQSLV**) must be the sparse solver or the PCG solver. The command **PCGOPT**,,,,,,ON is also required in order to use the PCG solver.
- The element coordinate system (**/PSYMB**,ESYS) is not relevant.

MPC184 Orient Joint Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- No special features are allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

MPC184-Spherical

Multipoint Constraint Element: Spherical Joint

MP ME ST PR PRN <> <> <> PP EME MFS
Product Restrictions

MPC184 Spherical Joint Element Description

The MPC184 spherical joint element is a two-node element with the relative displacement degrees of freedom constrained. The relative rotational degrees of freedom are left unconstrained. These rotations cannot be controlled. The kinematic constraints are imposed using the Lagrange multiplier method.

Figure 184sphe.1: MPC184 Spherical Joint Geometry

MPC184 Spherical Joint Input Data

Set KEYOPT(1) = 15 to define a two-node spherical joint element.

Figure 184sphe.1: MPC184 Spherical Joint Geometry (p. 933) shows the geometry and node locations for this element. Two nodes define the element. The two nodes (I and J) are expected to have identical spatial locations initially. If the two nodes are not coincident, the relative positions of the two nodes are maintained.

A local Cartesian coordinate system should be specified at the first node, I, of the element. The specification of the second local coordinate system at node J is optional. If the local coordinate system is not specified at node J, the local coordinate system at node J is assumed to be the same as that at node I. Use the **SECJOINT** command to specify the identifiers of the local coordinate systems.

The constraints imposed in a spherical joint element are described below. Referring to Figure 184sphe.1: MPC184 Spherical Joint Geometry (p. 933), the constraints imposed at any given time are as follows:

$$\mathbf{e}_1^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - E_1^I \cdot (\mathbf{X}^J - \mathbf{X}^I) = 0$$

$$\mathbf{e}_2^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - E_2^I \cdot (\mathbf{X}^J - \mathbf{X}^I) = 0$$

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\mathbf{e}_3^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - E_3^I \cdot (\mathbf{X}^J - \mathbf{X}^I) = 0$$

For output purposes, the relative rotations between nodes I and J are characterized by the Cardan (or Bryant) angles as follows:

$$\phi = -\tan^{-1}\left(\frac{e_2^I \cdot e_3^J}{e_3^I \cdot e_3^J}\right)$$

$$\varphi = -\sin^{-1}(e_1^I \cdot e_3^J)$$

$$\chi = -\tan^{-1}\left(\frac{e_1^I \cdot e_2^J}{e_1^I \cdot e_1^J}\right)$$

Since the output of relative rotations is characterized by the Cardan (or Bryant) angles, the rotation around the local e_2 axis is limited to between -PI/2 to +PI/2 (see the expression for Φ above). When this rotation value reaches |PI/2|, the other two angles become indeterminate. Therefore, if the accumulated angles around an axis of rotation is greater than |PI/2|, the axis of rotation should typically be specified as the local e_1 or e_3 axis.

Since the relative rotational degrees of freedom cannot be controlled, the spherical joint element does not allow stops and locks or material behavior specifications. Other input data that are common to all joint elements are described in "Joint Input Data" (p. 839) in the MPC184 element description.

MPC184 Spherical Joint Input Summary

This input summary applies to the spherical joint element option of MPC184: KEYOPT(1) = 15.

Nodes

I, J,

Note

For a grounded spherical joint element, specify either node I or node J in the element definition and leave the other node (the grounded node) blank.

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants

None

Material Properties

None

Surface Loads

None

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Body Loads
None

Element Loads
None

Special Features

Large deflection
Linear perturbation

KEYOPT(1)
Element behavior:

15 --
Spherical joint element

MPC184 Spherical Joint Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 184sphe.1: MPC184 Spherical Joint Element Output Definitions (p. 935) and Table 184sphe.2: MPC184 Spherical Joint Element - NMISC Output (p. 936).

These tables use the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 184sphe.1: MPC184 Spherical Joint Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Element node numbers (I, J)	-	Y
FX	Constraint force in X direction	-	Y
FY	Constraint force in Y direction	-	Y
FZ	Constraint force in Z direction	-	Y
JRP4	Joint relative position of DOF 4	-	Y
JRP5	Joint relative position of DOF 5	-	Y
JRP6	Joint relative position of DOF 6	-	Y
JRU4	Joint relative rotation of DOF 4	-	Y
JRU5	Joint relative rotation of DOF 5	-	Y
JRU6	Joint relative rotation of DOF 6	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
JRV4	Joint relative rotational velocity of DOF 4	-	Y
JRV5	Joint relative rotational velocity of DOF 5	-	Y
JRV6	Joint relative rotational velocity of DOF 6	-	Y
JRA4	Joint relative rotational acceleration of DOF 4	-	Y
JRA5	Joint relative rotational acceleration of DOF 5	-	Y
JRA6	Joint relative rotational acceleration of DOF 6	-	Y

The following table shows additional non-summable miscellaneous (NMISC) output available for the spherical joint element.

Note

This output is intended for use in the ANSYS Workbench program to track the evolution of local coordinate systems specified at the nodes of joint elements.

Table 184sphe.2: MPC184 Spherical Joint Element - NMISC Output

Name	Definition	O	R
E1X-I, E1Y-I, E1Z-I	X, Y, Z components of the evolved e_1 axis at node I	-	Y
E2X-I, E2Y-I, E2Z-I	X, Y, Z components of the evolved e_2 axis at node I	-	Y
E3X-I, E3Y-I, E3Z-I	X, Y, Z components of the evolved e_3 axis at node I	-	Y
E1X-J, E1Y-J, E1Z-J	X, Y, Z components of the evolved e_1 axis at node J	-	Y
E2X-J, E2Y-J, E2Z-J	X, Y, Z components of the evolved e_2 axis at node J	-	Y
E3X-J, E3Y-J, E3Z-J	X, Y, Z components of the evolved e_3 axis at node J	-	Y
JFX, JFY, JFZ	Constraint forces expressed in the evolved co-ordinate system specified at node I	-	Y
JMX, JMY, JMZ	Constraint moments expressed in the evolved coordinate system specified at node I	-	Y

Table 184sphe.3: MPC184 Spherical Joint Item and Sequence Numbers -SMISC Items (p. 937) and Table 184sphe.4: MPC184 Spherical Joint Item and Sequence Numbers - NMISC Items (p. 937) list output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) for further information. The table uses the following notation:

Name

output quantity as defined in the Element Output Definitions table.

Item

predetermined Item label for **ETABLE** command

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

E

sequence number for single-valued or constant element data

Table 184sphe.3: MPC184 Spherical Joint Item and Sequence Numbers -SMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FX	SMISC	1
FY	SMISC	2
FZ	SMISC	3
JRP4-6	SMISC	34-36
JRU4-6	SMISC	64-66
JRV4-6	SMISC	70-72
JRA4-6	SMISC	76-78

Table 184sphe.4: MPC184 Spherical Joint Item and Sequence Numbers - NMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
E1X-I	NMISC	1
E1Y-I	NMISC	2
E1Z-I	NMISC	3
E2X-I	NMISC	4
E2Y-I	NMISC	5
E2Z-I	NMISC	6
E3X-I	NMISC	7
E3Y-I	NMISC	8
E3Z-I	NMISC	9
E1X-J	NMISC	10
E1Y-J	NMISC	11
E1Z-J	NMISC	12
E2X-J	NMISC	13
E2Y-J	NMISC	14
E2Z-J	NMISC	15
E3X-J	NMISC	16
E3Y-J	NMISC	17
E3Z-J	NMISC	18
JFX	NMISC	19
JFY	NMISC	20
JFZ	NMISC	21
JMX	NMISC	22
JMY	NMISC	23

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
JMZ	NMISC	24

MPC184 Spherical Joint Assumptions and Restrictions

- The nodes I and J should be coincident. If the nodes are not coincident, the relative positions between the two nodes are maintained.
- Boundary conditions cannot be applied on the nodes forming the spherical element.
- Rotational degrees of freedom are activated at the nodes forming the element. When these elements are used in conjunction with solid elements, the rotational degrees of freedom must be suitably constrained. Since boundary conditions cannot be applied to the nodes of the spherical joint, a beam or shell element with very weak stiffness may be used with the underlying solid elements at the nodes forming the joint element to avoid any rigid body modes.
- Stops (**SECSTOP**) and locks (**SECLOCK**) are not applicable to this element.
- In a nonlinear analysis, the components of relative motion are accumulated over all the substeps. For the values to be accumulated correctly, it is essential that the substep size be restricted such that the rotation in a given substep is less than π.
- The element currently does not support birth or death options.
- The equation solver (**EQSLV**) must be the sparse solver or the PCG solver. The command **PCGOPT**,,,,,,ON is also required in order to use the PCG solver.
- The element coordinate system (**/PSYMB**,ESYS) is not relevant.

MPC184 Spherical Joint Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- No special features are allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MPC184-General

Multipoint Constraint Element: General Joint

MP ME ST PR PRN <> <> <> PP EME MFS
Product Restrictions

MPC184 General Joint Element Description

The MPC184 general joint is a two-node element. By default, no relative degrees of freedom are fixed. However, you can specify which relative degrees of freedom need to be constrained. By specifying as many relative degrees of freedom to be constrained as needed, you can simulate different joint elements.

Figure 184gen.1: MPC184 General Joint Geometry

By default, a general joint has both displacement and rotational degrees of freedom activated at the nodes of the element. In some cases only displacement degrees of freedom are needed (as in a model consisting of only continuum elements). In such cases, you can specify a general joint with only displacement degrees of freedom activated by setting KEYOPT(4) = 1.

MPC184 General Joint Input Data

Set KEYOPT(1) = 16 to define a two-node general joint element.

Use KEYOPT(4) to specify the active degree-of-freedom set:

KEYOPT(4) = 0 (default) - both displacement and rotational degrees of freedom are activated.
KEYOPT(4) = 1 - only displacement degrees of freedom are activated.

For this element, you can specify which relative degrees of freedom need to be constrained. First, define the section type (**SECTYPE** command) for this joint. Then define the **SECJOINT** command as follows:

```
SECJ,LSYS,local cs1,local cs2  ! Defines the local coordinate systems for the joints.
SECJ,RDOF,dof1,dof2,…,dof6   ! Defines the relative DOFs to be constrained.
```

Note that the **SECJOINT** command is issued twice when the general joint is used with some degrees of freedom constrained. The first **SECJOINT** command defines the local coordinate systems for the joint. In the second **SECJOINT** command, specify as many relative degrees of freedom as needed (a maximum of 6 may be specified). The general joint element allows you to simulate different joint elements by specifying different relative degrees of freedom to be constrained. The following examples highlight the different joint elements that can be simulated:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
SECJ,RDOF,dof1,dof2,dof3,dof5,dof6  ! Simulates a revolute joint with local e_1 axis as the axis of revolute.

SECJ,RDOF,dof2,dof3,dof5,dof6   ! Simulates a cylindrical joint with local e_1 axis as the axis of rotation.

SECJ,RDOF,dof1,dof2,dof3,dof4,dof5,dof6   ! Simulates a weld joint.
```

When KEYOPT(4) = 1, the local coordinate systems specified at nodes I and J remain fixed in their initial orientation. The rotation at the nodes, if any, is ignored.

When KEYOPT(4) = 0, the local coordinate systems specified at nodes I and J are assumed to evolve with the rotations at the nodes.

For an unconstrained general joint (KEYOPT(4) = 0 or 1), the relative displacements between nodes I and J are as follows:

$$u_1 = \mathbf{e}_1^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - \mathbf{E}_1^I \cdot (\mathbf{X}^J - \mathbf{X}^I)$$

$$u_2 = \mathbf{e}_2^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - E_2^I \cdot (\mathbf{X}^J - \mathbf{X}^I)$$

$$u_3 = \mathbf{e}_3^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - E_3^I \cdot (\mathbf{X}^J - \mathbf{X}^I)$$

The relative rotations between nodes I and J are characterized by the Cardan (or Bryant) angles as follows (only if KEYOPT(4) = 0):

$$\phi = -\tan^{-1}\left(\frac{e_2^I \cdot e_3^J}{e_3^I \cdot e_3^J}\right)$$

$$\varphi = -\sin^{-1}(e_1^I \cdot e_3^J)$$

$$\chi = -\tan^{-1}\left(\frac{e_1^I \cdot e_2^J}{e_1^I \cdot e_1^J}\right)$$

The relative displacements and rotations are suitably constrained when some or all of the relative degrees of freedom are fixed.

Note that the relative angular positions for the general joint are characterized by the Cardan (or Bryant) angles. This requires that the rotations about the local e_2 axis be restricted between –PI/2 to +PI/2. Thus, the local e_2 axis should not be used to simulate the axis of rotation.

For an unconstrained general joint, the constitutive calculations use the following definitions for relative displacement:

$$\mathbf{u}_1 = \mathbf{e}_1^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - \ell_1^{ref}$$

$$\mathbf{u}_2 = \mathbf{e}_2^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - \ell_2^{ref}$$

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\mathbf{u}_3 = \mathbf{e}_3^{\mathrm{I}} \cdot (\mathbf{x}^{\mathrm{J}} - \mathbf{x}^{\mathrm{I}}) - \ell_3^{\mathrm{ref}}$$

where:

ℓ_1^{ref}, ℓ_2^{ref}, and ℓ_3^{ref} = reference lengths, length1, length2, and length3, specified on the **SECDATA** command.

The following definitions are for relative rotations:

$$u_{r4} = \phi - \phi_1^{\mathrm{ref}}$$

$$u_{r5} = \varphi - \phi_2^{\mathrm{ref}}$$

$$u_{r6} = \chi - \phi_3^{\mathrm{ref}}$$

where:

ϕ_1^{ref}, ϕ_3^{ref}, and ϕ_2^{ref} = reference angle specifications, angle1, angle2, and angle3 on the **SECDATA** command.

Other input data that are common to all joint elements (material behavior, stops and limits, locks, etc.) are described in "Joint Input Data" (p. 839) in the MPC184 element description.

MPC184 General Joint Input Summary

This input summary applies to the general joint element option of MPC184: KEYOPT(1) = 16.

Nodes

I, J

Note

For a grounded joint element, specify either node I or node J in the element definition and leave the other node (the grounded node) blank.

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ (KEYOPT(4) = 0)

UX, UY, UZ (KEYOPT(4) = 1)

Real Constants

None

Material Properties

Use the JOIN label on the **TB** command to define stiffness and damping behavior. (See MPC184 Joint in the *Material Reference* for detailed information on defining joint materials.)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface Loads
None

Body Loads

Temperatures --
T(I), T(J)

Element Loads
For an unconstrained general joint:

Displacement (KEYOPT(4) = 0 or 1) --
UX, UY, UZ

Rotation (KEYOPT(4) = 0) --
ROTX, ROTY, ROTZ

For a constrained general joint, loads are based on the free relative degrees of freedom in the joint.

Special Features

Large deflection
Linear perturbation

KEYOPT(1)
Element behavior:

16 --
General joint element

KEYOPT(4)
Element configuration:

0 --
General joint with both displacement and rotational degrees of freedom activated.

1 --
General joint with only displacement degrees of freedom activated.

MPC184 General Joint Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 184gen.1: MPC184 General Joint Element Output Definitions (p. 943) and Table 184gen.2: MPC184 General Joint Element - NMISC Output (p. 944).

These tables use the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 184gen.1: MPC184 General Joint Element Output Definitions

Name	Definition	O	R
General joint with displacement and rotation DOF (KEYOPT(4) = 0)			
EL	Element number	-	Y
NODES	Element node numbers (I, J)	-	Y
The constraint force and moment output depends on which of the relative DOFs are constrained.			
FX	Constraint force in X direction	-	Y
FY	Constraint force in Y direction	-	Y
FZ	Constraint force in Z direction	-	Y
MX	Constraint moment in X direction	-	Y
MY	Constraint moment in Y direction	-	Y
MZ	Constraint moment in Z direction	-	Y
The following output depends on which of the relative DOFs are unconstrained.			
CSTOP1-6	Constraint force/moment if stop is specified on DOFs 1-6	-	Y
CLOCK1-6	Constraint force/moment if lock is specified on DOFs 1-6	-	Y
CSST1-6	Constraint stop status on relative DOFs 1-6[1]	-	Y
CLST1-6	Constraint lock status on relative DOFs 1-6[2]	-	Y
JRP1-6	Joint relative position of DOFs 1-6	-	Y
JCD1-6	Joint constitutive displacement/rotation of DOFs 1-6	-	Y
JEF1-6	Joint elastic force/moment 1-6	-	Y
JDF1-6	Joint damping force/moment 1-6	-	Y
JRU1-6	Joint relative displacement/rotation 1-6	-	Y
JRV1-6	Joint relative velocity (or rotational velocity) 1-6	-	Y
JRA1-6	Joint relative acceleration (or rotational acceleration) 1-6	-	Y
JTEMP	Average temperature in the element[3]	-	Y
General joint with displacement DOF (KEYOPT(4) =1)			
EL	Element number	-	Y
NODES	Element node numbers (I, J)	-	Y
The constraint force and moment output depends on which of the relative DOFs are constrained.			
FX	Constraint force in X direction	-	Y
FY	Constraint force in Y direction	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
FZ	Constraint force in Z direction	-	Y
The following output depends on which of the relative DOFs are unconstrained.			
CSTOP1-3	Constraint force if stop is specified on DOFs 1-3	-	Y
CLOCK1-3	Constraint force if lock is specified on DOFs 1-3	-	Y
CSST1-3	Constraint stop status on relative DOFs 1-3[1]	-	Y
CLST1-3	Constraint lock status on relative DOFs 1-3[2]	-	Y
JRP1-3	Joint relative position of DOFs 1-3	-	Y
JCD1-3	Joint constitutive displacement of DOFs 1-3	-	Y
JEF1-3	Joint elastic force 1-3	-	Y
JDF1-3	Joint damping force 1-3	-	Y
JRU1-3	Joint relative displacement 1-3	-	Y
JRV1-3	Joint relative velocity 1-3	-	Y
JRA1-3	Joint relative acceleration 1-3	-	Y
JTEMP	Average temperature in the element[3]	-	Y

1. Constraint stop status:

   0 = stop not active, or deactivated
   1 = stopped at minimum value
   2 = stopped at maximum value

2. Constraint lock status:

   0 = lock not active
   1 = locked at minimum value
   2 = locked at maximum value

3. Average temperature in the element when temperatures are applied on the nodes of the element using the **BF** command, or when temperature are applied on the element using the **BFE** command.

The following table shows additional non-summable miscellaneous (NMISC) output available for all forms of the general joint element.

Note

This output is intended for use in the ANSYS Workbench program to track the evolution of local coordinate systems specified at the nodes of joint elements.

Table 184gen.2: MPC184 General Joint Element - NMISC Output

Name	Definition	O	R
The following output is available for all general joint elements (KEYOPT(4) = 0 and 1)			

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
E1X-I, E1Y-I, E1Z-I	X, Y, Z components of the evolved e_1 axis at node I	-	Y
E2X-I, E2Y-I, E2Z-I	X, Y, Z components of the evolved e_2 axis at node I	-	Y
E3X-I, E3Y-I, E3Z-I	X, Y, Z components of the evolved e_3 axis at node I	-	Y
E1X-J, E1Y-J, E1Z-J	X, Y, Z components of the evolved e_1 axis at node J	-	Y
E2X-J, E2Y-J, E2Z-J	X, Y, Z components of the evolved e_2 axis at node J	-	Y
E3X-J, E3Y-J, E3Z-J	X, Y, Z components of the evolved e_3 axis at node J	-	Y
JFX, JFY, JFZ	Constraint forces expressed in the evolved co-ordinate system specified at node I	-	Y
JMX, JMY, JMZ	Constraint moments expressed in the evolved coordinate system specified at node I	-	Y

Table 184gen.3: MPC184 General Joint Item and Sequence Numbers - SMISC Items (p. 945) and Table 184gen.4: MPC184 General Joint Item and Sequence Numbers - NMISC Items (p. 946) list output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) for further information. The tables use the following notation:

Name
output quantity as defined in the Element Output Definitions table.

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 184gen.3: MPC184 General Joint Item and Sequence Numbers - SMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
General joint with displacement and rotation DOF (KEYOPT(4) = 0) (Some of these values may be zero depending on which relative degrees of freedom are constrained.)		
FX	SMISC	1
FY	SMISC	2
FZ	SMISC	3
MX	SMISC	4
MY	SMISC	5
MZ	SMISC	6
CSTOP1-6	SMISC	7-12

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
CLOCK1-6	SMISC	13-18
CSST1-6	SMISC	19-24
CLST1-6	SMISC	25-30
JRP1-6	SMISC	31-36
JCD1-6	SMISC	37-42
JEF1-6	SMISC	43-48
JDF1-6	SMISC	49-54
JRU1-6	SMISC	61-66
JRV1-6	SMISC	67-72
JRA1-6	SMISC	73-78
JTEMP	SMISC	79
General joint with displacement DOF (KEYOPT(4) = 1) (Some of these values may be zero depending on which relative degrees of freedom are constrained.)		
FX	SMISC	1
FY	SMISC	2
FZ	SMISC	3
CSTOP1-3	SMISC	7-9
CLOCK1-3	SMISC	13-15
CSST1-3	SMISC	19–21
CLST1-3	SMISC	25-27
JRP1-3	SMISC	31-33
JCD1-3	SMISC	37-39
JEF1-3	SMISC	43-45
JDF1-3	SMISC	49-51
JRU1-3	SMISC	61-63
JRV1-3	SMISC	67-69
JRA1-3	SMISC	73-78
JTEMP	SMISC	79

Table 184gen.4: MPC184 General Joint Item and Sequence Numbers - NMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
The following output is available for all general joint elements (KEYOPT(4) = 0 and 1)		
E1X-I	NMISC	1
E1Y-I	NMISC	2
E1Z-I	NMISC	3
E2X-I	NMISC	4

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
E2Y-I	NMISC	5
E2Z-I	NMISC	6
E3X-I	NMISC	7
E3Y-I	NMISC	8
E3Z-I	NMISC	9
E1X-J	NMISC	10
E1Y-J	NMISC	11
E1Z-J	NMISC	12
E2X-J	NMISC	13
E2Y-J	NMISC	14
E2Z-J	NMISC	15
E3X-J	NMISC	16
E3Y-J	NMISC	17
E3Z-J	NMISC	18
JFX	NMISC	19
JFY	NMISC	20
JFZ	NMISC	21
JMX	NMISC	22
JMY	NMISC	23
JMZ	NMISC	24

MPC184 General Joint Assumptions and Restrictions

- Boundary conditions cannot be applied on the nodes forming the general joint.
- Rotational degrees of freedom are activated at the nodes forming the element. When these elements are used in conjunction with solid elements, the rotational degrees of freedom must be suitably constrained. Since boundary conditions cannot be applied to the nodes of the general joint, a beam or shell element with very weak stiffness may be used with the underlying solid elements at the nodes forming the joint element to avoid any rigid body modes.
- If both stops and locks are specified, then lock specification takes precedence. That is, if the degree of freedom is locked at a given value, then it will remain locked for the rest of the analysis.
- In a nonlinear analysis, the components of relative motion are accumulated over all the substeps. It is essential that the substep size be restricted such that these rotations in a given substep are less than π for the values to be accumulated correctly.
- The element currently does not support birth or death options.
- The equation solver (**EQSLV**) must be the sparse solver or the PCG solver. The command **PCGOPT**,,,,,,ON is also required in order to use the PCG solver.
- The element coordinate system (**/PSYMB**,ESYS) is not relevant.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MPC184 General Joint Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- No special features are allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MPC184-Screw

Multipoint Constraint Element: Screw Joint

MP ME ST PR PRN <> <> <> PP EME MFS
Product Restrictions

MPC184 Screw Joint Element Description

The MPC184 screw joint element is a two-node element which is very similar to the cylindrical joint element in construction. Whereas the cylindrical Joint element has two free relative degrees of freedom, the screw Joint has only one. In a screw joint, the "pitch" of the screw relates the relative rotation angle (around the cylindrical or screw axis) to the relative translational displacement along the axis of the screw. All other relative degrees of freedom are fixed.

Figure 184scr.1: MPC184 Screw Joint Geometry

MPC184 Screw Joint Input Data

Set KEYOPT(1) = 17 to define a two-node screw joint element.

Figure 184scr.1: MPC184 Screw Joint Geometry (p. 949) shows the geometry and node locations for this element. Two nodes (I and J) define the element.

A local Cartesian coordinate system must be specified at the first node, I, of the element. The local coordinate system specification at the second node is optional. The local coordinate systems specified at nodes I and J evolve with the rotations at the respective nodes. Use the **SECJOINT** command to specify

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the identifiers of the local coordinate systems. The e_3 axes of the local coordinate systems specified at the nodes must align to form the axis of the screw joint.

Note that the **SECJOINT** command is issued twice for the screw joint element. The first **SECJOINT** command defines the local coordinate systems for the joint. The second **SECJOINT** command specifies the screw pitch that relates the relative rotation angle to the relative translational displacement along the axis of the screw.

The basic constraints imposed in a screw joint element are described below:

$$\mathbf{e}_1^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - E_1^I \cdot (\mathbf{X}^J - \mathbf{X}^I) = 0$$

$$\mathbf{e}_2^I \cdot (\mathbf{x}^J - \mathbf{x}^I) - E_2^I \cdot (\mathbf{X}^J - \mathbf{X}^I) = 0$$

$$\mathbf{e}_2^I \cdot \mathbf{e}_3^J - \mathbf{E}_2^I \cdot \mathbf{E}_3^J = 0$$

$$\mathbf{e}_1^I \cdot \mathbf{e}_3^J - \mathbf{E}_1^I \cdot \mathbf{E}_3^J = 0$$

The constraint relating the relative rotational angle to the relative translational displacement along the axis is given by:

$$e_3^I \cdot (x^J - x^I) - E_3^I \cdot (x^J - x^I) - p(\phi - \phi_0) = 0$$

where p is the pitch of the screw defined as the ratio of relative axial displacement (length units) to the relative rotation (in radians) , φ is defined as:

$$\phi = -\tan^{-1}\left(\frac{e_1^I \cdot e_2^J}{e_1^I \cdot e_1^J}\right)$$

and φ_0 is defined in the reference configuration with the same expression given above.

The change in the relative position of the nodes I and J is given by:

$$u_3 = (\ell - \ell_0)$$

where:

$$\ell = e_3^I \cdot (x^J - x^I) - E_3^I \cdot (x^J - x^I)$$

and ℓ_0 is computed in the reference configuration using the same expression above.

The relative rotation between nodes I and J is given by:

$$\phi = -\tan^{-1}\left(\frac{e_1^I \cdot e_2^J}{e_1^I \cdot e_2^J}\right)$$

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The change in the relative angular position between the two local coordinate systems is given by:

$$u_r = \phi - \phi_0 + m\pi$$

where φ_0 is the initial angular offset between the two coordinate systems and m is an integer accounting for multiple rotations about the screw axis.

The constitutive calculations use the following definition of the joint displacement:

$$u_3^c = e_3^I \cdot (x^J - x^I) - \ell_3^{ref}$$

where:

ℓ_3^{ref} = reference length specified on **SECDATA** command.

The constitutive calculations use the following definition of the joint rotation:

$$u_{r6}^c = \phi + m\pi - \phi_3^{ref}$$

where:

ϕ_3^{ref} = reference angle, angle3, specified on the **SECDATA** command. If this value is not specified, then Φ_0 is used in place of ϕ_3^{ref}

Other input data that are common to all joint elements (material behavior, stops and limits, locks, etc.) are described in "Joint Input Data" (p. 839) in the MPC184 element description.

MPC184 Screw Joint Input Summary

This input summary applies to the screw joint element option of MPC184: KEYOPT(1) = 17.

Nodes

I, J

Note

For a grounded joint element, specify either node I or node J in the element definition and leave the other node (the grounded node) blank.

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants

None

Material Properties

Use the JOIN label on the **TB** command to define stiffness and damping. (See MPC184 Joint in the *Material Reference* for detailed information on defining joint materials.)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface Loads
None

Body Loads

Temperatures --
T(I), T(J)

Element Loads:

Displacements/Rotations --
UZ, ROTZ

Velocities --
VELZ, OMGZ

Accelerations --
ACCZ, DMGZ

Force/Moments --
FZ, MZ

Special Features

Large deflection
Linear perturbation

KEYOPT(1)
Element behavior:

17 --
Screw joint element

MPC184 Screw Joint Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 184scr.1: MPC184 Screw Joint Element Output Definitions (p. 953) and Table 184scr.2: MPC184 Screw Joint Element - NMISC Output (p. 954).

These tables use the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 184scr.1: MPC184 Screw Joint Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
NODES	Element node numbers (I, J)	-	Y
FX	Constraint Force in X direction	-	Y
FY	Constraint Force in Y direction	-	Y
MX	Constraint Moment in X direction	-	Y
MY	Constraint Moment in Y direction	-	Y
CSTOP3	Constraint force if stop is specified on DOF 3	-	Y
CSTOP6	Constraint moment if stop is specified on DOF 6	-	Y
CLOCK3	Constraint force if lock is specified on DOF 3	-	Y
CLOCK6	Constraint moment if lock is specified on DOF 6	-	Y
CSST3	Constraint stop status on DOF 3[1]	-	Y
CLST3	Constraint lock status on DOF 3[2]	-	Y
CSST6	Constraint stop status on DOF 6[1]	-	Y
CLST6	Constraint lock status on DOF 6[2]	-	Y
JRP3	Joint relative position of DOF 3	-	Y
JRP6	Joint relative position of DOF 6	-	Y
JCD3	Joint constitutive displacement on DOF 3	-	Y
JCD6	Joint constitutive rotation on DOF 6	-	Y
JEF3	Joint elastic force in direction -3	-	Y
JEF6	Joint elastic moment in direction -6	-	Y
JDF3	Joint damping force in direction -3	-	Y
JDF6	Joint damping moment in direction -6	-	Y
JRU3	Joint relative displacement of DOF 3	-	Y
JRU6	Joint relative rotation of DOF 6	-	Y
JRV3	Joint relative velocity of DOF 3	-	Y
JRV6	Joint relative rotational velocity of DOF 6	-	Y
JRA3	Joint relative acceleration of DOF 3	-	Y
JRA6	Joint relative rotational acceleration of DOF 6	-	Y
JTEMP	Average temperature in the element[3]	-	Y

1. Constraint stop status:

   0 = stop not active, or deactivated
   1 = stopped at minimum value
   2 = stopped at maximum value

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Constraint lock status:

   0 = lock not active
   1 = locked at minimum value
   2 = locked at maximum value

3. Average temperature in the element when temperatures are applied on the nodes of the element using the **BF** command, or when temperature are applied on the element using the **BFE** command.

The following table shows additional non-summable miscellaneous (NMISC) output available for all forms of the screw joint element.

Note

This output is intended for use in the ANSYS Workbench program to track the evolution of local coordinate systems specified at the nodes of joint elements.

Table 184scr.2: MPC184 Screw Joint Element - NMISC Output

Name	Definition	O	R
The following output is available for all screw joint elements (KEYOPT(4) = 0 and 1)			
E1X-I, E1Y-I, E1Z-I	X, Y, Z components of the evolved e_1 axis at node I	-	Y
E2X-I, E2Y-I, E2Z-I	X, Y, Z components of the evolved e_2 axis at node I	-	Y
E3X-I, E3Y-I, E3Z-I	X, Y, Z components of the evolved e_3 axis at node I	-	Y
E1X-J, E1Y-J, E1Z-J	X, Y, Z components of the evolved e_1 axis at node J	-	Y
E2X-J, E2Y-J, E2Z-J	X, Y, Z components of the evolved e_2 axis at node J	-	Y
E3X-J, E3Y-J, E3Z-J	X, Y, Z components of the evolved e_3 axis at node J	-	Y
JFX, JFY, JFZ	Constraint forces expressed in the evolved co-ordinate system specified at node I	-	Y
JMX, JMY, JMZ	Constraint moments expressed in the evolved coordinate system specified at node I	-	Y

Table 184scr.3: MPC184 Screw Joint Item and Sequence Numbers - SMISC Items (p. 955) and Table 184scr.4: MPC184 Screw Joint Item and Sequence Numbers - NMISC Items (p. 955) list output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) for further information. The table uses the following notation:

Name
output quantity as defined in the Element Output Definitions table.

Item
predetermined Item label for **ETABLE** command

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

E

sequence number for single-valued or constant element data

Table 184scr.3: MPC184 Screw Joint Item and Sequence Numbers - SMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FX	SMISC	1
FY	SMISC	2
MX	SMISC	4
MY	SMISC	5
CSTOP3	SMISC	9
CSTOP6	SMISC	12
CLOCK3	SMISC	15
CLOCK6	SMISC	18
CSST3	SMISC	21
CSST6	SMISC	24
CLST3	SMISC	27
CLST6	SMISC	30
JRP3	SMISC	33
JRP6	SMISC	36
JCD3	SMISC	39
JCD6	SMISC	42
JEF3	SMISC	45
JEF6	SMISC	48
JDF3	SMISC	51
JDF6	SMISC	54
JRU3	SMISC	63
JRU6	SMISC	66
JRV3	SMISC	69
JRV6	SMISC	72
JRA3	SMISC	75
JRA6	SMISC	78
JTEMP	SMISC	79

Table 184scr.4: MPC184 Screw Joint Item and Sequence Numbers - NMISC Items

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
E1X-I	NMISC	1
E1Y-I	NMISC	2
E1Z-I	NMISC	3

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
E2X-I	NMISC	4
E2Y-I	NMISC	5
E2Z-I	NMISC	6
E3X-I	NMISC	7
E3Y-I	NMISC	8
E3Z-I	NMISC	9
E1X-J	NMISC	10
E1Y-J	NMISC	11
E1Z-J	NMISC	12
E2X-J	NMISC	13
E2Y-J	NMISC	14
E2Z-J	NMISC	15
E3X-J	NMISC	16
E3Y-J	NMISC	17
E3Z-J	NMISC	18
JFX	NMISC	19
JFY	NMISC	20
JFZ	NMISC	21
JMX	NMISC	22
JMY	NMISC	23
JMZ	NMISC	24

MPC184 Screw Joint Assumptions and Restrictions

- Boundary conditions cannot be applied on the nodes forming the screw joint.
- Rotational degrees of freedom are activated at the nodes forming the element. When these elements are used in conjunction with solid elements, the rotational degrees of freedom must be suitably constrained. Since boundary conditions cannot be applied to the nodes of the screw joint, a beam or shell element with very weak stiffness may be used with the underlying solid elements at the nodes forming the joint element to avoid any rigid body modes.
- The pitch of the screw joint is defined as the ratio of relative axial displacement (length units) to relative rotation (in radians). Note that the relative rotation is expressed in radians and not as "number of revolutions." Thus, the ANSYS definition for the pitch of a screw joint differs from some commonly used definitions for pitch.
- If both stops and locks are specified, then lock specification takes precedence. That is, if the degree of freedom is locked at a given value, then it will remain locked for the rest of the analysis.
- In a nonlinear analysis, the components of relative motion are accumulated over all the substeps. It is essential that the substep size be restricted such that these rotations in a given substep are less than π for the values to be accumulated correctly.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The relative rotation and relative translation degrees of freedom for this joint are not independent. Loads or boundary conditions (applied with the **FJ** or **DJ** command) can be specified on only one of these two relative degrees of freedom, while the other relative degree of freedom is automatically defined via the constraint equations of the element.
- The element currently does not support birth or death options.
- The equation solver (**EQSLV**) must be the sparse solver.
- The element coordinate system (**/PSYMB**,ESYS) is not relevant.

MPC184 Screw Joint Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- No special features are allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID185

3-D 8-Node Structural Solid

MP ME ST PR PRN DS <> <> PP EME MFS
Product Restrictions

SOLID185 Element Description

SOLID185 is used for 3-D modeling of solid structures. It is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, hyperelasticity, stress stiffening, creep, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials.

SOLID185 is available in two forms:

- Homogeneous Structural Solid (KEYOPT(3) = 0, the default) -- See "SOLID185 Homogeneous Structural Solid Element Description" (p. 959).
- Layered Structural Solid (KEYOPT(3) = 1) -- See "SOLID185 Layered Structural Solid Element Description" (p. 966).

See SOLID185 in the *Mechanical APDL Theory Reference* for more details about this element.

A higher-order version of the SOLID185 element is SOLID186.

SOLID185 Homogeneous Structural Solid Element Description

SOLID185 Structural Solid is suitable for modeling general 3-D solid structures. It allows for prism, tetrahedral, and pyramid degenerations when used in irregular regions. Various element technologies such as B-bar, uniformly reduced integration, and enhanced strains are supported.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 185.1: SOLID185 Homogeneous Structural Solid Geometry

SOLID185 Homogeneous Structural Solid Input Data

The geometry and node locations for this element are shown in Figure 185.1: SOLID185 Homogeneous Structural Solid Geometry (p. 960). The element is defined by eight nodes and the orthotropic material properties. The default element coordinate system is along global directions. You may define an element coordinate system using **ESYS**, which forms the basis for orthotropic material directions.

Element loads are described in Nodal Loading (p. 49). Pressures may be input as surface loads on the element faces as shown by the circled numbers in Figure 185.1: SOLID185 Homogeneous Structural Solid Geometry (p. 960). Positive pressures act into the element. Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input temperature pattern, unspecified temperatures default to TUNIF. Similar defaults occurs for fluence except that zero is used instead of TUNIF.

KEYOPT(6) = 1 sets the element for using mixed formulation. For details on the use of mixed formulation, see Applications of Mixed u-P Formulations (p. 73) in the *Element Reference*.

You can apply an initial stress state to this element via the **INISTATE** command. For more information, see Initial State in the *Basic Analysis Guide*.

As described in Coordinate Systems (p. 55), you can use **ESYS** to orient the material properties and strain/stress output. Use **RSYS** to choose output that follows the material coordinate system or the global coordinate system. For the case of hyperelastic materials, the output of stress and strain is always

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

with respect to the global Cartesian coordinate system rather than following the material/element coordinate system.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetrical matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

"SOLID185 Homogeneous Structural Solid Input Summary" (p. 961) contains a summary of element input. For a general description of element input, see Element Input (p. 43).

SOLID185 Homogeneous Structural Solid Input Summary

Nodes

I, J, K, L, M, N, O, P

Degrees of Freedom

UX, UY, UZ

Real Constants

None, if KEYOPT(2) = 0,
HGSTF - Hourglass Stiffness Scaling factor if KEYOPT(2) = 1 (Default is 1.0; any positive number is valid. If set to 0.0, value is automatically reset to 1.0.)

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), GXY, GYZ, GXZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ or THSX, THSY, THSZ), DENS, ALPD, BETD

Surface Loads

Pressures --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperatures --

T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

Body force densities --

The element values in the global X, Y, and Z directions.

Special Features

Birth and death
Element technology autoselect
Initial stress import
Large deflection
Large strain
Linear perturbation
Material force evaluation
Nonlinear stabilization
Rezoning

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stress stiffening

KEYOPT(2)
Element technology:

0 --
Full integration with $\bar{B}$ method (default)

1 --
Uniform reduced integration with hourglass control

2 --
Enhanced strain formulation

3 --
Simplified enhanced strain formulation

KEYOPT(3)
Layer construction:

0 --
Structural Solid (default) -- nonlayered

1 --
Layered Solid (not applicable to SOLID185 Structural Solid)

KEYOPT(6)
Element formulation:

0 --
Use pure displacement formulation (default)

1 --
Use mixed formulation

SOLID185 Homogeneous Structural Solid Element Technology

SOLID185 Homogeneous Structural Solid uses the full-integration $\bar{B}$ method (also known as the selective reduced integration method), enhanced strain formulation, simplified enhanced strain formulation, or uniform reduced integration.

When enhanced strain formulation (KEYOPT(2) = 2) is selected, the element introduces nine *internal* (user-inaccessible) degrees of freedom to handle shear locking, and four internal degrees of freedom to handle volumetric locking.

For more information, see Element Technologies.

SOLID185 Homogeneous Structural Solid Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Additional element output as shown in Table 185.1: SOLID185 Homogeneous Structural Solid Element Output Definitions (p. 963)

Several items are illustrated in Figure 185.2: SOLID185 Homogeneous Structural Solid Stress Output (p. 963). See Element Table for Variables Identified By Sequence Number in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this document for more information.

Figure 185.2: SOLID185 Homogeneous Structural Solid Stress Output

Stress directions shown are for global directions.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 185.1: SOLID185 Homogeneous Structural Solid Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
NODES	Nodes - I, J, K, L, M, N, O, P	-	Y
MAT	Material number	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	3
PRES	Pressures P1 at nodes J, I, L, K; P2 at I, J, N, M; P3 at J, K, O, N; P4 at K, L, P, O; P5 at L, I, M, P; P6 at M, N, O, P	-	Y
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	-	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	Y	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
S:1, 2, 3	Principal stresses	-	Y
S:INT	Stress intensity	-	Y
S:EQV	Equivalent stress	-	Y
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	Y	Y
EPEL:EQV	Equivalent elastic strains [6]	-	Y
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	2	2
EPTH:EQV	Equivalent thermal strains [6]	2	2
EPPL:X, Y, Z, XY, YZ, XZ	Plastic strains [7]	1	1
EPPL:EQV	Equivalent plastic strains [6]	1	1
EPCR:X, Y, Z, XY, YZ, XZ	Creep strains	1	1
EPCR:EQV	Equivalent creep strains [6]	1	1
EPTO:X, Y, Z, XY, YZ, XZ	Total mechanical strains (EPEL + EPPL + EPCR)	Y	-
EPTO:EQV	Total equivalent mechanical strains (EPEL + EPPL + EPCR)	Y	-
NL:SEPL	Plastic yield stress	1	1
NL:EPEQ	Accumulated equivalent plastic strain	1	1
NL:CREQ	Accumulated equivalent creep strain	1	1
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	1	1
NL:HPRES	Hydrostatic pressure	1	1
SEND:ELASTIC, PLASTIC, CREEP	Strain energy densities	-	1
LOCI:X, Y, Z	Integration point locations	-	4
SVAR:1, 2, ... , N	State variables	-	5

1. Nonlinear solution, output only if the element has a nonlinear material, or if large-deflection effects are enabled (**NLGEOM**,ON) for SEND..

2. Output only if element has a thermal load.

3. Available only at centroid as a ***GET** item.

4. Available only if **OUTRES**,LOCI is used.

5. Available only if the `UserMat` subroutine and **TB**,STATE command are used.

6. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.

7. For the shape memory alloy material model, transformation strains are reported as plasticity strain EPPL.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Table 185.2: SOLID185 Homogeneous Structural Solid Item and Sequence Numbers (p. 965) lists output available via **ETABLE** using the Sequence Number method. See Element Table for Variables Identified By Sequence Number in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this document for more information. The following notation is used in Table 185.2: SOLID185 Homogeneous Structural Solid Item and Sequence Numbers (p. 965):

Name
output quantity as defined in the Table 185.1: SOLID185 Homogeneous Structural Solid Element Output Definitions (p. 963)

Item
predetermined Item label for **ETABLE** command

I,J,...,P
sequence number for data at nodes I, J, ..., P

Table 185.2: SOLID185 Homogeneous Structural Solid Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	I	J	K	L	M	N	O	P
P1	SMISC	2	1	4	3	-	-	-	-
P2	SMISC	5	6	-	-	8	7	-	-
P3	SMISC	-	9	10	-	-	12	11	-
P4	SMISC	-	-	13	14	-	-	16	15
P5	SMISC	18	-	-	17	19	-	-	20
P6	SMISC	-	-	-	-	21	22	23	24

SOLID185 Homogeneous Structural Solid Assumptions and Restrictions

- Zero-volume elements are not allowed.
- Elements may be numbered either as shown in Figure 185.1: SOLID185 Homogeneous Structural Solid Geometry (p. 960) or may have the planes IJKL and MNOP interchanged. The element may not be twisted such that the element has two separate volumes (which occurs most frequently when the elements are not numbered properly).
- All elements must have eight nodes. You can form a prism-shaped element by defining duplicate K and L and duplicate O and P node numbers. (See Degenerated Shape Elements (p. 39).) Pyramid and tetrahedral shapes are also available.
- For the degenerated shape elements where the $\bar{B}$ or enhanced strain formulations are specified, degenerated shape functions and a conventional integration scheme are used.
- If you use the mixed formulation (KEYOPT(6) = 1), the damped eigensolver is not supported. You must use the sparse solver (default).
- For modal cyclic symmetry analyses, ANSYS, Inc. recommends using enhanced strain formulation.
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). Prestress effects can be activated by the **PSTRES** command.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This element has a layered option (KEYOPT(3) = 1). See "SOLID185 Layered Structural Solid Assumptions and Restrictions" (p. 973) for additional information.

SOLID185 Homogeneous Structural Solid Product Restrictions

None.

SOLID185 Layered Structural Solid Element Description

Use SOLID185 Layered Solid to model layered thick shells or solids. The layered section definition is given by ANSYS section (SEC*xxx*) commands. A prism degeneration option is also available.

Figure 185.3: SOLID185 Layered Structural Solid Geometry

x_o = Element x-axis if ESYS is not supplied.

x = Element x-axis if ESYS is supplied.

SOLID185 Layered Structural Solid Input Data

The geometry and node locations for this element are shown in Figure 185.3: SOLID185 Layered Structural Solid Geometry (p. 966). The element is defined by eight nodes. A prism-shaped element may be formed by defining the same node numbers for nodes K and L, and O and P.

In addition to the nodes, the element input data includes the anisotropic material properties. Anisotropic material directions correspond to the layer coordinate directions which are based on the element coordinate system. The element coordinate system follows the shell convention where the z axis is normal to the surface of the shell. The nodal ordering must follow the convention that I-J-K-L and M-N-O-P element faces represent the bottom and top shell surfaces, respectively. You can change the orientation within the plane of the layers via the **ESYS** command in the same way that you would for shell elements (as described in Coordinate Systems (p. 55)). To achieve the correct nodal ordering for a volume mapped (hexahedron) mesh, you can use the **VEORIENT** command to specify the desired volume orientation before executing the **VMESH** command. Alternatively, you can use the **EORIENT** command after auto-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

matic meshing to reorient the elements to be in line with the orientation of another element, or to be as parallel as possible to a defined **ESYS** axis.

Layered Section Definition Using Section Commands

You can associate SOLID185 Layered Solid with a shell section (**SECTYPE**). The layered composite specifications (including layer thickness, material, orientation, and number of integration points through the thickness of the layer) are specified via shell section (SEC*xxx*) commands. You can use the shell section commands even with a single-layered element. ANSYS obtains the actual layer thicknesses used for element calculations by scaling the input layer thickness so that they are consistent with the thickness between the nodes. A section can be partially defined using data from a FiberSIM `.xml` file.

You can designate the number of integration points (1, 3, 5, 7, or 9) located through the thickness of each layer. Two points are located on the top and bottom surfaces respectively and the remaining points are distributed equal distance between the two points. The element requires at least two points through the entire thickness. When no shell section definition is provided, the element is treated as single-layered and uses two integration points through the thickness.

SOLID185 Layered Solid does not support real constant input for defining layer sections.

Other Input

The default orientation for this element has the S1 (shell surface coordinate) axis aligned with the first parametric direction of the element at the center of the element and is shown as x_o in Figure 185.3: SOLID185 Layered Structural Solid Geometry (p. 966).

The default first surface direction S1 can be reoriented in the element reference plane (as shown in Figure 185.3: SOLID185 Layered Structural Solid Geometry (p. 966)) via the **ESYS** command. You can further rotate S1 by angle THETA (in degrees) for each layer via the **SECDATA** command to create layer-wise coordinate systems. See Coordinate Systems (p. 55) for details.

Element loads are described in Nodal Loading (p. 49). Pressures may be input as surface loads on the element faces as shown by the circled numbers in Figure 185.3: SOLID185 Layered Structural Solid Geometry (p. 966). Positive pressures act into the element.

If you specify no element body load for defining temperatures--that is, if you define temperatures with commands other than **BFE**--SOLID185 Layered Solid adopts an element-wise temperature pattern and requires only eight temperatures for the eight element corner nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). `For any other input temperature pattern, unspecified nodal temperatures default to TUNIF. ANSYS computes all layer interface temperatures by interpolating nodal temperatures.

Alternatively, you can input temperatures as element body loads at the corners of the outside faces of the element and at the corners of the interfaces between layers. In such a case, the element uses a layer-wise pattern. Temperatures T1, T2, T3, T4 are used for the bottom of layer 1, temperatures T5, T6, T7, T8 are used for interface corners between layers 1 and 2, and so on between successive layers, ending with temperatures at the top layer NLayer. If you input exactly NLayer+1 temperatures, one temperature is used for the four bottom corners of each layer, and the last temperature is used for the four top corner temperatures of the top layer. The first corner temperature T1 defaults to TUNIF. If all other corner temperatures are unspecified, they default to T1. For any other input pattern, unspecified temperatures default to TUNIF.

KEYOPT(6) = 1 sets the element for using mixed formulation. For details on the use of mixed formulation, see Applications of Mixed u-P Formulations (p. 73) in the *Element Reference*.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can apply an initial stress state to this element via the **INISTATE** command. For more information, see Initial State in the *Basic Analysis Guide*.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetrical matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

The following table summarizes the element input. Element Input (p. 43) provides a general description of element input.

SOLID185 Layered Structural Solid Input Summary

Nodes

I, J, K, L, M, N, O, P

Degrees of Freedom

UX, UY, UZ

Real Constants

None

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ),
DENS, GXY, GYZ, GXZ, ALPD, BETD

Surface Loads

Pressures --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperatures --

T1, T2, T3, T4 at bottom of layer 1; T5, T6, T7, T8 between layers 1-2; similarly for between successive layers, ending with temperatures at top of layer NLayer (4 * (NLayer + 1) maximum)

Body force densities --

The element values in the global X, Y, and Z directions.

Special Features

Birth and death
Element technology autoselect
Initial stress import
Large deflection
Large strain
Linear perturbation
Material force evaluation
Nonlinear stabilization
Rezoning

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stress stiffening

KEYOPT(2)
Element technology:

2 --
Enhanced strain formulation

3 --
Simplified enhanced strain formulation (default)

KEYOPT(3)
Layer construction:

0 --
Structural Solid (not applicable to SOLID185 Layered Solid)

1 --
Layered Solid

KEYOPT(6)
Element formulation:

0 --
Use pure displacement formulation (default)

1 --
Use mixed formulation

KEYOPT(8)
Layer data storage:

0 --
Store data for bottom of bottom layer and top of top layer (default)

1 --
Store top and bottom data for all layers. (The volume of data may be considerable.)

SOLID185 Layered Structural Solid Element Technology

SOLID185 Layered Structural Solid uses enhanced strain formulation or simplified enhanced strain formulation.

When enhanced strain formulation (KEYOPT(2) = 2) is selected, the element introduces nine *internal* (user-inaccessible) degrees of freedom to handle shear locking, and four internal degrees of freedom to handle volumetric locking.

For more information, see Element Technologies.

SOLID185 Layered Structural Solid Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Additional element output as shown in Table 185.3: SOLID185 Layered Structural Solid Element Output Definitions (p. 970)

Several items are illustrated in Figure 185.4: SOLID185 Layered Structural Solid Stress Output (p. 970). See Filling the Element Table for Variables Identified By Sequence Number in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this document for more information.

Figure 185.4: SOLID185 Layered Structural Solid Stress Output

*Note: Layer Coordinate System x-y plane is parallel to the reference plane (KREF)

Stress directions shown are for global directions.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 185.3: SOLID185 Layered Structural Solid Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
NODES	Nodes - I, J, K, L, M, N, O, P	-	Y
MAT	Material number	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	3
PRES	Pressures P1 at nodes J, I, L, K; P2 at I, J, N, M; P3 at J, K, O, N; P4 at K, L, P, O; P5 at L, I, M, P; P6 at M, N, O, P	-	Y
TEMP	T1, T2, T3, T4 at bottom of layer 1; T5, T6, T7, T8 between layers 1-2; similarly for between successive layers, ending with temperatures at top of layer NL (4 * (NL + 1) maximum)	-	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	Y	Y
S:1, 2, 3	Principal stresses	-	Y
S:INT	Stress intensity	-	Y
S:EQV	Equivalent stress	-	Y
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	Y	Y
EPEL:EQV	Equivalent elastic strains [6]	-	Y
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	2	2
EPTH:EQV	Equivalent thermal strains [6]	2	2
EPPL:X, Y, Z, XY, YZ, XZ	Plastic strains [7]	1	1
EPPL:EQV	Equivalent plastic strains [6]	1	1
EPCR:X, Y, Z, XY, YZ, XZ	Creep strains	1	1
EPCR:EQV	Equivalent creep strains [6]	1	1
EPTO:X, Y, Z, XY, YZ, XZ	Total mechanical strains (EPEL + EPPL + EPCR)	Y	-
EPTO:EQV	Total equivalent mechanical strains (EPEL + EPPL + EPCR)	Y	-
NL:SEPL	Plastic yield stress	1	1
NL:EPEQ	Accumulated equivalent plastic strain	1	1
NL:CREQ	Accumulated equivalent creep strain	1	1
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	1	1
NL:HPRES	Hydrostatic pressure	1	1
SEND:ELASTIC, PLASTIC, CREEP	Strain energy densities	-	1
N11, N22, N12	In-plane forces (per unit length)	-	Y
M11, M22, M12	Out-of-plane moments (per unit length)	-	Y
Q13, Q23	Transverse-shear forces (per unit length)	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
LOCI:X, Y, Z	Integration point locations	-	4
SVAR:1, 2, ... , N	State variables	-	5
ILSXZ	SXZ interlaminar shear stress	-	9
ILSYZ	SYZ interlaminar shear stress	-	9
ILSUM	Magnitude of the interlaminar shear stress vector	-	8, 9
ILANG	Angle of interlaminar shear stress vector (measured from the element x-axis toward the element y-axis in degrees)	-	9

1. Nonlinear solution, output only if the element has a nonlinear material, or if large-deflection effects are enabled (**NLGEOM**,ON) for SEND.

2. Output only if element has a thermal load.

3. Available only at centroid as a ***GET** item.

4. Available only if **OUTRES**,LOCI is used.

5. Available only if the `UserMat` subroutine and **TB**,STATE command are used.

6. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.

7. For the shape memory alloy material model, transformation strains are reported as plasticity strain EPPL.

8. The components are combined as $\sigma_{il} = \sqrt{(\sigma_{xz})^2 + (\sigma_{yz})^2}$ and the largest value of σ_{il} is output as the maximum interlaminar shear stress.

9. Available only if a valid shell section (**SECTYPE**,,SHELL) is defined for the element.

Table 185.4: SOLID185 Layered Structural Solid Item and Sequence Numbers (p. 973) lists output available via **ETABLE** using the Sequence Number method. See Element Table for Variables Identified By Sequence Number in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this document for more information. The following notation is used in Table 185.4: SOLID185 Layered Structural Solid Item and Sequence Numbers (p. 973):

Name

output quantity as defined in Table 185.3: SOLID185 Layered Structural Solid Element Output Definitions (p. 970)

Item

predetermined Item label for **ETABLE** command

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

I,J,...,P
sequence number for data at nodes I, J, ..., P

Table 185.4: SOLID185 Layered Structural Solid Item and Sequence Numbers

Output Quantity Name		ETABLE and ESOL Command Input								
	Item	E	I	J	K	L	M	N	O	P
P1	SMISC	-	2	1	4	3	-	-	-	-
P2	SMISC	-	5	6	-	-	8	7	-	-
P3	SMISC	-	-	9	10	-	-	12	11	-
P4	SMISC	-	-	-	13	14	-	-	16	15
P5	SMISC	-	18	-	-	17	19	-	-	20
P6	SMISC	-	-	-	-	-	21	22	23	24
THICK	SMISC	27	-	-	-	-	-	-	-	-
N11	SMISC	28	-	-	-	-	-	-	-	-
N22	SMISC	29	-	-	-	-	-	-	-	-
N12	SMISC	30	-	-	-	-	-	-	-	-
M11	SMISC	31	-	-	-	-	-	-	-	-
M22	SMISC	32	-	-	-	-	-	-	-	-
M12	SMISC	33	-	-	-	-	-	-	-	-
Q13	SMISC	34	-	-	-	-	-	-	-	-
Q23	SMISC	35	-	-	-	-	-	-	-	-

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	Bottom of Layer i	Top of Layer NL
ILSXZ	SMISC	8 * (i - 1) + 41	8 * (NL - 1) + 42
ILSYZ	SMISC	8 * (i - 1) + 43	8 * (NL - 1) + 44
ILSUM	SMISC	8 * (i - 1) + 45	8 * (NL - 1) + 46
ILANG	SMISC	8 * (i - 1) + 47	8 * (NL - 1) + 48

SOLID185 Layered Structural Solid Assumptions and Restrictions

- Zero-volume elements are not allowed.
- Elements may be numbered either as shown in Figure 185.3: SOLID185 Layered Structural Solid Geometry (p. 966) or may have the planes IJKL and MNOP interchanged. The element may not be twisted such that the element has two separate volumes (which occurs most frequently when the elements are not numbered properly).
- All elements must have eight nodes. You can form a prism-shaped element by defining duplicate K and L and duplicate O and P node numbers. (See Degenerated Shape Elements (p. 39).)
- This element is primarily intended for conveniently modeling the in-plane effects in layered thick shells or solids. The in-plane stiffness is the average of the individual layer stiffnesses. For complicated

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

through-thickness behaviors, consider using multiple layers of homogeneous (non-layered) SOLID185 elements.

- If you use the mixed formulation (KEYOPT(6) = 1), the damped eigensolver is not supported. You must use the sparse solver (default).
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). It is ignored in geometrically linear analyses (**NLGEOM**,OFF). Prestress effects can be activated via the **PSTRES** command.
- If the material of a layer is hyperelastic, the layer orientation angle has no effect.
- To obtain more accurate transverse shear results, use multiple elements though the thickness.

SOLID185 Layered Structural Solid Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID186

3-D 20-Node Structural Solid

MP ME ST PR PRN DS <> <> PP EME MFS
Product Restrictions

SOLID186 Element Description

SOLID186 is a higher order 3-D 20-node solid element that exhibits quadratic displacement behavior. The element is defined by 20 nodes having three degrees of freedom per node: translations in the nodal x, y, and z directions. The element supports plasticity, hyperelasticity, creep, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials.

SOLID186 is available in two forms:

- Homogeneous Structural Solid (KEYOPT(3) = 0, the default) -- See "SOLID186 Homogeneous Structural Solid Element Description " (p. 975).
- Layered Structural Solid (KEYOPT(3) = 1) -- See "SOLID186 Layered Structural Solid Element Description" (p. 982).

A lower-order version of the SOLID186 element is SOLID185.

SOLID186 Homogeneous Structural Solid Element Description

SOLID186 Homogeneous Structural Solid is well suited to modeling irregular meshes (such as those produced by various CAD/CAM systems). The element may have any spatial orientation.

Various printout options are available. See SOLID186 in the *Mechanical APDL Theory Reference* for more details.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 186.1: SOLID186 Homogeneous Structural Solid Geometry

SOLID186 Homogeneous Structural Solid Input Data

The geometry, node locations, and the element coordinate system for this element are shown in Figure 186.1: SOLID186 Homogeneous Structural Solid Geometry (p. 976). A prism-shaped element may be formed by defining the same node numbers for nodes K, L, and S; nodes A and B; and nodes O, P, and W. A tetrahedral-shaped element and a pyramid-shaped element may also be formed as shown in Figure 186.1: SOLID186 Homogeneous Structural Solid Geometry (p. 976). SOLID187 is a similar, but 10-node tetrahedron element.

In addition to the nodes, the element input data includes the anisotropic material properties. Anisotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems (p. 55).

Element loads are described in Nodal Loading (p. 49). Pressures may be input as surface loads on the element faces as shown by the circled numbers on Figure 186.1: SOLID186 Homogeneous Structural Solid Geometry (p. 976). Positive pressures act into the element. Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). If all corner node temperatures are specified, each midside node temperature defaults to the average temperature of its adjacent corner nodes. For any other input temperature pattern, unspecified temperatures default to TUNIF.

As described in Coordinate Systems (p. 55), you can use **ESYS** to orient the material properties and strain/stress output. Use **RSYS** to choose output that follows the material coordinate system or the global coordinate system. For the case of hyperelastic materials, the output of stress and strain is always with respect to the global Cartesian coordinate system rather than following the material/element coordinate system.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(6) = 1 sets the element for using mixed formulation. For details on the use of mixed formulation, see Applications of Mixed u-P Formulations (p. 73) in the *Element Reference*.

You can apply an initial stress state to this element via the **INISTATE** command. For more information, see Initial State in the *Basic Analysis Guide*.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

The following table summarizes the element input. Element Input (p. 43) provides a general description of element input.

SOLID186 Homogeneous Structural Solid Input Summary

Nodes

I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B

Degrees of Freedom

UX, UY, UZ

Real Constants

None

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, EY, EZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX,THSY, THSZ), PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), DENS, GXY, GYZ, GXZ, ALPD, BETD

Surface Loads

Pressures --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),

face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperatures --

T(I), T(J),T(K), T(L), T(M), T(N), T(O), T(P), T(Q), T(R), T(S), T(T), T(U), T(V), T(W), T(X), T(Y), T(Z), T(A), T(B)

Body force densities --

The element values in the global X, Y, and Z directions.

Special Features

Birth and death
Element technology autoselect
Initial stress import
Large deflection
Large strain
Linear perturbation
Material force evaluation
Nonlinear stabilization
Stress stiffening

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(2)
Element technology:

0 --
Uniform reduced integration (default)

1 --
Full integration

KEYOPT(3)
Layer construction:

0 --
Homogeneous Structural Solid (default) -- nonlayered

1 --
Layered Structural Solid (not applicable to SOLID186 Homogeneous Structural Solid)

KEYOPT(6)
Element formulation:

0 --
Use pure displacement formulation (default)

1 --
Use mixed formulation

SOLID186 Homogeneous Structural Solid Element Technology

SOLID186 uses the uniform reduced integration method or the full integration method, as follows:

- **Uniform reduced integration method**

  Helps to prevent volumetric mesh locking in nearly incompressible cases. However, hourglass mode might propagate in the model if there are not at least two layers of elements in each direction.

- **Full integration**

  The full integration method does not cause hourglass mode, but can cause volumetric locking in nearly incompressible cases. This method is used primarily for purely linear analyses, or when the model has only one layer of elements in each direction.

SOLID186 Homogeneous Structural Solid Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 186.1: SOLID186 Homogeneous Structural Solid Element Output Definitions (p. 979)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 186.2: SOLID186 Homogeneous Structural Solid Stress Output

The element stress directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 186.1: SOLID186 Homogeneous Structural Solid Element Output Definitions

Name	Definition	O	R
EL	Element number and name	-	Y
NODES	Nodes - I, J, K, L, M, N, O, P	-	Y
MAT	Material number	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	3
PRES	Pressures P1 at nodes J, I, L, K; P2 at I, J, N, M; P3 at J, K, O, N; P4 at K, L, P, O; P5 at L, I, M, P; P6 at M, N, O, P	-	Y
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	-	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	Y	Y
S:1, 2, 3	Principal stresses	-	Y
S:INT	Stress intensity	-	Y
S:EQV	Equivalent stress	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	Y	Y
EPEL:EQV	Equivalent elastic strains [6]	Y	Y
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	2	2
EPTH:EQV	Equivalent thermal strains [6]	2	2
EPPL:X, Y, Z, XY, YZ, XZ	Plastic strains [7]	1	1
EPPL:EQV	Equivalent plastic strains [6]	1	1
EPCR:X, Y, Z, XY, YZ, XZ	Creep strains	1	1
EPCR:EQV	Equivalent creep strains [6]	1	1
EPTO:X, Y, Z, XY, YZ, XZ	Total mechanical strains (EPEL + EPPL + EPCR)	Y	-
EPTO:EQV	Total equivalent mechanical strains (EPEL + EPPL + EPCR)	Y	-
NL:SEPL	Plastic yield stress	1	1
NL:EPEQ	Accumulated equivalent plastic strain	1	1
NL:CREQ	Accumulated equivalent creep strain	1	1
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	1	1
NL:HPRES	Hydrostatic pressure	1	1
SEND:ELASTIC, PLASTIC, CREEP	Strain energy density	-	1
LOCI:X, Y, Z	Integration point locations	-	4
SVAR:1, 2, ... , N	State variables	-	5

1. Nonlinear solution, output only if the element has a nonlinear material, or if large-deflection effects are enabled (**NLGEOM**,ON) for SEND.

2. Output only if element has a thermal load

3. Available only at centroid as a ***GET** item.

4. Available only if **OUTRES**,LOCI is used.

5. Available only if the `UserMat` subroutine and **TB**,STATE command are used.

6. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.

7. For the shape memory alloy material model, transformation strains are reported as plasticity strain EPPL.

Table 186.2: SOLID186 Homogeneous Structural Solid Item and Sequence Numbers (p. 981) lists output available through **ETABLE** using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this document for more

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

information. The following notation is used in Table 186.2: SOLID186 Homogeneous Structural Solid Item and Sequence Numbers (p. 981):

Name
output quantity as defined in Table 186.1: SOLID186 Homogeneous Structural Solid Element Output Definitions (p. 979)

Item
predetermined Item label for **ETABLE**

I,J,...,B
sequence number for data at nodes I, J, ..., B

Table 186.2: SOLID186 Homogeneous Structural Solid Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input									
	Item	I	J	K	L	M	N	O	P	Q,...,B
P1	SMISC	2	1	4	3	-	-	-	-	-
P2	SMISC	5	6	-	-	8	7	-	-	-
P3	SMISC	-	9	10	-	-	12	11	-	-
P4	SMISC	-	-	13	14	-	-	16	15	-
P5	SMISC	18	-	-	17	19	-	-	20	-
P6	SMISC	-	-	-	-	21	22	23	24	-

SOLID186 Homogeneous Structural Solid Assumptions and Restrictions

- The element must not have a zero volume. Also, the element may not be twisted such that the element has two separate volumes (which occurs most frequently when the element is not numbered properly). Elements may be numbered either as shown in Figure 186.1: SOLID186 Homogeneous Structural Solid Geometry (p. 976) or may have the planes IJKL and MNOP interchanged.

- An edge with a removed midside node implies that the displacement varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information on the use of midside nodes.

- Use at least two elements in each direction to avoid hourglass mode if uniform reduced integration is used (KEYOPT(2) = 0).

- When degenerated into a tetrahedron, wedge, or pyramid element shape (see Degenerated Shape Elements (p. 39)), the corresponding degenerated shape functions are used. Degeneration to a pyramidal form should be used with caution. The element sizes, when degenerated, should be small to minimize the stress gradients. Pyramid elements are best used as filler elements or in meshing transition zones.

- For mixed formulation (KEYOPT(6) = 1), no midside nodes can be missed, and no degenerated shapes are recommended. If you use the mixed formulation, you must use the sparse solver (default).

- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). Prestress effects can be activated via the **PSTRES** command.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This element has a layered option (KEYOPT(3) = 1). See "SOLID186 Layered Structural Solid Assumptions and Restrictions" (p. 989) for additional information.

SOLID186 Homogeneous Structural Solid Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The only special feature allowed is stress stiffening.

SOLID186 Layered Structural Solid Element Description

Use SOLID186 Layered Structural Solid to model layered thick shells or solids. The layered section definition is given by ANSYS section (SEC*xxx*) commands. A prism degeneration option is also available.

Figure 186.3: SOLID186 Layered Structural Solid Geometry

x_o = Element x-axis if ESYS is not supplied.

x = Element x-axis if ESYS is supplied.

SOLID186 Layered Structural Solid Input Data

The geometry, node locations, and the element coordinate system for this element are shown in Figure 186.3: SOLID186 Layered Structural Solid Geometry (p. 982). A prism-shaped element may be formed by defining the same node numbers for nodes K, L, and S; nodes A and B; and nodes O, P, and W.

In addition to the nodes, the element input data includes the anisotropic material properties. Anisotropic material directions correspond to the layer coordinate directions which are based on the element coordinate system. The element coordinate system follows the shell convention where the z axis is normal to the surface of the shell. The nodal ordering must follow the convention that I-J-K-L and M-N-O-P

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

element faces represent the bottom and top shell surfaces, respectively. You can change the orientation within the plane of the layers via the **ESYS** command in the same way that you would for shell elements (as described in Coordinate Systems (p. 55)). To achieve the correct nodal ordering for a volume mapped (hexahedron) mesh, you can use the **VEORIENT** command to specify the desired volume orientation before executing the **VMESH** command. Alternatively, you can use the **EORIENT** command after automatic meshing to reorient the elements to be in line with the orientation of another element, or to be as parallel as possible to a defined **ESYS** axis.

Layered Section Definition Using Section Commands

You can associate SOLID186 Layered Structural Solid with a shell section (**SECTYPE**). The layered composite specifications (including layer thickness, material, orientation, and number of integration points through the thickness of the layer) are specified via shell section (SEC*xxx*) commands. You can use the shell section commands even with a single-layered element. ANSYS obtains the actual layer thicknesses used for element calculations by scaling the input layer thickness so that they are consistent with the thickness between the nodes. A section can be partially defined using data from a FiberSIM `.xml` file.

You can designate the number of integration points (1, 3, 5, 7, or 9) located through the thickness of each layer. Two points are located on the top and bottom surfaces respectively and the remaining points are distributed equal distance between the two points. The element requires at least two points through the entire thickness. When no shell section definition is provided, the element is treated as single-layered and uses two integration points through the thickness.

SOLID186 Layered Structural Solid does not support real constant input for defining layer sections.

Other Input

The default orientation for this element has the S1 (shell surface coordinate) axis aligned with the first parametric direction of the element at the center of the element and is shown as x_o in Figure 186.3: SOLID186 Layered Structural Solid Geometry (p. 982).

The default first surface direction S1 can be reoriented in the element reference plane (as shown in Figure 186.3: SOLID186 Layered Structural Solid Geometry (p. 982)) via the **ESYS** command. You can further rotate S1 by angle THETA (in degrees) for each layer via the **SECDATA** command to create layer-wise coordinate systems. See Coordinate Systems (p. 55) for details.

Element loads are described in Nodal Loading (p. 49). Pressures may be input as surface loads on the element faces as shown by the circled numbers in Figure 186.3: SOLID186 Layered Structural Solid Geometry (p. 982). Positive pressures act into the element.

If you specify no element body load for defining temperatures--that is, if you define temperatures with commands other than **BFE**--SOLID186 Layered Structural Solid adopts an element-wise temperature pattern and requires only eight temperatures for the eight element corner nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). If all corner node temperatures are specified, each midside node temperature defaults to the average temperature of its adjacent corner nodes. For any other input temperature pattern, unspecified nodal temperatures default to TUNIF. ANSYS computes all layer interface temperatures by interpolating nodal temperatures.

Alternatively, you can input temperatures as element body loads at the corners of the outside faces of the element and at the corners of the interfaces between layers. In such a case, the element uses a layer-wise pattern. Temperatures T1, T2, T3, T4 are used for the bottom of layer 1, temperatures T5, T6, T7, T8 are used for interface corners between layers 1 and 2, and so on between successive layers, ending with temperatures at the top layer NLayer. If you input exactly NLayer+1 temperatures, one temperature is used for the four bottom corners of each layer, and the last temperature is used for the

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

four top corner temperatures of the top layer. The first corner temperature T1 defaults to TUNIF. If all other corner temperatures are unspecified, they default to T1. For any other input pattern, unspecified temperatures default to TUNIF.

As described in Coordinate Systems (p. 55), you can use the **ESYS** command to orient the material properties and strain/stress output. Use **RSYS** to choose output that follows the material coordinate system or the global coordinate system. For the case of hyperelastic materials, the output of stress and strain is always with respect to the global Cartesian coordinate system rather than the material/element coordinate system.

KEYOPT(6) = 1 sets the element for using u-P mixed formulation. For details about the use of mixed formulation, see Applications of Mixed u-P Formulations (p. 73) in the *Element Reference*.

You can apply an initial stress state to this element via the **INISTATE** command. For more information, see Initial State in the *Basic Analysis Guide*.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

The following table summarizes the element input. Element Input (p. 43) provides a general description of element input.

SOLID186 Layered Structural Solid Input Summary

Nodes

I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B

Degrees of Freedom

UX, UY, UZ

Real Constants

None

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, EY, EZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX,THSY, THSZ), PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), DENS, GXY, GYZ, GXZ, ALPD, BETD

Surface Loads

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperatures --

T1, T2, T3, T4 at bottom of layer 1; T5, T6, T7, T8 between layers 1-2; similarly for between successive layers, ending with temperatures at top of layer NLayer (4 * (NLayer + 1) maximum)

Body force densities --

The element values in the global X, Y, and Z directions.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Special Features

Birth and death
Element technology autoselect
Initial stress import
Large deflection
Large strain
Linear perturbation
Material force evaluation
Nonlinear stabilization
Stress stiffening

KEYOPT(2)
Element technology:

0 --
Uniform reduced integration (default)

KEYOPT(3)
Layer construction:

0 --
Homogeneous Structural Solid (not applicable to SOLID186 Layered Structural Solid)

1 --
Layered Structural Solid

KEYOPT(6)
Element formulation:

0 --
Use pure displacement formulation (default)

1 --
Use mixed formulation

KEYOPT(8)
Layer data storage:

0 --
Store data for bottom of bottom layer and top of top layer

1 --
Store top and bottom data for all layers. (The volume of data may be excessive.)

SOLID186 Layered Structural Solid Element Technology

SOLID186 Layered Structural Solid supports only the uniform reduced integration method (KEYOPT(2) = 0), which helps to prevent volumetric mesh locking in nearly incompressible cases. However, hourglass mode might propagate in the model if there are not at least two layers of elements in each direction.

SOLID186 Layered Structural Solid Output Data

The solution output associated with the element is in two forms:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 186.3: SOLID186 Layered Structural Solid Element Output Definitions (p. 986)

Figure 186.4: SOLID186 Layered Structural Solid Stress Output

The element stress directions are parallel to the layer coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 186.3: SOLID186 Layered Structural Solid Element Output Definitions

Name	Definition	O	R
EL	Element number and name	-	Y
NODES	Nodes - I, J, K, L, M, N, O, P	-	Y
MAT	Material number	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	3
PRES	Pressures P1 at nodes J, I, L, K; P2 at I, J, N, M; P3 at J, K, O, N; P4 at K, L, P, O; P5 at L, I, M, P; P6 at M, N, O, P	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
TEMP	T1, T2, T3, T4 at bottom of layer 1; T5, T6, T7, T8 between layers 1-2; similarly for between successive layers, ending with temperatures at top of layer NL (4 * (NL + 1) maximum)	-	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	Y	Y
S:1, 2, 3	Principal stresses	-	Y
S:INT	Stress intensity	-	Y
S:EQV	Equivalent stress	-	Y
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	Y	Y
EPEL:EQV	Equivalent elastic strains [6]	Y	Y
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	2	2
EPTH:EQV	Equivalent thermal strains [6]	2	2
EPPL:X, Y, Z, XY, YZ, XZ	Plastic strains [7]	1	1
EPPL:EQV	Equivalent plastic strains [6]	1	1
EPCR:X, Y, Z, XY, YZ, XZ	Creep strains	1	1
EPCR:EQV	Equivalent creep strains [6]	1	1
EPTO:X, Y, Z, XY, YZ, XZ	Total mechanical strains (EPEL + EPPL + EPCR)	Y	-
EPTO:EQV	Total equivalent mechanical strains (EPEL + EPPL + EPCR)	Y	-
NL:SEPL	Plastic yield stress	1	1
NL:EPEQ	Accumulated equivalent plastic strain	1	1
NL:CREQ	Accumulated equivalent creep strain	1	1
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	1	1
NL:HPRES	Hydrostatic pressure	1	1
SEND:ELASTIC, PLASTIC, CREEP	Strain energy density	-	1
N11, N22, N12	In-plane forces (per unit length)	-	Y
M11, M22, M12	Out-of-plane moments (per unit length)	-	Y
Q13, Q23	Transverse-shear forces (per unit length)	-	Y
LOCI:X, Y, Z	Integration point locations	-	4
SVAR:1, 2, ... , N	State variables	-	5
ILSXZ	SXZ interlaminar shear stress	-	8
ILSYZ	SYZ interlaminar shear stress	-	8
ILSUM	Interlaminar shear stress vector sum	-	8

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
ILANG	Angle of interlaminar shear stress vector (measured from the element x-axis toward the element y-axis in degrees)	-	8

1. Nonlinear solution, output only if the element has a nonlinear material, or if large-deflection effects are enabled (**NLGEOM**,ON) for SEND.

2. Output only if element has a thermal load.

3. Available only at centroid as a ***GET** item.

4. Available only if **OUTRES**,LOCI is used.

5. Available only if the `UserMat` subroutine and **TB**,STATE command are used.

6. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.

7. For the shape memory alloy material model, transformation strains are reported as plasticity strain EPPL.

8. Available only if a valid shell section (**SECTYPE**,,SHELL) is defined for the element.

Table 186.4: SOLID186 Layered Structural Solid Item and Sequence Numbers (p. 988) lists output available via **ETABLE** using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this document for more information. The following notation is used in Table 186.4: SOLID186 Layered Structural Solid Item and Sequence Numbers (p. 988):

Name
output quantity as defined in Table 186.3: SOLID186 Layered Structural Solid Element Output Definitions (p. 986)

Item
predetermined Item label for **ETABLE**

I,J,...,B
sequence number for data at nodes I, J, ..., B

Table 186.4: SOLID186 Layered Structural Solid Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input										
	Item	E	I	J	K	L	M	N	O	P	Q,...,B
P1	SMISC	-	2	1	4	3	-	-	-	-	-
P2	SMISC	-	5	6	-	-	8	7	-	-	-
P3	SMISC	-	-	9	10	-	-	12	11	-	-
P4	SMISC	-	-	-	13	14	-	-	16	15	-
P5	SMISC	-	18	-	-	17	19	-	-	20	-
P6	SMISC	-	-	-	-	-	21	22	23	24	-
THICK	SMISC	27	-	-	-	-	-	-	-	-	-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input										
	Item	E	I	J	K	L	M	N	O	P	Q,...,B
N11	SMISC	28	-	-	-	-	-	-	-	-	-
N22	SMISC	29	-	-	-	-	-	-	-	-	-
N12	SMISC	30	-	-	-	-	-	-	-	-	-
M11	SMISC	31	-	-	-	-	-	-	-	-	-
M22	SMISC	32	-	-	-	-	-	-	-	-	-
M12	SMISC	33	-	-	-	-	-	-	-	-	-
Q13	SMISC	34	-	-	-	-	-	-	-	-	-
Q23	SMISC	35	-	-	-	-	-	-	-	-	-

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	Bottom of Layer i	Top of Layer NL
ILSXZ	SMISC	8 * (i - 1) + 41	8 * (NL - 1) + 42
ILSYZ	SMISC	8 * (i - 1) + 43	8 * (NL - 1) + 44
ILSUM	SMISC	8 * (i - 1) + 45	8 * (NL - 1) + 46
ILANG	SMISC	8 * (i - 1) + 47	8 * (NL - 1) + 48

SOLID186 Layered Structural Solid Assumptions and Restrictions

- The element must not have a zero volume. Also, the element may not be twisted such that the element has two separate volumes (which occurs most frequently when the element is not numbered properly). Elements may be numbered either as shown in Figure 186.3: SOLID186 Layered Structural Solid Geometry (p. 982) or may have the planes IJKL and MNOP interchanged.

- An edge with a removed midside node implies that the displacement varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information on the use of midside nodes.

- This element is primarily intended for conveniently modeling the in-plane effects in layered thick shells or solids. The in-plane stiffness is the average of the individual layer stiffnesses. For complicated through-thickness behaviors, consider using multiple layers of homogeneous (non-layered) SOLID186 elements.

- Use at least two elements in each direction to avoid hourglass mode.

- When degenerated into a wedge element shape (see Degenerated Shape Elements (p. 39)), the corresponding degenerated shape functions are used. The element sizes, when degenerated, should be small to minimize the stress gradients.

- For mixed formulation (KEYOPT(6) = 1), no midside nodes can be missed, and no degenerated shapes are recommended. If you use the mixed formulation, you must use the sparse solver (default).

- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). It is ignored in geometrically linear analyses (**NLGEOM**,OFF). Prestress effects can be activated via the **PSTRES** command.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If the material of a layer is hyperelastic, the layer orientation angle has no effect.
- To obtain more accurate transverse shear results, use multiple elements though the thickness.

SOLID186 Layered Structural Solid Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The only special feature allowed is stress stiffening.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID187

3-D 10-Node Tetrahedral Structural Solid

MP ME ST PR PRN DS <> <> PP EME MFS
Product Restrictions

SOLID187 Element Description

SOLID187 element is a higher order 3-D, 10-node element. SOLID187 has a quadratic displacement behavior and is well suited to modeling irregular meshes (such as those produced from various CAD/CAM systems).

The element is defined by 10 nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, hyperelasticity, creep, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials. See SOLID187 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 187.1: SOLID187 Geometry

SOLID187 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 187.1: SOLID187 Geometry (p. 991).

In addition to the nodes, the element input data includes the orthotropic or anisotropic material properties. Orthotropic and anisotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Linear Material Properties in the *Material Reference*.

Element loads are described in Nodal Loading (p. 49). Pressures may be input as surface loads on the element faces as shown by the circled numbers on Figure 187.1: SOLID187 Geometry (p. 991). Positive pressures act into the element. Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). If all corner node temperatures are specified, each midside node temperature defaults to the average temperature of its adjacent corner nodes. For any other input temperature pattern, unspecified temperatures default to TUNIF.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

As described in Coordinate Systems (p. 55), you can use **ESYS** to orient the material properties and strain/stress output. Use **RSYS** to choose output that follows the material coordinate system or the global coordinate system. For the case of hyperelastic materials, the output of stress and strain is always with respect to the global Cartesian coordinate system rather than following the material/element coordinate system.

KEYOPT(6) = 1 or 2 sets the element for using mixed formulation. For details on the use of mixed formulation, see Applications of Mixed u-P Formulations (p. 73) in the *Element Reference*.

You can apply an initial stress state to this element via the **INISTATE** command. For more information, see the **INISTATE** command, and also Initial Stress Loading in the *Basic Analysis Guide*.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

The next table summarizes the element input. Element Input (p. 43) gives a general description of element input.

SOLID187 Input Summary

Nodes

I, J, K, L, M, N, O, P, Q, R

Degrees of Freedom

UX, UY, UZ

Real Constants

None

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, EY, EZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ), PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), DENS, GXY, GYZ, GXZ, ALPD, BETD

Surface Loads

Pressures --

face 1 (J-I-K), face 2 (I-J-L), face 3 (J-K-L), face 4 (K-I-L)

Body Loads

Temperatures --

T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P), T(Q), T(R)

Body force densities --

The element values in the global X, Y, and Z directions.

Special Features

Birth and death
Element technology autoselect
Initial stress import
Large deflection
Large strain

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Linear perturbation
Material force evaluation
Nonlinear stabilization
Stress stiffening

KEYOPT(6)

Element formulation:

0 --

Use pure displacement formulation (default)

1 --

Use mixed formulation, hydrostatic pressure is constant in an element (recommended for hyperelastic materials)

2 --

Use mixed formulation, hydrostatic pressure is interpolated linearly in an element (recommended for nearly incompressible elastoplastic materials)

SOLID187 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 187.1: SOLID187 Element Output Definitions (p. 994)

Several items are illustrated in Figure 187.2: SOLID187 Stress Output (p. 993). The element stress directions are parallel to the element coordinate system. The surface stress outputs are in the surface coordinate system and are available for any face (KEYOPT(6)). The coordinate system for face JIK is shown in Figure 187.2: SOLID187 Stress Output (p. 993). The other surface coordinate systems follow similar orientations as indicated by the pressure face node description. Surface stress printout is valid only if the conditions described in Element Solution (p. 51) are met. A general description of solution output is given in The Item and Sequence Number Table (p. 52). See the *Basic Analysis Guide* for ways to view results.

Figure 187.2: SOLID187 Stress Output

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 187.1: SOLID187 Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
NODES	Nodes - I, J, K, L	-	Y
MAT	Material number	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	3
PRES	Pressures P1 at nodes J, I, K; P2 at I, J, L; P3 at J, K, L; P4 at K, I, L	-	Y
TEMP	Temperatures T(I), T(J), T(K), T(L)	-	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	Y	Y
S:1, 2, 3	Principal stresses	-	Y
S:INT	Stress intensity	-	Y
S:EQV	Equivalent stress	-	Y
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	Y	Y
EPEL:EQV	Equivalent elastic strains [6]	-	Y
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	1	1
EPTH: EQV	Equivalent thermal strains [6]	1	1
EPPL:X, Y, Z, XY, YZ, XZ	Plastic strains [7]	1	1
EPPL:EQV	Equivalent plastic strains [6]	1	1
EPCR:X, Y, Z, XY, YZ, XZ	Creep strains	1	1
EPCR:EQV	Equivalent creep strains [6]	1	1
EPTO:X, Y, Z, XY, YZ, XZ	Total mechanical strains (EPEL + EPPL + EPCR)	Y	-
EPTO:EQV	Total equivalent mechanical strains (EPEL + EPPL + EPCR)	Y	-
NL:SEPL	Plastic yield stress	1	1
NL:EPEQ	Accumulated equivalent plastic strain	1	1
NL:CREQ	Accumulated equivalent creep strain	1	1
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	1	1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
NL:HPRES	Hydrostatic pressure	1	1
SEND: ELASTIC, PLASTIC, CREEP	Strain energy density	-	1
LOCI:X, Y, Z	Integration point locations	-	4
SVAR:1, 2, ... , N	State variables	-	5

1. Nonlinear solution, output only if the element has a nonlinear material, or if large-deflection effects are enabled (**NLGEOM**,ON) for SEND.

2. Output only if element has a thermal load

3. Available only at centroid as a ***GET** item.

4. Available only if **OUTRES**,LOCI is used.

5. Available only if the `UserMat` subroutine and **TB**,STATE command are used.

6. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.

7. For the shape memory alloy material model, transformation strains are reported as plasticity strain EPPL.

Table 187.2: SOLID187 Item and Sequence Numbers (p. 995) lists output available through **ETABLE** using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 187.2: SOLID187 Item and Sequence Numbers (p. 995):

Name
output quantity as defined in Table 187.1: SOLID187 Element Output Definitions (p. 994)

Item
predetermined Item label for **ETABLE** command

I,J,...,R
sequence number for data at nodes I, J, ..., R

Table 187.2: SOLID187 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	I	J	K	L	M,...,R
P1	SMISC	2	1	3	-	-
P2	SMISC	4	5	-	6	-
P3	SMISC	-	7	8	9	-
P4	SMISC	11	-	10	12	-

SOLID187 Assumptions and Restrictions

- The element must not have a zero volume.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Elements may be numbered either as shown in Figure 187.1: SOLID187 Geometry (p. 991) or may have node L below the I, J, K plane.
- An edge with a removed midside node implies that the displacement varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for information about using midside nodes.
- When mixed formulation is used (KEYOPT(6) = 1 or 2), no midside nodes can be missed.
- If you use the mixed formulation (KEYOPT(6) = 1 or 2), the damped eigensolver is not supported. You must use the sparse solver (default).
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). Prestress effects can be activated by the **PSTRES** command.

SOLID187 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The only special feature allowed is stress stiffening.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BEAM188

3-D 2-Node Beam

MP ME ST PR PRN DS <> <> PP EME MFS
Product Restrictions

BEAM188 Element Description

BEAM188 is suitable for analyzing slender to moderately stubby/thick beam structures. The element is based on Timoshenko beam theory which includes shear-deformation effects. The element provides options for unrestrained warping and restrained warping of cross-sections.

The element is a linear, quadratic, or cubic two-node beam element in 3-D. BEAM188 has six or seven degrees of freedom at each node. These include translations in the x, y, and z directions and rotations about the x, y, and z directions. A seventh degree of freedom (warping magnitude) is optional. This element is well-suited for linear, large rotation, and/or large strain nonlinear applications.

The element includes stress stiffness terms, by default, in any analysis with large deflection. The provided stress-stiffness terms enable the elements to analyze flexural, lateral, and torsional stability problems (using eigenvalue buckling, or collapse studies with arc length methods or nonlinear stabilization).

Elasticity, plasticity, creep and other nonlinear material models are supported. A cross-section associated with this element type can be a built-up section referencing more than one material. Added mass, hydrodynamic added mass and loading, and buoyant loading are available.

For more detailed information about this element, see BEAM188 - 3-D 2-Node Beam in the *Mechanical APDL Theory Reference*.

Figure 188.1: BEAM188 Geometry

BEAM188 Element Technology and Usage Recommendations

BEAM188 is based on Timoshenko beam theory, which is a first-order shear-deformation theory: transverse-shear strain is constant through the cross-section (that is, cross-sections remain plane and undistorted after deformation).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The element can be used for slender or stout beams. Due to the limitations of first-order shear-deformation theory, slender to moderately thick beams can be analyzed. Use the slenderness ratio of a beam structure (GAL^2 / (EI)) to judge the applicability of the element, where:

G
Shear modulus

A
Area of the cross-section

L
Length of the member (not the element length)

EI
Flexural rigidity

Calculate the ratio using some global distance measures, rather than basing it upon individual element dimensions. The following illustration shows an estimate of transverse-shear deformation in a cantilever beam subjected to a tip load. Although the results cannot be extrapolated to any other application, the example serves well as a general guideline. A slenderness ratio greater than 30 is recommended.

Figure 188.2: Transverse-Shear Deformation Estimation

Slenderness Ratio (GAL^2/(EI))	δ Timoshenko / δ Euler-Bernoulli
25	1.120
50	1.060
100	1.030
1000	1.003

The element supports an elastic relationship between transverse-shear forces and transverse-shear strains. You can override default values of transverse-shear stiffnesses via the **SECCONTROL** command.

BEAM188 does not use higher-order theories to account for variation in distribution of shear stresses. Use solid elements if such effects must be considered.

BEAM188 supports “restrained warping” analysis by making available a seventh degree of freedom at each beam node. By default, BEAM188 elements assume that the warping of a cross-section is small enough that it can be neglected (KEYOPT(1) = 0). You can activate the warping degree of freedom by using KEYOPT(1) = 1. With the warping degree of freedom activated, each node has seven degrees of freedom: UX, UY, UZ, ROTX, ROTY, ROTZ, and WARP. With KEYOPT(1) = 1, bimoment and bicurvature are output.

When KEYOPT(3) = 0 (linear, default), BEAM188 is based on linear shape functions. It uses one point of integration along the length; therefore, all element solution quantities are constant along the length. For example, when SMISC quantities are requested at nodes I and J, the centroidal values are reported

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

for both end nodes. This option is recommended if the element is used as stiffener and it is necessary to maintain compatibility with a first-order shell element (such as SHELL181). Only constant bending moments can be represented exactly with this option. Mesh refinement is generally required in typical applications.

When KEYOPT(3) = 2 (quadratic), BEAM188 has an internal node in the interpolation scheme, effectively making this a beam element based on quadratic shape functions. Two points of integration are used, resulting in linear variation of element solution quantities along the length. Linearly varying bending moments are represented exactly.

When KEYOPT(3) = 3 (cubic), BEAM188 has two internal nodes and adopts cubic shape functions. Quadratically varying bending moments are represented exactly. Three points of integration along the length are used, resulting in quadratic variation of element solution quantities along the length. Unlike typical cubic (Hermitian) formulations, cubic interpolation is used for all displacements and rotations.

Quadratic and cubic options are recommended when higher-order element interpolations are desired in situations where:

- The element is associated with tapered cross-sections.
- Nonuniform loads (including tapered distributed loads) exist within the element; in this case, the cubic option gives superior results over the quadratic option.

  (For partially distributed loads and non-nodal point loads, only the cubic option is valid.)
- The element may undergo highly nonuniform deformation (for example, when individual frame members in civil engineering structures are modeled with single elements).

In practice, when two elements with “restrained warping” come together at a sharp angle, you need to couple the displacements and rotations, but leave the out-of-plane warping decoupled. This is normally accomplished by having two nodes at a physical location and using appropriate constraints. This process is made easier (or automated) by the **ENDRELEASE** command, which decouples the out-of plane warping for any adjacent elements with cross-sections intersecting at an angle greater than 20 degrees.

BEAM188 allows change in cross-sectional inertia properties as a function of axial elongation. By default, the cross-sectional area changes such that the volume of the element is preserved after deformation. The default is suitable for elastoplastic applications. By using KEYOPT(2), you can choose to keep the cross-section constant or rigid. Scaling is not an option for nonlinear general beam sections (**SECTYPE**,,GENB) or for its effect on ocean loading.

Two limitations are associated with the quadratic and cubic options in BEAM188:

- Although the elements employ higher-order interpolations, the initial geometry of BEAM188 is treated as straight.
- Because the internal nodes are inaccessible, no boundary/loading/initial conditions are allowed on these internal nodes.

As a result of the limitations associated with the quadratic and cubic options, you will notice discrepancies in the results between BEAM189 and the quadratic option of BEAM188 if the midside nodes of the BEAM189 model have specified boundary/loading/initial conditions and/or the midside nodes are not located exactly at the element midpoint. Similarly, the cubic option of BEAM188 may not be identical to a traditional cubic (Hermitian) beam element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For the mass matrix and evaluation of consistent load vectors, a higher order integration rule than that used for stiffness matrix is employed. The elements support both consistent and lumped mass matrices. Use **LUMPM**,ON to activate lumped mass matrix. Consistent mass matrix is used by default. You can add mass per unit length via the **SECCONTROL** command's *ADDMAS* values. See "BEAM188 Input Summary" (p. 1004).

When ocean loading is applied, the loading is nonlinear (that is, based on the square of the relative velocity between the structure and the water). Accordingly, the full Newton-Raphson option (**NROPT**,FULL) may be necessary to achieve optimal results. (Full Newton-Raphson is applied automatically in an analysis involving large-deflection effects [**NLGEOM**,ON].)

The St. Venant warping functions for torsional behavior are determined in the undeformed state, and are used to define shear strain even after yielding. No options are available for recalculating in deformed configuration the torsional shear distribution on cross-sections during the analysis and possible partial plastic yielding of cross-sections. As such, large inelastic deformation due to torsional loading should be treated and verified with caution. Under such circumstances, alternative modeling using solid or shell elements is recommended.

BEAM188 Input Data

The geometry, node locations, coordinate system, and pressure directions for this element are shown in Figure 188.1: BEAM188 Geometry (p. 997). BEAM188 is defined by nodes I and J in the global coordinate system.

Node K is a preferred way to define the orientation of the element. For information about orientation nodes and beam meshing, see Generating a Beam Mesh With Orientation Nodes in the *Modeling and Meshing Guide*. See the **LMESH** and **LATT** command descriptions for details on generating the K node automatically.

BEAM188 can also be defined without the orientation node K. In this case, the element x-axis is oriented from node I (end 1) toward node J (end 2). If no orientation node is used, the default orientation of the element y-axis is automatically calculated to be parallel to the global X-Y plane. For the case where the element is parallel to the global Z-axis (or within a 0.01 percent slope of it), the element y-axis is oriented parallel to the global Y-axis (as shown). To control the element orientation about the element x-axis, use the orientation-node option. If both are defined, the orientation-node option takes precedence. The orientation node K, if used, defines a plane (with I and J) containing the element x and z-axes (as shown). If using this element in a large-deflection analysis, be aware that the location of the orientation node K is used only to *initially* orient the element.

The number of degrees of freedom depends on the value of KEYOPT(1). When KEYOPT(1) = 0 (the default), six degrees of freedom occur at each node. These include translations in the x, y, and z directions and rotations about the x, y, and z directions. When KEYOPT(1) = 1, a seventh degree of freedom (warping magnitude) is also considered.

The beam element is a one-dimensional line element in space. The cross-section details are provided separately via the **SECTYPE** and **SECDATA** commands. (See Beam Analysis and cross-sections in the *Structural Analysis Guide* for details). A section is associated with the beam elements by specifying the section ID number (**SECNUM**). A section number is an independent element attribute. In addition to a constant cross-section, you can also define a tapered cross-section by using the TAPER option on the **SECTYPE** command (see Defining a Tapered Beam).

BEAM188 ignores any real constant data. See the **SECCONTROL** command for defining the transverse-shear stiffness and added mass.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A summary of the element input is given in "BEAM188 Input Summary" (p. 1004).

BEAM188 Cross-Sections

BEAM188 can be associated with these cross-section types:

- Standard library section types or user meshes which define the geometry of the beam cross-section (**SECTYPE**,,BEAM). The material of the beam is defined either as an element attribute (MAT), or as part of section buildup (for multi-material cross-sections).
- Generalized beam cross-sections (**SECTYPE**,,GENB), where the relationships of generalized stresses to generalized strains are input directly.
- Tapered beam cross-sections (**SECTYPE**,,TAPER), where a standard library section or user mesh defines each end of the beam.

Standard Library Sections

BEAM188 elements are provided with section-relevant quantities (area of integration, position, etc.) automatically at a number of section points using **SECTYPE** and **SECDATA**. Each section is assumed to be an assembly of a predetermined number of nine-node cells. Each cross-section cell has four integration points and each can be associated with an independent material type.

Figure 188.3: Cross-Section Cells

The number of cells in the cross-sections influences the accuracy of section properties and ability to model nonlinear stress-strain relationship through the cross-section. The element has a nested structure of integration (along the length and in the cross-section).

When the material associated with the elements has inelastic behavior or when the temperature varies across the section, constitutive calculations are performed at the section integration points. For more common elastic applications, the element uses precalculated properties of the section at the element

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

integration points; however, the stresses and strains are calculated in the output pass at the section integration points. Element output is available at the integration points, as well as values extrapolated to the element and section nodes.

If the section is assigned the subtype ASEC, only the generalized stresses and strains (axial force, bending moments, transverse shears, curvatures, and shear strains) are available for output. 3-D contour plots and deformed shapes are not available. The ASEC subtype is displayed only as a thin rectangle to verify beam orientation.

BEAM188 is helpful for analyzing built-up beams (that is, those fabricated of two or more pieces of material joined together to form a single, solid beam). The pieces are assumed to be perfectly bonded together; therefore, the beam behaves as a single member.

The multi-material cross-section capability is applicable only where the assumptions of a beam behavior (Timoshenko or Bernoulli-Euler beam theory) holds.

In other words, what is supported is a simple extension of a conventional Timoshenko beam theory. It can be used in applications such as:

- Bimetallic strips
- Beams with metallic reinforcement
- Sensors where layers of a different material has been deposited

BEAM188 does not account for coupling of bending and twisting at the section stiffness level. The transverse shears are also treated in an uncoupled manner. This can have a significant effect on layered composite and sandwich beams if the lay-up is unbalanced.

Always validate the application of BEAM188, either with experiments or other numerical analysis. Use the restrained warping option with built-up sections after due verification.

KEYOPT(15) specifies the format of the `.rst` results file. For KEYOPT(15) = 0, the format gives only one averaged result at each section corner node; therefore, this option typically applies to homogeneous sections. For KEYOPT(15) = 1, the format gives one result for each section integration point; therefore, this option typically applies to built-up sections with multiple materials (and generates a larger results file).

Generalized Beam Cross-Sections

When using nonlinear general beam sections, neither the geometric properties nor the material is explicitly specified. *Generalized stress* implies the axial force, bending moments, torque, and transverse-shear forces. Similarly, *generalized strain* implies the axial strain, bending curvatures, twisting curvature, and transverse-shear strains. (For more information, see nonlinear general beam sections.) This is an abstract method for representing cross-section behavior; therefore, input often consists of experimental data or the results of other analyses.

Generally, BEAM188 supports an elastic relationship between transverse-shear forces and transverse-shear strains. You can override default values of transverse-shear stiffnesses via the **SECCONTROL** command.

When the beam element is associated with a generalized beam (**SECTYPE**,,GENB) cross-section type, the relationship of transverse-shear force to the transverse-shear strain can be nonlinear elastic or plastic, an especially useful capability when flexible spot welds are modeled. In such a case, the **SECCONTROL** command does not apply.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tapered Beam Cross-Sections

A linearly tapered beam is defined by specifying a standard library section or user mesh at each end of the beam. The section geometries are specified at global coordinates, then linear interpolated and evaluated at the element. The sections at the end points must be topologically identical. (For more information, see Defining a Tapered Beam.)

BEAM188 Loads

Forces are applied at the nodes (which also define the element x-axis). If the centroidal axis is not colinear with the element x-axis, applied axial forces will cause bending. Applied shear forces cause torsional strains and moment if the centroid and shear center of the cross-section are different. The nodes should therefore be located at the desired points where you want to apply the forces. Use the *OFFSETY* and *OFFSETZ* arguments of the **SECOFFSET** command appropriately. By default, the program uses the centroid as the reference axis for the beam elements.

Element loads are described in Nodal Loading (p. 49). Pressures can be input as surface loads on the element faces as shown by the circled numbers in Figure 188.1: BEAM188 Geometry (p. 997). Positive normal pressures act into the element. Lateral pressures are input as force per unit length. End "pressures" are input as forces.

At both ends of the element, temperatures can be input at these locations:

- At the element x-axis (T(0,0))
- At one unit from the x-axis in the element y-direction (T(1,0))
- At one unit from the x-axis in the element z-direction (T(0,1))

Element locations (T(y,z)) are given according to the convention used in Figure 188.1: BEAM188 Geometry (p. 997).

For beam elements, element body load commands (**BFE**) accept an element number and a list of values, 1 through 6 for temperatures $T_I(0,0)$, $T_I(1,0)$, $T_I(0,1)$, $T_J(0,0)$, $T_J(1,0)$, and $T_J(0,1)$. This input can be used to specify temperature gradients that vary linearly both over the cross section and along the length of the element.

The following defaults apply to element temperature input:

- If all temperatures after the first are unspecified, they default to the first. This pattern applies a uniform temperature over the entire element. (The first coordinate temperature, if unspecified, defaults to TUNIF.)
- If all three temperatures at node I are input, and all temperatures at node J are unspecified, the node J temperatures default to the corresponding node I temperatures. This pattern applies a temperature gradient that varies linearly over the cross section but remains constant along the length of the element.
- For any other input pattern, unspecified temperatures default to TUNIF.

Alternatively, temperatures at nodes I and J can be defined using nodal body loads (**BF**,*NODE*,TEMP,*VAL1*). When using a nodal body load to define a temperature, a uniform temperature is applied over the cross section at the specified node.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Temperature gradients across the cross-section are not allowed when the beam section has an arbitrary (ASEC) subtype, where the integrated cross-section inertia properties are user-defined (**SECTYPE**,,BEAM,ASEC).

You can apply an initial stress state to this element via the **INISTATE** command. For more information, see Initial State in the *Basic Analysis Guide*.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

For ocean loading, hydrodynamic added mass and loading, and buoyant loading, are available via the **OCDATA** and **OCTABLE** commands.

If ocean loading is present, the global origin is normally at the mean sea level, with the global Z-axis pointing away from the center of the earth; however, the vertical location can be adjusted via *Zmsl* (*Val6*) on the **OCDATA** command (following the **OCTYPE**,BASIC command).

BEAM188 Input Summary

Nodes

I, J, K (K, the orientation node, is optional but recommended)

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ if KEYOPT(1) = 0
UX, UY, UZ, ROTX, ROTY, ROTZ, WARP if KEYOPT(1) = 1

Section Controls

TXZ, TXY, ADDMAS (See **SECCONTROL**)
(TXZ and TXY default to A*GXZ and A*GXY, respectively, where A = cross-sectional area)

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.

MP command: EX, (PRXY,or NUXY), GXY, GXZ, ALPX, (or CTEX, or THSX), DENS, ALPD, BETD

Surface Loads

Pressure --

face 1 (I-J) (-z normal direction)
face 2 (I-J) (-y normal direction)
face 3 (I-J) (+x tangential direction)
face 4 (I) (+x axial direction)
face 5 (J) (-x axial direction)

I and J denote the end nodes.
Use a negative value for loading in the opposite direction.
Issue the **SFBEAM** command to specify surface loads.
For faces 1, 2, and 3, offsets apply only if you are using the cubic option (KEYOPT(3) = 3).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Body Loads

Temperatures --
T(0,0), T(1,0), T(0,1) at each end node

Special Features

Birth and death (requires KEYOPT(11) = 1)
Element technology autoselect
Generalized cross-section
Initial state
Large deflection
Large strain
Linear perturbation
Nonlinear stabilization
Ocean loading
Stress stiffening

KEYOPT(1)
Warping degree of freedom:

0 --
Six degrees of freedom per node, unrestrained warping (default)

1 --
Seven degrees of freedom per node (including warping). Bimoment and bicurvature are output.

KEYOPT(2)
Cross-section scaling, applies only if **NLGEOM**,ON has been invoked:

0 --
Cross-section is scaled as a function of axial stretch (default)

1 --
Section is assumed to be rigid (classical beam theory)

KEYOPT(3)
Shape functions along the length:

0 --
Linear (default)

2 --
Quadratic

3 --
Cubic

KEYOPT(4)
Shear stress output:

0 --
Output only torsion-related shear stresses (default)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
Output only flexure-related transverse-shear stresses

2 --
Output a combined state of the previous two types

KEYOPT(6), KEYOPT(7), and KEYOPT(9)
active only when **OUTPR**,ESOL is active:

KEYOPT(6)
Output control for section forces/moments and strains/curvatures:

0 --
Output section forces/moments and strains/curvatures at integration points along the length (default)

1 --
Same as KEYOPT(6) = 0 plus current section area

2 --
Same as KEYOPT(6) = 1 plus element basis directions (X,Y,Z)

3 --
Output section forces/moments and strains/curvatures extrapolated to the element nodes

KEYOPT(7)
Output control at integration points (not available when section subtype = ASEC):

0 --
None (default)

1 --
Maximum and minimum stresses/strains

2 --
Same as KEYOPT(7) = 1 plus stresses and strains at each section point

KEYOPT(9)
Output control for values extrapolated to the element and section nodes (not available when section subtype = ASEC):

0 --
None (default)

1 --
Maximum and minimum stresses/strains

2 --
Same as KEYOPT(9) = 1 plus stresses and strains along the exterior boundary of the cross-section

3 --
Same as KEYOPT(9) = 1 plus stresses and strains at all section nodes

KEYOPT(11)
Set section properties:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0 --
Automatically determine if preintegrated section properties can be used (default)

1 --
Use numerical integration of section

KEYOPT(12)
Tapered section treatment:

0 --
Linear tapered section analysis; cross-section properties are evaluated at each Gauss point (default). This is more accurate, but computationally intensive.

1 --
Average cross-section analysis; for elements with tapered sections, cross-section properties are evaluated at the centroid only. This is an approximation of the order of the mesh size; however, it is faster.

KEYOPT(13)
Hydrodynamic output (not available in harmonic analyses that include ocean wave effects (**HROCEAN**)):

0 --
None (default)

1 --
Additional centroidal hydrodynamic printout

KEYOPT(15)
Results file format:

0 --
Store averaged results at each section corner node (default).

1 --
Store non-averaged results at each section integration point. (The volume of data may be excessive. This option is typically useful for built-up sections with multiple materials only.)

BEAM188 Output Data

The solution output associated with these elements is in two forms:

- Nodal displacements and reactions included in the overall nodal solution
- Additional element output as described in Table 188.1: BEAM188 Element Output Definitions (p. 1008)

To view 3-D deformed shapes for BEAM188, issue an **OUTRES**,MISC or **OUTRES**,ALL command for static or transient analyses. To view 3-D mode shapes for a modal or eigenvalue buckling analysis, you must expand the modes with element results calculation active (via the **MXPAND** command's `Elcalc` = YES option).

Linearized Stress

It is customary in beam design to employ components of axial stress that contribute to axial loads and bending in each direction separately; therefore, BEAM188 provides a linearized stress output as part of its SMISC output record, as indicated in the following definitions:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SDIR is the stress component due to axial load.

SDIR = Fx/A, where Fx is the axial load (SMISC quantities 1 and 14) and A is the area of the cross-section.

SByT and SByB are bending-stress components.

$SByT = -Mz * y_{max} / Izz$
$SByB = -Mz * y_{min} / Izz$
$SBzT = My * z_{max} / Iyy$
$SBzB = My * z_{min} / Iyy$

where My, Mz are bending moments in the beam coordinate system (SMISC quantities 2,15,3,16), as shown in Figure 188.1: BEAM188 Geometry (p. 997). Coordinates y_{max}, y_{min}, z_{max}, and z_{min} are the maximum and minimum y, z coordinates in the cross-section measured from the centroid. Values Iyy and Izz are moments of inertia of the cross-section. Except for the ASEC type of beam cross-section, the program uses the maximum and minimum cross-section dimensions. For the ASEC type of cross-section, the maximum and minimum in each of y and y direction is assumed to be +0.5 to -0.5, respectively.

Corresponding definitions for the component strains are:

EPELDIR = Ex
$EPELByT = -Kz * y_{max}$
$EPELByB = -Kz * y_{min}$
$EPELBzT = Ky * z_{max}$
$EPELBzB = Ky * z_{min}$

where Ex, Ky, and Kz are generalized strains and curvatures (SMISC quantities 7,8,9, 20,21 and 22).

The reported stresses are strictly valid only for elastic behavior of members. BEAM188 always employs combined stresses in order to support nonlinear material behavior. When the elements are associated with nonlinear materials, the component stresses can at best be regarded as linearized approximations and should be interpreted with caution.

When using KEYOPT(7) with the cubic option (KEYOPT(3) = 3), the integration point at the middle of the element is reported *last* in the integration-point printout.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 188.1: BEAM188 Element Output Definitions

Name	Definition	O	R
EL	Element number	Y	Y
NODES	Element connectivity	Y	Y
MAT	Material number	Y	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
C.G.:X, Y, Z	Element center of gravity	Y	1
Area	Area of cross-section	2	Y
SF:y, z	Section shear forces	2	Y
SE:y, z	Section shear strains	2	Y
S:xx, xy, xz	Section point stresses	3	Y
EPEL:xx, xy, xz	Elastic strains	3	Y
EPTO:xx, xy, xz	Section point total mechanical strains (EPEL + EPPL + EPCR)	3	Y
EPTT:xx, xy, xz	Section point total strains (EPEL + EPPL + EP-CR+EPTH)	3	Y
EPPL:xx, xy, xz	Section point plastic strains	3	Y
EPCR:xx, xy, xz	Section point creep strains	3	Y
EPTH:xx	Section point thermal strains	3	Y
NL:SEPL	Plastic yield stress	-	4
NL:EPEQ	Accumulated equivalent plastic strain	-	4
NL:CREQ	Accumulated equivalent creep strain	-	4
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	-	4
NL:PLWK	Plastic work/volume	-	4
NL:EPEQ	Accumulated equivalent plastic strain	-	4
SEND:ELASTIC, PLASTIC, CREEP	Strain energy densities	-	4
TQ	Torsional moment	Y	Y
TE	Torsional strain	Y	Y
Ky, Kz	Curvature	Y	Y
Ex	Axial strain	Y	Y
Fx	Axial force	Y	Y
My, Mz	Bending moments	Y	Y
BM	Warping bimoment	6	6
BK	Warping bicurvature	6	6
EXT PRESS	External pressure at integration point	5	5
EFFECTIVE TENS	Effective tension on beam	5	5
SDIR	Axial direct stress	-	2
SByT	Bending stress on the element +Y side of the beam	-	Y
SByB	Bending stress on the element -Y side of the beam	-	Y
SBzT	Bending stress on the element +Z side of the beam	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
SBzB	Bending stress on the element -Z side of the beam	-	Y
EPELDIR	Axial strain at the end	-	Y
EPELByT	Bending strain on the element +Y side of the beam.	-	Y
EPELByB	Bending strain on the element -Y side of the beam.	-	Y
EPELBzT	Bending strain on the element +Z side of the beam.	-	Y
EPELBzB	Bending strain on the element -Z side of the beam.	-	Y
TEMP	Temperatures at all section corner nodes.	-	Y
LOCI:X, Y, Z	Integration point locations	-	8
SVAR:1, 2, ... , N	State variables	-	9
The following values apply to ocean loading only:			
GLOBAL COORD	Element centroid location	7	Y
VR, VZ	Radial and vertical fluid particle velocities (VR is always > 0)	7	Y
AR, AZ	Radial and vertical fluid particle accelerations	7	Y
PHDYN	Dynamic fluid pressure head	7	Y
ETA	Wave amplitude over integration point	7	Y
TFLUID	Fluid temperature (printed if VISC is nonzero)	7	Y
VISC	Viscosity (output if VISC is nonzero)	7	Y
REN, RET	Normal and tangential Reynolds numbers (if VISC is nonzero)	7	Y
CT	Input tangential drag coefficients evaluated at Reynolds numbers	7	Y
CDY, CDZ	Input normal drag coefficients evaluated at Reynolds numbers	7	Y
CMY, CMZ	Input inertia coefficients evaluated at Reynolds numbers	7	Y
URT, URN	Tangential (parallel to element axis) and normal relative velocities	7	Y
ABURN	Vector sum of normal (URN) velocities	7	Y
AN	Accelerations normal to element	7	Y
FX, FY, FZ	Hydrodynamic tangential and normal forces in element coordinates	7	Y
ARGU	Effective position of wave (radians)	7	Y

1. Available only at the centroid as a ***GET** item.

2. See KEYOPT(6) description.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. See KEYOPT(7) and KEYOPT(9) descriptions.
4. Available if the element has a nonlinear material.
5. Available only if ocean loading is present.
6. See KEYOPT(1) description.
7. Applies to ocean loading only.
8. Available only if **OUTRES**,LOCI command is used.
9. Available only if the `UserMat` subroutine and **TB**,STATE command are used.

More output is described via the **PRESOL** and ***GET**,,SECR commands in POST1.

Table 188.2: BEAM188 Item and Sequence Numbers (p. 1011) lists output available via **ETABLE** using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. Table 188.2: BEAM188 Item and Sequence Numbers (p. 1011) uses the following notation:

Name
output quantity as defined in the Table 188.1: BEAM188 Element Output Definitions (p. 1008)

Item
predetermined Item label for **ETABLE**

I,J
sequence number for data at nodes I and J

Table 188.2: BEAM188 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
Fx	SMISC	--	1	14
My	SMISC	--	2	15
Mz	SMISC	--	3	16
TQ	SMISC	--	4	17
SFz	SMISC	--	5	18
SFy	SMISC	--	6	19
Ex	SMISC	--	7	20
Ky	SMISC	--	8	21
Kz	SMISC	--	9	22
TE	SMISC	--	10	23
SEz	SMISC	--	11	24
SEy	SMISC	--	12	25
Area	SMISC	--	13	26
BM	SMISC	--	27	29
BK	SMISC	--	28	30

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
SDIR	SMISC	--	31	36
SByT	SMISC	--	32	37
SByB	SMISC	--	33	38
SBzT	SMISC	--	34	39
SBzB	SMISC	--	35	40
EPELDIR	SMISC	--	41	46
EPELByT	SMISC	--	42	47
EPELByB	SMISC	--	43	48
EPELBzT	SMISC	--	44	49
EPELBzB	SMISC	--	45	50
TEMP	SMISC	--	51-53	54-56
EXT PRESS [1]	SMISC	--	62	66
EFFECTIVE TENS [1]	SMISC	--	63	67
S:xx, xy, xz	LS	--	CI[1], DI[2]	CJ[1], DJ[2]
EPEL:xx, xy, xz	LEPEL	--	CI[1], DI[2]	CJ[1], DJ[2]
EPTH:xx	LEPTH	--	AI[3], BI[4]	AJ[3], BJ[4]
EPPL:xx, xy, xz	LEPPL	--	CI[1], DI[2]	CJ[1], DJ[2]
EPCR:xx, xy, xz	LEPCR	--	CI[1], DI[2]	CJ[1], DJ[2]
EPTO:xx, xy, xz	LEPTO	--	CI[1], DI[2]	CJ[1], DJ[2]
EPTT:xx, xy, xz	LEPTT	--	CI[1], DI[2]	CJ[1], DJ[2]
The following output quantities are valid for ocean loading only:				
GLOBAL COORD	NMISC	1, 2, 3	--	--
VR, VZ	NMISC	4, 5	--	--
AR, AZ	NMISC	6, 7 [6]	--	--
PHDYN	NMISC	8 [6]	--	--
ETA	NMISC	9 [6]	--	--
TFLUID	NMISC	10	--	--
VISC	NMISC	11	--	--
REN, RET	NMISC	12, 13 [7]	--	--
CT	NMISC	14	--	--
CDY, CDZ	NMISC	15, 16	--	--
CMY, CMZ	NMISC	17, 18 [6]	--	--
URT, URN	NMISC	19, 20, 21	--	--
ABURN	NMISC	22 [6]	--	--
AN	NMISC	23, 24 [6]	--	--
FX, FY, FZ	NMISC	25, 26, 27	--	--
ARGU	NMISC	28 [6]	--	--

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. External pressure (EXT PRESS) and effective tension (EFFECTIVE TENS) occur at integration points, and not at end nodes.

2. CI and CJ are the sequence numbers for accessing the averaged line element solution quantities (LS, LEPEL, LEPPL, LEPCR, LEPTO, and LEPTT) at RST section nodes (section corner nodes where results are available), at element Node I and J respectively. CI and CJ are applicable only when KEYOPT(15) = 0. For a given section corner node *nn*, CI and CJ are given as follows:

   CI = (*nn* - 1) * 3+ *COMP*

   CJ = (*nnMax* + *nn* - 1) * 3+ *COMP*

   Where *nnMax* is the total number of RST section nodes, and COMP is the stress or strain component (1 - xx, 2 - xy, 3 - xz). Locations of RST section nodes can be visualized with **SECPLOT**,,6.

3. DI and DJ are the sequence numbers for accessing the non-averaged line element solution quantities (LS, LEPEL, LEPPL, LEPCR, LEPTO, and LEPTT) at RST section integration points (section integration points where results are available), at element Node I and J respectively. DI and DJ are applicable only when KEYOPT(15) = 1. For the ith integration point (i = 1, 2, 3, or 4) in section cell *nc*, DI and DJ are given as follows:

   DI = (*nc* - 1) * 12 + (i - 1) * 3 + COMP

   DJ = (*ncMax* + *nc* - 1) * 12 + (i - 1) * 3 + COMP

   Where *ncMax* is the total number of RST section cells, and COMP is the stress or strain component (1 - xx, 2 - xy, 3 - xz). Locations of RST section cells can be visualized with **SECPLOT**,,7.

4. AI and AJ are the sequence numbers for accessing the averaged line element thermal strain quantities LEPTH at RST section nodes (section corner nodes where results are available), at element Node I and J respectively. AI and AJ are applicable only when KEYOPT(15) = 0. For a given section corner node *nn*, AI and AJ are given as follows:

   AI = *nn*

   AJ = *nnMax* + *nn*

   Where *nnMax* is the total number of RST section nodes. Locations of RST section nodes can be visualized with **SECPLOT**,,6.

5. BI and BJ are the sequence numbers for accessing the non-averaged line element thermal strain quantities LEPTH at RST section integration points (section integration points where results are available), at element Node I and J respectively. BI and BJ are applicable only when KEYOPT(15) = 1. For the ith integration point (i = 1, 2, 3, or 4) in section cell *nc*, BI and BJ are given as follows:

   BI = (*nc* - 1) * 4 + i

   BJ = (*ncMax* + *nc* - 1) * 4 + i

   Where *ncMax* is the total number of RST section cells. Locations of RST section cells can be visualized with **SECPLOT**,,7.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Applies to ocean loading only.

7. These quantities are output only if a Reynold's number dependency is used.

Transverse-Shear Stress Output

The BEAM188 formulation is based on three stress components:

- one axial
- two shear stress

The shear stresses are caused by torsional and transverse loads. BEAM188 is based on first-order shear-deformation theory, also popularly known as Timoshenko beam theory. The transverse-shear strain is constant for the cross-section; therefore, the shear energy is based on a transverse-shear force. The shear force is redistributed by predetermined shear-stress distribution coefficients across the beam cross-section, and made available for output purposes. By default, the program outputs only the shear stresses caused by torsional loading. Use KEYOPT(4) to activate output of shear stresses caused by flexure or transverse loading.

The accuracy of transverse-shear distribution is directly proportional to the mesh density of cross-section modeling (for determination of warping, shear center and other section geometric properties). The traction-free state at the edges of a cross-section is met only in a well-refined model of the cross-section.

By default, the program uses a mesh density (for cross-section modeling) that provides accurate results for torsional rigidity, warping rigidity, inertia properties, and shear-center determination. The default mesh employed is also appropriate for nonlinear material calculations; however, more refined cross-section models may be necessary if the shear stress distribution due to transverse loads must be captured very accurately. Increasing cross-section mesh size does not imply larger computational cost if the cross-section is homogeneous and the associated material is linear. Use the **SECTYPE** and **SECDATA** commands to adjust cross-section mesh density.

The transverse-shear distribution calculation ignores the effects of Poisson's ratio. The Poisson's ratio affects the shear-correction factor and shear-stress distribution slightly, and this effect is ignored.

BEAM188 Assumptions and Restrictions

- The beam must not have zero length.
- By default (KEYOPT(1) = 0), the effect of warping restraint is assumed to be negligible.
- Cross-section failure or folding is not accounted for.
- Rotational degrees of freedom are not included in the lumped mass matrix if offsets are present.
- The element works best with the full Newton-Raphson solution scheme (that is, the default choice in solution control).
- Only moderately "thick" beams can be analyzed. See "BEAM188 Element Technology and Usage Recommendations" (p. 997) for more information.
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). Prestress effects can be activated by the **PSTRES** command.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When the element is associated with nonlinear general beam sections (**SECTYPE**,,GENB), additional restrictions apply. For more information, see Considerations for Using Nonlinear General Beam Sections.
- The output axial force may not be exact when using ocean loading with nonlinear materials.
- For a random vibration (PSD) analysis, equivalent stress is not calculated.
- When the element is used in an ocean environment:
  - For subtype ASEC, the calculated perimeter includes the surfaces of any internal voids.
  - Hydrodynamic output via KEYOPT(13) is not available in harmonic analyses that include ocean wave effects (**HROCEAN**).
  - The three-dimensional effect of water pressure on the element is adjusted, as the element has only one direct stress. For more information, see Hydrostatic Loads in the *Mechanical APDL Theory Reference*.
  - Enclosed spaces used by subtypes CTUBE or HREC are assumed to have the same pressure as the surrounding ocean (that is, as if the flooding option used with PIPE288 or PIPE289 is always enabled).
  - Generally, it is better to use a pipe element (PIPE288 or PIPE289) rather than beam subtype CTUBE, as the pipe element has a degree of freedom to account for cross-sectional compression and can also have an independent internal pressure applied.

BEAM188 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The only special features allowed are stress stiffening and large deflections.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

BEAM189

3-D 3-Node Beam

MP ME ST PR PRN DS <> <> PP EME MFS
Product Restrictions

BEAM189 Element Description

The BEAM189 element is suitable for analyzing slender to moderately stubby/thick beam structures. The element is based on Timoshenko beam theory which includes shear-deformation effects. The element provides options for unrestrained warping and restrained warping of cross-sections.

The element is a quadratic three-node beam element in 3-D. With default settings, six degrees of freedom occur at each node; these include translations in the x, y, and z directions and rotations about the x, y, and z directions. An optional seventh degree of freedom (warping magnitude) is available. The element is well-suited for linear, large rotation, and/or large-strain nonlinear applications.

The element includes stress stiffness terms, by default, in any analysis with **NLGEOM**,ON. The provided stress-stiffness terms enable the elements to analyze flexural, lateral, and torsional stability problems (using eigenvalue buckling, or collapse studies with arc length methods or nonlinear stabilization).

Elasticity, plasticity, creep and other nonlinear material models are supported. A cross-section associated with this element type can be a built-up section referencing more than one material. Added mass, hydrodynamic added mass and loading, and buoyant loading are available.

For more detailed information about this element, see BEAM189 - 3-D 3-Node Beam in the *Mechanical APDL Theory Reference*.

Figure 189.1: BEAM189 Geometry

BEAM189 Element Technology and Usage Recommendations

BEAM189 is based on Timoshenko beam theory, which is a first-order shear-deformation theory: transverse-shear strain is constant through the cross-section (that is, cross-sections remain plane and undistorted after deformation).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The element can be used for slender or stout beams. Due to the limitations of first-order shear-deformation theory, slender to moderately thick beams can be analyzed. Use the slenderness ratio of a beam structure ($GAL^2/(EI)$) to judge the applicability of the element, where:

G
Shear modulus

A
Area of the cross-section

L
Length of the member (not the element length)

EI
Flexural rigidity

Calculate the ratio using some global distance measures, rather than basing it upon individual element dimensions. The following illustration shows an estimate of transverse-shear deformation in a cantilever beam subjected to a tip load. Although the results cannot be extrapolated to any other application, the example serves well as a general guideline. A slenderness ratio greater than 30 is recommended.

Figure 189.2: Transverse-Shear Deformation Estimation

Slenderness Ratio ($GAL^2/(EI)$)	δ Timoshenko / δ Euler-Bernoulli
25	1.120
50	1.060
100	1.030
1000	1.003

These elements support an elastic relationship between transverse-shear forces and transverse-shear strains. You can override default values of transverse-shear stiffnesses using the **SECCONTROL** command.

BEAM189 does not use higher-order theories to account for variation in distribution of shear stresses. Use solid elements if such effects must be considered.

BEAM189 supports “restrained warping” analysis by making available a seventh degree of freedom at each beam node. By default, BEAM189 elements assume that the warping of a cross-section is small enough that it may be neglected (KEYOPT(1) = 0). You can activate the warping degree of freedom by using KEYOPT(1) = 1. With the warping degree of freedom activated, each node has seven degrees of freedom: UX, UY, UZ, ROTX, ROTZ, ROTY, and WARP. With KEYOPT(1) = 1, bimoment and bicurvature are output.

Unlike other cubic (Hermitian) polynomial-based elements, BEAM189 is based on quadratic polynomials; therefore, offsets in specification of distributed pressure loads are not allowed.BEAM189 has linear bending-moment variation. Refinement of the mesh is recommended in order to accommodate such

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

loading. BEAM189 is computationally efficient and has super-convergence properties with respect to mesh refinement. For example, the quadratic beam with a two point Gaussian integration is known to be of same accuracy as a Hermitian element.

In practice, when two elements with "restrained warping" come together at a sharp angle, you need to couple the displacements and rotations, but leave the out-of-plane warping decoupled. This is normally accomplished by having two nodes at a physical location and using appropriate constraints. This process is made easier (or automated) by the **ENDRELEASE** command, which decouples the out-of plane warping for any adjacent elements with cross-sections intersecting at an angle greater than 20 degrees.

BEAM189 allows change in cross-sectional inertia properties as a function of axial elongation. By default, the cross-sectional area changes such that the volume of the element is preserved after deformation. The default is suitable for elastoplastic applications. By using KEYOPT(2), you can choose to keep the cross-section constant or rigid. Scaling is not an option for nonlinear general beam sections (**SECTYPE**,,GENB) or for its effect on ocean loading..

For the mass matrix and evaluation of consistent load vectors, a higher order integration rule than that used for stiffness matrix is employed. The elements support both consistent and lumped mass matrices. Avoid using **LUMPM**,ON as BEAM189 is a higher-order element. Consistent mass matrix is used by default. You can add mass per unit length via the **SECCONTROL** command's *ADDMAS* values. See "BEAM189 Input Summary" (p. 1023).

When ocean loading is applied, the loading is nonlinear (that is, based on the square of the relative velocity between the structure and the water). Accordingly, the full Newton-Raphson option (**NROPT**,FULL) may be necessary to achieve optimal results. (Full Newton-Raphson is applied automatically in an analysis involving large-deflection effects [**NLGEOM**,ON].)

The St. Venant warping functions for torsional behavior are determined in the undeformed state, and are used to define shear strain even after yielding. No options are available for recalculating in deformed configuration the torsional shear distribution on cross-sections during the analysis and possible partial plastic yielding of cross-sections. As such, large inelastic deformation due to torsional loading should be treated and verified with caution. Under such circumstances, alternative modeling using solid or shell elements is recommended.

BEAM189 Input Data

The geometry, node locations, coordinate system, and pressure directions for this element are shown in Figure 189.1: BEAM189 Geometry (p. 1017). BEAM189 is defined by nodes I, J, and K in the global coordinate system.

Node L is a preferred way to define the orientation of the element. For information about orientation nodes and beam meshing, see Generating a Beam Mesh With Orientation Nodes in the *Modeling and Meshing Guide*. Also, see Quadratic Elements (Midside Nodes) in the same manual for the use of midside nodes. See the **LMESH** and **LATT** command descriptions for details on generating the L node automatically. For a description of the low-order beam, see BEAM188.

The element can also be defined without the orientation node L. In this case, the element x-axis is oriented from node I (end 1) toward node J (end 2). When no orientation node is used, the default orientation of the element y-axis is automatically calculated to be parallel to the global X-Y plane. For the case where the element is parallel to the global Z-axis (or within a 0.01 percent slope of it), the element y-axis is oriented parallel to the global Y-axis (as shown). For user control of the element orientation about the element x-axis, use the L node option. If both are defined, the orientation node option takes precedence. The orientation node L, if used, defines a plane (with I and J) containing the element x and

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

z-axes (as shown). If using this element in a large-deflection analysis, be aware that the location of the orientation node L is used only to *initially* orient the element.

The number of degrees of freedom depends on the value of KEYOPT(1). When KEYOPT(1) = 0 (the default), six degrees of freedom occur at each node. These include translations in the x, y, and z directions and rotations about the x, y, and z directions. When KEYOPT(1) = 1, a seventh degree of freedom (warping magnitude) is also considered.

The element is a one-dimensional line element in space. The cross-section details are provided separately via the **SECTYPE** and **SECDATA** commands. (See Beam Analysis and Cross-Sections in the *Structural Analysis Guide* for details.) A section is associated with the beam elements by specifying the section ID number (**SECNUM**). A section number is an independent attribute. In addition to a constant cross-section, you can also define a tapered cross-section by using the TAPER option on the **SECTYPE** command. (For more information, see Defining a Tapered Beam.)

BEAM189 ignores any real constant data beginning with Release 6.0. See **SECCONTROL** command for defining the transverse-shear stiffness, and added mass.

A summary of the element input follows in "BEAM189 Input Summary" (p. 1023).

BEAM189 Cross-Sections

BEAM189 can be associated with these cross-section types:

- Standard library section types or user meshes which define the geometry of the beam cross-section (**SECTYPE**,,BEAM). The material of the beam is defined either as an element attribute (MAT), or as part of section buildup (for multi-material cross-sections).
- Generalized beam cross-sections (**SECTYPE**,,GENB), where the relationships of generalized stresses to generalized strains are input directly.
- Tapered beam cross-sections (**SECTYPE**,,TAPER), where a standard library section or user mesh defines each end of the beam.

Standard Library Sections

BEAM189 is provided with section-relevant quantities (area of integration, position, etc.) automatically at a number of section points using **SECTYPE** and **SECDATA**. Each section is assumed to be an assembly of a predetermined number of nine-node cells. Each cross-section cell has four integration points and each can be associated with an independent material type.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 189.3: Cross-Section Cells

The number of cells in the cross-sections influences the accuracy of section properties and ability to model nonlinear stress-strain relationship through the cross-section. The element has a nested structure of integration (along the length and in the cross-section).

When the material associated with the elements has inelastic behavior or when the temperature varies across the section, constitutive calculations are performed at the section integration points. For more common elastic applications, the element uses precalculated properties of the section at the element integration points. However, the stresses and strains are calculated in the output pass at the section nodes.

If the section is assigned the subtype ASEC, only the generalized stresses and strains (axial force, bending moments, transverse shears, curvatures, and shear strains) are available for output. 3-D contour plots and deformed shapes are not available. The ASEC subtype is displayed only as a thin rectangle to verify beam orientation. BEAM189 treats ASEC as a section type with just one cross-section integration point.

BEAM189 is helpful for analyzing built-up beams, (that is, those fabricated of two or more pieces of material joined together to form a single, solid beam). The pieces are assumed to be perfectly bonded together; therefore, the beam behaves as a single member.

The multi-material cross-section capability is applicable only where the assumptions of a beam behavior (Timoshenko or Bernoulli-Euler beam theory) holds.

In other words, what is supported is a simple extension of a conventional Timoshenko beam theory. It may be used in applications such as:

- bimetallic strips
- beams with metallic reinforcement

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- sensors where layers of a different material has been deposited

BEAM189 does not account for coupling of bending and twisting at the section stiffness level. The transverse shears are also treated in an uncoupled manner. This may have a significant effect on layered composite and sandwich beams if the lay-up is unbalanced.

Always validate the application of BEAM189, either with experiments or other numerical analysis. Use the restrained warping option with built-up sections after due verification.

KEYOPT(15) specifies the format of the `.rst` results file. For KEYOPT(15) = 0, the format gives only one averaged result at each section corner node; therefore, this option typically applies to homogeneous sections. For KEYOPT(15) = 1, the format gives one result for each section integration point; therefore, this option typically applies to built-up sections with multiple materials (and generates a larger results file).

Generalized Beam Cross-Sections

When using nonlinear general beam sections, neither the geometric properties nor the material is explicitly specified. *Generalized stress* implies the axial force, bending moments, torque, and transverse-shear forces. Similarly, *generalized strain* implies the axial strain, bending curvatures, twisting curvature, and transverse-shear strains. (For more information, see nonlinear general beam sections.) This is an abstract method for representing cross-section behavior; therefore, input often consists of experimental data or the results of other analyses.

Generally, BEAM189 supports an elastic relationship between transverse-shear forces and transverse-shear strains. You can override default values of transverse-shear stiffnesses via the **SECCONTROL** command.

When the beam element is associated with a generalized beam (**SECTYPE**,,GENB) cross-section type, the relationship of transverse-shear force to the transverse-shear strain can be nonlinear elastic or plastic, an especially useful capability when flexible spot welds are modeled. In such a case, the **SECCONTROL** command does not apply.

Tapered Beam Cross-Sections

A linearly tapered beam is defined by specifying a standard library section or user mesh at each end of the beam. The section geometries are specified at global coordinates, then linear interpolated and evaluated at the element. The sections at the end points must be topologically identical. (For more information, see Defining a Tapered Beam.)

BEAM189 Loads

Forces are applied at the nodes (which also define the element x-axis). If the centroidal axis is not colinear with the element x-axis, applied axial forces will cause bending. Applied shear forces will cause torsional strains and moment if the centroid and shear center of the cross-section are different. The nodes should therefore be located at the points where you want to apply the forces. Use the *OFFSETY* and *OFFSETZ* arguments of the **SECOFFSET** command appropriately.

Element loads are described in Nodal Loading (p. 49). Pressures may be input as surface loads on the element faces as shown by the circled numbers on Figure 189.1: BEAM189 Geometry (p. 1017). Positive normal pressures act into the element. Lateral pressures are input as force per unit length. End "pressures" are input as forces.

At both ends of the element, temperatures can be input at these locations:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- At the element x-axis (T(0,0))
- At one unit from the x-axis in the element y-direction (T(1,0))
- At one unit from the x-axis in the element z-direction (T(0,1))

Element locations (T(y,z)) are given according to the convention used in Figure 189.1: BEAM189 Geometry (p. 1017).

For beam elements, element body load commands (**BFE**) accept an element number and a list of values, 1 through 6 for temperatures $T_I(0,0)$, $T_I(1,0)$, $T_I(0,1)$, $T_J(0,0)$, $T_J(1,0)$, and $T_J(0,1)$. This input can be used to specify temperature gradients that vary linearly both over the cross section and along the length of the element.

The following defaults apply to element temperature input:

- If all temperatures after the first are unspecified, they default to the first. This pattern applies a uniform temperature over the entire element. (The first coordinate temperature, if unspecified, defaults to TUNIF.)
- If all three temperatures at node I are input, and all temperatures at node J are unspecified, the node J temperatures default to the corresponding node I temperatures. This pattern applies a temperature gradient that varies linearly over the cross section but remains constant along the length of the element.
- For any other input pattern, unspecified temperatures default to TUNIF.

Alternatively, temperatures at nodes I and J can be defined using nodal body loads (**BF**,*NODE*,TEMP,*VAL1*). When using a nodal body load to define a temperature, a uniform temperature is applied over the cross section at the specified node. (**BF** command input is not accepted at node K.)

Temperature gradients across the cross-section are not allowed when the beam section has an arbitrary (ASEC) subtype, where the integrated cross-section inertia properties are user-defined (**SECTYPE**,,BEAM,ASEC).

You can apply an initial stress state to this element via the **INISTATE** command. For more information, see Initial State in the *Basic Analysis Guide*.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

For ocean loading, hydrodynamic added mass and loading, and buoyant loading, are available via the **OCDATA** and **OCTABLE** commands.

If ocean loading is present, the global origin is normally at the mean sea level, with the global Z-axis pointing away from the center of the earth; however, the vertical location can be adjusted via `Zmsl` (`Val6`) on the **OCDATA** command (following the **OCTYPE**,BASIC command).

BEAM189 Input Summary

Nodes

I, J, K, L (the orientation node L is optional but recommended)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ if KEYOPT(1) = 0
UX, UY, UZ, ROTX, ROTY, ROTZ, WARP if KEYOPT(1) = 1

Section Controls

TXZ, TXY, ADDMAS (see **SECCONTROL**)
(TXZ and TXY default to A*GXZ and A*GXY, respectively, where A = cross-sectional area)

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.

MP command: EX, (PRXY,or NUXY), GXY, GXZ, ALPX, (or CTEX, or THSX), DENS, ALPD, BETD

Surface Loads

Pressure --

face 1 (I-J) (-z normal direction)
face 2 (I-J) (-y normal direction)
face 3 (I-J) (+x tangential direction)
face 4 (I) (+x axial direction)
face 5 (J) (-x axial direction)

I and J denote the end nodes.
Use a negative value for loading in the opposite direction.
Issue the **SFBEAM** command to specify surface loads.
Distributed pressure offsets are not available for faces 1, 2, and 3.

Body Loads

Temperatures --
T(0,0), T(1,0), T(0,1) at each end node

Special Features

Birth and death (requires KEYOPT(11) = 1)
Element technology autoselect
Generalized cross-section
Initial state
Large deflection
Large strain
Linear perturbation
Nonlinear stabilization
Ocean loading
Stress stiffening

KEYOPT(1)

Warping degree of freedom:

0 --
Six degrees of freedom per node, unrestrained warping (default)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
Seven degrees of freedom per node (including warping). Bimoment and bicurvature are output.

KEYOPT(2)
Cross-section scaling:

0 --
Cross-section is scaled as a function of axial stretch (default); applies only if **NLGEOM**,ON has been invoked

1 --
Section is assumed to be rigid (classical beam theory)

KEYOPT(4)
Shear stress output:

0 --
Output only torsion-related shear stresses (default)

1 --
Output only flexure-related transverse-shear stresses

2 --
Output a combined state of the previous two types.

KEYOPT(6), KEYOPT(7), and KEYOPT(9)
Active only when **OUTPR**,ESOL is active:

KEYOPT(6)
Output control for section forces/moments and strains/curvatures:

0 --
Output section forces/moments and strains/curvatures at integration points along the length (default)

1 --
Same as KEYOPT(6) = 0 plus current section area

2 --
Same as KEYOPT(6) = 1 plus element basis directions (X,Y,Z)

3 --
Output section forces/moments and strains/curvatures extrapolated to the element nodes

KEYOPT(7)
Output control at section integration point (not available when section subtype = ASEC):

0 --
None (default)

1 --
Maximum and minimum stresses/strains

2 --
Same as KEYOPT(7) = 1 plus stresses and strains at each section point

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(9)

Output control for values extrapolated to the element and section nodes (not available when section subtype = ASEC):

0 --

None (default)

1 --

Maximum and minimum stresses/strains

2 --

Same as KEYOPT(9) = 1 plus stresses and strains along the exterior boundary of the cross-section

3 --

Same as KEYOPT(9) = 1 plus stresses and strains at all section nodes

KEYOPT(11)

Set section properties:

0 --

Automatically determine if preintegrated section properties can be used (default)

1 --

Use numerical integration of section

KEYOPT(12)

Tapered section treatment:

0 --

Linear tapered section analysis; cross-section properties are evaluated at each Gauss point (default). This is more accurate, but computationally intensive.

1 --

Average cross-section analysis; for elements with tapered sections, cross-section properties are evaluated at the centroid only. This is an approximation of the order of the mesh size; however, it is faster.

KEYOPT(13)

Hydrodynamic output (not available in harmonic analyses that include ocean wave effects (**HROCEAN**)):

0 --

None (default)

1 --

Additional centroidal hydrodynamic printout

KEYOPT(15)

Results file format:

0 --

Store averaged results at each section corner node (default).

1 --

Store non-averaged results at each section integration point. (The volume of data may be excessive. This option is typically useful for built-up sections with multiple materials only.)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BEAM189 Output Data

The solution output associated with these elements is in two forms:

- Nodal displacements and reactions included in the overall nodal solution
- Additional element output as described in Table 189.1: BEAM189 Element Output Definitions (p. 1028)

To view 3-D deformed shapes for BEAM189, issue an **OUTRES**,MISC or **OUTRES**,ALL command for static or transient analyses. To view 3-D mode shapes for a modal or eigenvalue buckling analysis, you must expand the modes with element results calculation active (via the **MXPAND** command's `Elcalc` = YES option).

Linearized Stress

It is customary in beam design to employ components of axial stress that contribute to axial loads and bending in each direction separately; therefore, BEAM189 provides a linearized stress output as part of its SMISC output record, as indicated in the following definitions:

SDIR is the stress component due to axial load.

SDIR = Fx/A, where Fx is the axial load (SMISC quantities 1 and 14) and A is the area of the cross-section.

SByT and SByB are bending stress components.

$$SByT = -Mz * y_{max} / Izz$$
$$SByB = -Mz * y_{min} / Izz$$
$$SBzT = My * z_{max} / Iyy$$
$$SBzB = My * z_{min} / Iyy$$

where My, Mz are bending moments (SMISC quantities 2,15,3,16). Coordinates y_{max}, y_{min}, z_{max}, and z_{min} are the maximum and minimum y, z coordinates in the cross-section measured from the centroid. Values Iyy and Izz are moments of inertia of the cross-section. Except for the ASEC type of beam cross-section, the program uses the maximum and minimum cross-section dimensions. For the ASEC type of cross-section, the maximum and minimum in each of y and z direction is assumed to be +0.5 to -0.5, respectively.

Corresponding definitions for the component strains are:

$$EPELDIR = Ex$$
$$EPELBYT = -Kz * y_{max}$$
$$EPELBYB = -Kz * y_{min}$$
$$EPELBZT = Ky * z_{max}$$
$$EPELBZB = Ky * z_{min}$$

where Ex, Ky, and Kz are generalized strains and curvatures (SMISC quantities 7,8,9, 20,21 and 22).

The reported stresses are strictly valid only for elastic behavior of members. BEAM189 always employs combined stresses in order to support nonlinear material behavior. When the elements are associated with nonlinear materials, the component stresses may at best be regarded as linearized approximations and should be interpreted with caution.

The Element Output Definitions table uses the following notation:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 189.1: BEAM189 Element Output Definitions

Name	Definition	O	R
EL	Element number	Y	Y
NODES	Element connectivity	Y	Y
MAT	Material number	Y	Y
C.G.:X, Y, Z	Element center of gravity	Y	Y
Area	Area of cross-section	1	Y
SF:y, z	Section shear forces	1	Y
SE:y, z	Section shear strains	1	Y
S:xx, xy, xz	Section point stresses	2	Y
EPEL:xx, xy, xz	Elastic strains	2	Y
EPTO:xx, xy, xz	Section point total mechanical strains (EPEL + EPPL + EPCR)	2	Y
EPTT:xx, xy, xz	Section point total strains (EPEL + EPPL + EPCR + EPTH)	2	Y
EPPL:xx, xy, xz	Section point plastic strains	2	Y
EPCR:xx, xy, xz	Section point creep strains	2	Y
EPTH:xx	Section point thermal strains	2	Y
NL:SEPL	Plastic yield stress	-	3
NL:EPEQ	Accumulated equivalent plastic strain	-	3
NL:CREQ	Accumulated equivalent creep strain	-	3
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	-	3
NL:PLWK	Plastic work/volume	-	3
NL:EPEQ	Accumulated equivalent plastic strain	-	3
SEND:ELASTIC, PLASTIC, CREEP	Strain energy densities	-	3
TQ	Torsional moment	Y	Y
TE	Torsional strain	Y	Y
Ky, Kz	Curvature	Y	Y
Ex	Axial strain	Y	Y
Fx	Axial force	Y	Y
My, Mz	Bending moments	Y	Y
BM	Warping bimoment	4	4

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
BK	Warping bicurvature	4	4
EXT PRESS	External pressure at integration point	5	5
EFFECTIVE TENS	Effective tension on beam	5	5
SDIR	Axial direct stress	-	1
SByT	Bending stress on the element +Y side of the beam	-	Y
SByB	Bending stress on the element -Y side of the beam	-	Y
SBzT	Bending stress on the element +Z side of the beam	-	Y
SBzB	Bending stress on the element -Z side of the beam	-	Y
EPELDIR	Axial strain at the end	-	Y
EPELByT	Bending strain on the element +Y side of the beam.	-	Y
EPELByB	Bending strain on the element -Y side of the beam.	-	Y
EPELBzT	Bending strain on the element +Z side of the beam.	-	Y
EPELBzB	Bending strain on the element -Z side of the beam.	-	Y
TEMP	Temperatures at all section corner nodes.	-	Y
LOCI:X, Y, Z	Integration point locations	-	7
SVAR:1, 2, ... , N	State variables	-	8
The following values apply to ocean loading only:			
GLOBAL COORD	Element centroid location	6	Y
VR, VZ	Radial and vertical fluid particle velocities (VR is always > 0)	6	Y
AR, AZ	Radial and vertical fluid particle accelerations	6	Y
PHDYN	Dynamic fluid pressure head	6	Y
ETA	Wave amplitude over integration point	6	Y
TFLUID	Fluid temperature (printed if VISC is nonzero)	6	Y
VISC	Viscosity (output if VISC is nonzero)	6	Y
REN, RET	Normal and tangential Reynolds numbers (if VISC is nonzero)	6	Y
CT	Input tangential drag coefficients evaluated at Reynolds numbers	6	Y
CDY, CDZ	Input normal drag coefficients evaluated at Reynolds numbers	6	Y
CMY, CMZ	Input inertia coefficients evaluated at Reynolds numbers	6	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
URT, URN	Tangential (parallel to element axis) and normal relative velocities	6	Y
ABURN	Vector sum of normal (URN) velocities	6	Y
AN	Accelerations normal to element	6	Y
FX, FY, FZ	Hydrodynamic tangential and normal forces in element coordinates	6	Y
ARGU	Effective position of wave (radians)	6	Y

1. See KEYOPT(6) description
2. See KEYOPT(7) and KEYOPT(9) descriptions
3. Available if the element has a nonlinear material.
4. See KEYOPT(1) description
5. Available only if ocean loading is present.
6. Applies to ocean loading only.
7. Available only if **OUTRES**,LOCI command is used.
8. Available only if the `UserMat` subroutine and **TB**,STATE command are used.

More output is described via the **PRESOL** and ***GET**,,SECR commands in POST1.

Table 189.2: BEAM 189 Item and Sequence Numbers (p. 1030) lists output available via **ETABLE** using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. Table 189.2: BEAM 189 Item and Sequence Numbers (p. 1030) uses the following notation:

Name
output quantity as defined in the Table 189.1: BEAM189 Element Output Definitions (p. 1028)

Item
predetermined Item label for **ETABLE**

I,J
sequence number for data at nodes I and J

Table 189.2: BEAM 189 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
Fx	SMISC	--	1	14
My	SMISC	--	2	15
Mz	SMISC	--	3	16
TQ	SMISC	--	4	17
SFz	SMISC	--	5	18
SFy	SMISC	--	6	19

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
Ex	SMISC	--	7	20
Ky	SMISC	--	8	21
Kz	SMISC	--	9	22
TE	SMISC	--	10	23
SEz	SMISC	--	11	24
SEy	SMISC	--	12	25
Area	SMISC	--	13	26
BM	SMISC	--	27	29
BK	SMISC	--	28	30
SDIR	SMISC	--	31	36
SByT	SMISC	--	32	37
SByB	SMISC	--	33	38
SBzT	SMISC	--	34	39
SBzB	SMISC	--	35	40
EPELDIR	SMISC	--	41	46
EPELByT	SMISC	--	42	47
EPELByB	SMISC	--	43	48
EPELBzT	SMISC	--	44	49
EPELBzB	SMISC	--	45	50
TEMP	SMISC	--	51-53	54-56
EXT PRESS [1]	SMISC	--	62	66
EFFECTIVE TENS [1]	SMISC	--	63	67
S:xx, xy, xz	LS	--	CI[1], DI[2]	CJ[1], DJ[2]
EPEL:xx,xz,xy	LEPEL	--	CI[1], DI[2]	CJ[1], DJ[2]
EPTH:xx	LEPTH	--	AI[3], BI[4]	AJ[3], BJ[4]
EPPL:xx,xz,xy	LEPPL	--	CI[1], DI[2]	CJ[1], DJ[2]
EPCR:xx,xz,xy	LEPCR	--	CI[1], DI[2]	CJ[1], DJ[2]
EPTO:xx, xy, xz	LEPTO	--	CI[1], DI[2]	CJ[1], DJ[2]
EPTT:xx, xy, xz	LEPTT	--	CI[1], DI[2]	CJ[1], DJ[2]
The following output quantities are valid for ocean loading only:				
GLOBAL COORD	NMISC	1, 2, 3	--	--
VR, VZ	NMISC	4, 5	--	--
AR, AZ	NMISC	6, 7 [6]	--	--
PHDYN	NMISC	8 [6]	--	--
ETA	NMISC	9 [6]	--	--
TFLUID	NMISC	10	--	--
VISC	NMISC	11	--	--

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
REN, RET	NMISC	12, 13 [7]	--	--
CT	NMISC	14	--	--
CDY, CDZ	NMISC	15, 16	--	--
CMY, CMZ	NMISC	17, 18 [6]	--	--
URT, URN	NMISC	19, 20, 21	--	--
ABURN	NMISC	22 [6]	--	--
AN	NMISC	23, 24 [6]	--	--
FX, FY, FZ	NMISC	25, 26, 27	--	--
ARGU	NMISC	28 [6]	--	--

1. External pressure (EXT PRESS) and effective tension (EFFECTIVE TENS) occur at integration points, and not at end nodes.

2. CI and CJ are the sequence numbers for accessing the averaged line element solution quantities (LS, LEPEL, LEPPL, LEPCR, LEPTO, and LEPTT) at RST section nodes (section corner nodes where results are available), at element Node I and J respectively. CI and CJ are applicable only when KEYOPT(15) = 0. For a given section corner node *nn*, CI and CJ are given as follows:

   CI = (*nn* - 1) * 3+ COMP

   CJ = (*nnMax* + *nn* - 1) * 3 + COMP

   Where *nnMax* is the total number of RST section nodes, and COMP is the stress or strain component (1 - xx, 2 - xy, 3 - xz). Locations of RST section nodes can be visualized with **SECPLOT**,,6.

3. DI and DJ are the sequence numbers for accessing the non-averaged line element solution quantities (LS, LEPEL, LEPPL, LEPCR, LEPTO, and LEPTT) at RST section integration points (section integration points where results are available), respectively at element Node I and J. DI and DJ are applicable only when KEYOPT(15) = 1. For the ith integration point (i = 1, 2, 3, or 4) in section cell *nc*, DI and DJ are given as follows:

   DI = (*nc* - 1) * 12 + (i - 1) * 3 + COMP

   DJ = (*ncMax* + *nc* - 1) * 12 + (i - 1) * 3 + COMP

   Where *ncMax* is the total number of RST section cells, and COMP is the stress or strain component (1 - xx, 2 - xy, 3 - xz). Locations of RST section cells can be visualized with **SECPLOT**,,7.

4. AI and AJ are the sequence numbers for accessing the averaged line element thermal strain quantities LEPTH at RST section nodes (section corner nodes where results are available), at element Node I and J respectively. AI and AJ are applicable only when KEYOPT(15) = 0. For a given section corner node *nn*, AI and AJ are given as follows:

   AI = *nn*

   AJ = *nnMax* + *nn*

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Where *nnMax* is the total number of RST section nodes. Locations of RST section nodes can be visualized with **SECPLOT**,,6.

5. BI and BJ are the sequence numbers for accessing the non-averaged line element thermal strain quantities LEPTH at RST section integration points (section integration points where results are available), at element Node I and J respectively. BI and BJ are applicable only when KEYOPT(15) = 1. For the ith integration point (i = 1, 2, 3, or 4) in section cell *nc*, BI and BJ are given as follows:

   BI = (*nc* - 1) * 4 + i

   BJ = (*ncMax* + *nc* - 1) * 4 + i

   Where *ncMax* is the total number of RST section cells. Locations of RST section cells can be visualized with **SECPLOT**,,7.

6. Applies to ocean loading only.

7. These quantities are output only if a Reynold's number dependency is used.

For more usage details, see Plot and Review the Section Results and Example Problem with Cantilever Beams.

Transverse-Shear Stress Output

The BEAM189 formulation is based on three stress components:

- one axial
- two shear stress

The shear stresses are caused by torsional and transverse loads. BEAM189 is based on first-order shear-deformation theory, also popularly known as Timoshenko beam theory. The transverse-shear strain is constant for the cross-section; therefore, the shear energy is based on a transverse-shear force. The shear force is redistributed by predetermined shear-stress distribution coefficients across the beam cross-section, and made available for output purposes. By default, the program outputs only the shear stresses caused by torsional loading. Use KEYOPT(4) to activate output of shear stresses caused by flexure or transverse loading.

The accuracy of transverse-shear distribution is directly proportional to the mesh density of cross-section modeling (for determination of warping, shear center and other section geometric properties). The traction-free state at the edges of a cross-section is met only in a well-refined model of the cross-section.

By default, the program uses a mesh density (for cross-section modeling) that provides accurate results for torsional rigidity, warping rigidity, inertia properties, and shear-center determination. The default mesh employed is also appropriate for nonlinear material calculations; however, more refined cross-section models may be necessary if the shear stress distribution due to transverse loads must be captured very accurately. Increasing cross-section mesh size does not imply larger computational cost if the cross-section is homogeneous and the associated material is linear. Use the **SECTYPE** and **SECDATA** commands to adjust cross-section mesh density.

The transverse-shear distribution calculation ignores the effects of Poisson's ratio. The Poisson's ratio affects the shear-correction factor and shear-stress distribution slightly, and this effect is ignored.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BEAM189 Assumptions and Restrictions

- The beam must not have zero length.
- By default (KEYOPT(1) = 0), the effect of warping restraint is assumed to be negligible.
- Cross-section failure or folding is not accounted for.
- Rotational degrees of freedom are not included in the lumped mass matrix if node-location offsets are present.
- The element allows curved member definition and linear-bending-moment variation. However, if a cubic representation for lateral displacement is necessary and the members are initially straight, ANSYS, Inc. recommends using BEAM188 with the cubic option (KEYOPT(3) = 3).
- The element includes the effects of transverse shear and accounts for the initial curvature of the beams.
- The element works best with the full Newton-Raphson solution scheme (that is, the default choice in solution control).
- Only moderately "thick" beams can be analyzed. See the "BEAM189 Element Technology and Usage Recommendations" (p. 1017) section for more information.
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). Prestress effects can be activated by the **PSTRES** command.
- When the element is associated with nonlinear general beam sections (**SECTYPE**,,GENB), additional restrictions apply. For more information, see Considerations for Using Nonlinear General Beam Sections.
- The output axial force may not be exact when using ocean loading with nonlinear materials.
- For a random vibration (PSD) analysis, equivalent stress is not calculated.
- When the element is used in an ocean environment:
  - For subtype ASEC, the calculated perimeter includes the surfaces of any internal voids.
  - Hydrodynamic output via KEYOPT(13) is not available in harmonic analyses that include ocean wave effects (**HROCEAN**).
  - The three-dimensional effect of water pressure on the element is adjusted, as the element has only one direct stress. For more information, see Hydrostatic Loads in the *Mechanical APDL Theory Reference*.
  - Enclosed spaces used by subtypes CTUBE or HREC are assumed to have the same pressure as the surrounding ocean (that is, as if the flooding option used with PIPE288 or PIPE289 is always enabled).
  - Generally, it is better to use a pipe element (PIPE288 or PIPE289) rather than beam subtype CTUBE, as the pipe element has a degree of freedom to account for cross-sectional compression and can also have an independent internal pressure applied.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BEAM189 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The only special features allowed are stress stiffening and large deflections.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

SOLSH190

3-D 8-Node Structural Solid Shell

MP ME ST PR PRN DS <> <> PP EME MFS
Product Restrictions

SOLSH190 Element Description

SOLSH190 is used for simulating shell structures with a wide range of thickness (from thin to moderately thick). The element possesses the continuum solid element topology and features eight-node connectivity with three degrees of freedom at each node: translations in the nodal x, y, and z directions. Thus, connecting SOLSH190 with other continuum elements requires no extra efforts. A degenerate prism option is available, but should only be used as filler elements in mesh generation. The element has plasticity, hyperelasticity, stress stiffening, creep, large deflection, and large strain capabilities. It also has mixed u-P formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials. The element formulation is based on logarithmic strain and true stress measures.

You can use SOLSH190 for layered applications such as modeling laminated shells or sandwich construction. The layered section definition is given by section (SEC*xxx*) commands. Accuracy in modeling composite shells is governed by the first-order shear-deformation theory (also known as Mindlin-Reissner shell theory).

See SOLSH190 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 190.1: SOLSH190 Geometry

x_o = Element x-axis if ESYS is not supplied.

x = Element x-axis if ESYS is supplied.

SOLSH190 Input Data

The geometry, node locations, and the element coordinate system for this element are shown in Figure 190.1: SOLSH190 Geometry (p. 1037). The element is defined by eight nodes. The element coordinate

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

system follows the shell convention where the z axis is normal to the surface of the shell. The node ordering must follow the convention that the I-J-K-L and M-N-O-P element faces represent the bottom and top shell surfaces, respectively. You can change the orientation within the plane of the layers via the **ESYS** command as you would for shell elements (as described in Coordinate Systems (p. 55)). To achieve the correct nodal ordering for a volume mapped (hexahedron) mesh, you can use the **VEORIENT** command to specify the desired volume orientation before executing the **VMESH** command. Alternatively, you can use the **EORIENT** command after automatic meshing to reorient the elements to be in line with the orientation of another element, or to be as parallel as possible to a defined **ESYS** axis.

Layered Section Definition Using Section Commands

You can associate SOLSH190 with a shell section (**SECTYPE**). The layered composite specifications (including layer thickness, material, orientation, and number of integration points through the thickness of the layer) are specified via shell section (SEC*xxx*) commands. You can use the shell section commands even with a single-layered SOLSH190 element. ANSYS obtains the actual layer thicknesses used for element calculations by scaling the input layer thickness so that they are consistent with the thickness between the nodes. A section can be partially defined using data from a FiberSIM `.xml` file.

You can designate the number of integration points (1, 3, 5, 7, or 9) located through the thickness of each layer. Two points are located on the top and bottom surfaces respectively and the remaining points are distributed equal distance between the two points. The element requires at least two points through the entire thickness. When no shell section definition is provided, the element is treated as single-layered and uses two integration points through the thickness.

SOLSH190 does not support real constant input for defining layer sections.

Other Input

The default orientation for this element has the S1 (shell surface coordinate) axis aligned with the first parametric direction of the element at the center of the element and is shown as x_o in Figure 190.1: SOLSH190 Geometry (p. 1037). The axis can be defined as shown:

$$S_1 = \frac{\partial\{x\}}{\partial s} / \left(\left| \frac{\partial\{x\}}{\partial s} \right| \right)$$

where:

$$\frac{\partial\{x\}}{\partial s} = \left(\frac{1}{8}\right)\left[-\{x\}^I + \{x\}^J + \{x\}^K - \{x\}^L - \{x\}^M + \{x\}^N + \{x\}^O - \{x\}^P\right]$$

$\{x\}^I, \{x\}^J, \ldots, \{x\}^P$ = global nodal coordinates

You can reorient the default first surface direction S1 in the element reference plane (see Figure 190.1) via the **ESYS** command. You can further rotate S1 by angle THETA (in degrees) for each layer via the **SECDATA** command to create layer-wise coordinate systems. See Coordinate Systems (p. 55) for details.

Element loads are described in Nodal Loading (p. 49). Pressures may be input as surface loads on the element faces as shown by the circled numbers on Figure 190.1: SOLSH190 Geometry (p. 1037). Positive pressures act into the element.

If you specify no element body load for defining temperatures--that is, you define temperatures with commands other than **BFE**--SOLSH190 adopts an element-wise temperature pattern and requires only eight temperatures for the eight element nodes. Unspecified nodal temperatures default to the assigned

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

uniform temperature (**TUNIF**). ANSYS computes all layer interface temperatures by interpolating nodal temperatures T1 ~ T8.

Alternatively, you can input temperatures as element body loads at the corners of the outside faces of the element and at the corners of the interfaces between layers. In such a case, the element uses a layer-wise pattern. Temperatures T1, T2, T3, T4 are used for the bottom of layer 1, temperatures T5, T6, T7, T8 are used for interface corners between layers 1 and 2, and so on between successive layers, ending with temperatures at the top layer NL. If you input exactly NL + 1 temperatures, one temperature is used for the four bottom corners of each layer, and the last temperature is used for the four top corner temperatures of the top layer. The first corner temperature T1 defaults to TUNIF. If all other corner temperatures are unspecified, they default to T1. For any other input pattern, unspecified temperatures default to TUNIF.

You can use the **MP** command to define the isotropic or orthotropic elastic material properties and the **TB**,ANEL command to define anisotropic elastic material properties. Other material properties include density, damping ratios, and coefficients of thermal expansion. You may also use the **TB** command to define nonlinear material behavior such as plasticity, hyperelasticity, viscoelasticity, creep, and viscoplasticity.

KEYOPT(2) = 1 activates the internal strain enhancements to the element transverse-shear strains. With this option, the element is capable of quadratic transverse-shear strain distributions through the entire thickness of the element. At least three integration points through the thickness are required for this option. Use a shell section to define more integration points through the thickness.

KEYOPT(6) = 1 sets the element for using u-P mixed formulation. For details on the use of mixed formulation, see Applications of Mixed u-P Formulations (p. 73) in the *Element Reference*.

You can apply an initial stress state to this element via the **INISTATE** command. For more information, see Initial State in the *Basic Analysis Guide*.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

"SOLSH190 Input Summary" (p. 1039) contains a summary of element input. For a general description of element input, see Element Input (p. 43).

SOLSH190 Input Summary

Nodes

I, J, K, L, M, N, O, P

Degrees of Freedom

UX, UY, UZ

Real Constants

None

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ),
DENS, GXY, GYZ, GXZ, ALPD, BETD

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface Loads

Pressures --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperatures --

Element-wise pattern (no element body load command issued): T1, T2, T3, T4, T5, T6, T7, T8 for 8 element nodes. Temperatures at layer interface corners are computed by interpolating nodal temperatures.
Layer-wise pattern (element body load command issued): T1, T2, T3, T4 (at bottom of layer 1), T5, T6, T7, T8 (between layers 1-2); similarly for temperatures between subsequent layers, ending with temperatures at top of layer NL (4 * (NL + 1) maximum). For a one-layer element, therefore, 8 temperatures are used.

Body force densities --
The element values in the global X, Y, and Z directions.

Special Features

Birth and death
Element technology autoselect
Generalized cross-section
Initial state
Large deflection
Large strain
Linear perturbation
Nonlinear stabilization
Stress stiffening

KEYOPT(2)
Enhanced transverse-shear strains:

0 --
No enhanced transverse-shear strains (default).

1 --
Include enhanced transverse-shear strains.

KEYOPT(6)
Element formulation:

0 --
Use pure displacement formulation (default).

1 --
Use mixed u-P formulation.

KEYOPT(8)
Storage of layer data:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0 --
For multilayer elements, store data for bottom of bottom layer and top of top layer (default).

1 --
For multilayer elements, store data for top and bottom for all layers. (Before using this option, be aware that the amount of data involved can be very large.)

SOLSH190 Element Technology

SOLSH190 employs incompatible modes to enhance the accuracy in in-plane bending situations. The satisfaction of the in-plane patch test is ensured. A separate set of incompatible modes is adopted to overcome the thickness locking in bending dominant problems. The incompatible modes introduce seven internal DOFs that are inaccessible to users and condensed out at the element level.

SOLSH190 utilizes a suite of special kinematic formulations to avoid locking when the shell thickness becomes extremely small. However, due to its shell-like behavior, SOLSH190 fails to pass the patch test if the element is distorted in the thickness direction.

SOLSH190 is fully compatible with 3-D constitutive relations. Compared to classical shell elements that are based on plane stress assumptions, SOLSH190 usually gives more accurate predictions when the shell is thick.

SOLSH190 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 190.1: SOLSH190 Element Output Definitions (p. 1042)

Several items are illustrated in Figure 190.2: SOLSH190 Stress Output (p. 1042). See Element Table for Variables Identified By Sequence Number in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 190.2: SOLSH190 Stress Output

x_o = Element x-axis if ESYS is not supplied.
x = Element x-axis if ESYS is supplied.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 190.1: SOLSH190 Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
NODES	Nodes - I, J, K, L, M, N, O, P	-	Y
MAT	Material number	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	2
PRES	Pressures P1 at nodes J, I, L, K; P2 at I, J, N, M; P3 at J, K, O, N; P4 at K, L, P, O; P5 at L, I, M, P; P6 at M, N, O, P	-	Y
TEMP	T1, T2, T3, T4 at bottom of layer 1; T5, T6, T7, T8 between layers 1-2; similarly for between successive	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
	layers, ending with temperatures at top of layer NL (4 * (NL + 1) maximum)		
S:X, Y, Z, XY, YZ, XZ	Stresses	Y	Y
S:1, 2, 3	Principal stresses	-	Y
S:INT	Stress intensity	-	Y
S:EQV	Equivalent stress	-	Y
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	Y	Y
EPEL:EQV	Equivalent elastic strains [5]	-	Y
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	Y	Y
EPTH:EQV	Equivalent thermal strains [5]	-	Y
EPPL:X, Y, Z, XY, YZ, XZ	Plastic strains [6]	1	1
EPPL:EQV	Equivalent plastic strains [5]	-	1
EPCR:X, Y, Z, XY, YZ, XZ	Creep strains	1	1
EPCR:EQV	Equivalent creep strains [5]	-	1
EPTO:X, Y, Z, XY, YZ, XZ	Total mechanical strains (EPEL + EPPL + EPCR)	Y	-
EPTO:EQV	Total equivalent mechanical strains (EPEL + EPPL + EPCR)	-	-
NL:SEPL	Plastic yield stress	1	1
NL:EPEQ	Accumulated equivalent plastic strain	1	1
NL:CREQ	Accumulated equivalent creep strain	1	1
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	1	1
NL:HPRES	Hydrostatic pressure	1	1
SEND:ELASTIC, PLASTIC, CREEP	Strain energy densities	-	1
N11, N22, N12	In-plane forces (per unit length)	-	Y
M11, M22, M12	Out-of-plane moments (per unit length)	-	Y
Q13, Q23	Transverse-shear forces (per unit length)	-	Y
LOCI:X, Y, Z	Integration point locations	-	3
SVAR:1, 2, ... , N	State variables	-	4
ILSXZ	SXZ interlaminar shear stress	-	7
ILSYZ	SYZ interlaminar shear stress	-	7
ILSUM	Magnitude of the interlaminar shear stress vector	-	7
ILANG	Angle of interlaminar shear stress vector (measured from the element x-axis toward the element y-axis in degrees)	-	7

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
Sm: 11, 22, 12	Membrane stresses	-	Y
Sb: 11, 22, 12	Bending stresses	-	Y
Sp: 11, 22, 12	Peak stresses	-	Y
St: 13, 23	Averaged transverse-shear stresses	-	Y

1. Nonlinear solution, output only if the element has a nonlinear material, or if large-deflection effects are enabled (**NLGEOM**,ON) for SEND.
2. Available only at centroid as a ***GET** item.
3. Available only if **OUTRES**,LOCI is used.
4. Available only if the `UserMat` subroutine and **TB**,STATE command are used.
5. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,NUXY); for plastic and creep this value is set at 0.5.
6. For the shape memory alloy material model, transformation strains are reported as plasticity strain EPPL.
7. Available only if a valid shell section (**SECTYPE**,,SHELL) is defined for the element.

Table 190.2: SOLSH190 Item and Sequence Numbers (p. 1044) lists output available through **ETABLE** using the Sequence Number method. See Element Table for Variables Identified By Sequence Number in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 190.2: SOLSH190 Item and Sequence Numbers (p. 1044):

Name
output quantity as defined in the Table 190.1: SOLSH190 Element Output Definitions (p. 1042)

Item
predetermined Item label for **ETABLE** command

I,J,...,P
sequence number for data at nodes I, J, ..., P

Table 190.2: SOLSH190 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input									
	Item	E	I	J	K	L	M	N	O	P
P1	SMISC	-	2	1	4	3	-	-	-	-
P2	SMISC	-	5	6	-	-	8	7	-	-
P3	SMISC	-	-	9	10	-	-	12	11	-
P4	SMISC	-	-	-	13	14	-	-	16	15
P5	SMISC	-	18	-	-	17	19	-	-	20
P6	SMISC	-	-	-	-	-	21	22	23	24
THICK	SMISC	27	-	-	-	-	-	-	-	-
N11	SMISC	28	-	-	-	-	-	-	-	-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input									
	Item	E	I	J	K	L	M	N	O	P
N22	SMISC	29	-	-	-	-	-	-	-	-
N12	SMISC	30	-	-	-	-	-	-	-	-
M11	SMISC	31	-	-	-	-	-	-	-	-
M22	SMISC	32	-	-	-	-	-	-	-	-
M12	SMISC	33	-	-	-	-	-	-	-	-
Q13	SMISC	34	-	-	-	-	-	-	-	-
Q23	SMISC	35	-	-	-	-	-	-	-	-
Sm: 11	SMISC	36	-	-	-	-	-	-	-	-
Sm: 22	SMISC	37	-	-	-	-	-	-	-	-
Sm: 12	SMISC	38	-	-	-	-	-	-	-	-
Sb: 11	SMISC	39	-	-	-	-	-	-	-	-
Sb: 22	SMISC	40	-	-	-	-	-	-	-	-
Sb: 12	SMISC	41	-	-	-	-	-	-	-	-
Sp: 11 (at bottom face)	SMISC	42	-	-	-	-	-	-	-	-
Sp: 22 (at bottom face)	SMISC	43	-	-	-	-	-	-	-	-
Sp: 12 (at bottom face)	SMISC	44	-	-	-	-	-	-	-	-
Sp: 11 (at top face)	SMISC	45	-	-	-	-	-	-	-	-
Sp: 22 (at top face)	SMISC	46	-	-	-	-	-	-	-	-
Sp: 12 (at top face)	SMISC	47	-	-	-	-	-	-	-	-
St: 13	SMISC	48	-	-	-	-	-	-	-	-
St: 23	SMISC	49	-	-	-	-	-	-	-	-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quant-ity Name	ETABLE and ESOL Command Input		
	Item	Bottom of Layer i	Top of Layer NL
ILSXZ	SMISC	8 * (i - 1) + 51	8 * (NL - 1) + 52
ILSYZ	SMISC	8 * (i - 1) + 53	8 * (NL - 1) + 54
ILSUM	SMISC	8 * (i - 1) + 55	8 * (NL - 1) + 56
ILANG	SMISC	8 * (i - 1) + 57	8 * (NL - 1) + 58

SOLSH190 Assumptions and Restrictions

- Zero-volume elements are not allowed.
- Elements may be numbered either as shown in Figure 190.1: SOLSH190 Geometry (p. 1037) or may have the planes IJKL and MNOP interchanged. The element may not be twisted such that the element has two separate volumes (which occurs most frequently when the elements are not numbered properly).
- All elements must have eight nodes. You can form a prism-shaped element by defining duplicate K and L and duplicate O and P node numbers (see Degenerated Shape Elements (p. 39)).
- If you use the mixed u-P formulation (KEYOPT(6) = 1), the damped eigensolver is not supported. You must use the sparse solver (default).
- If the material of a layer is hyperelastic, the layer orientation angle has no effect.
- Using both hyperelastic and elastoplastic layers in the same element can produce unpredictable results and is not recommended.
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). Prestress effects can be activated by the **PSTRES** command.
- To obtain more accurate transverse shear results, use multiple elements though the thickness.

SOLSH190 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

INTER192

2-D 4-Node Gasket

MP ME ST <> <> <> <> <> PP EME MFS
Product Restrictions

INTER192 Element Description

INTER192 is a 2-D four-node linear interface element used for 2-D structural assembly modeling . When used with 2-D linear structural elements (such as PLANE182), INTER192 simulates gasket joints. The element can be used either as a plane element (plane stress or plane strain) or as an axisymmetric element. It is defined by four nodes having two degrees of freedom at each node: translations in the nodal x and y directions.

See Gasket Material and INTER192 in the *Mechanical APDL Theory Reference* for more details about this element.

Also see Gasket Joints Simulation in the *Structural Analysis Guide* for more details on the gasket capability in ANSYS.

Figure 192.1: INTER192 Geometry

INTER192 Input Data

The element geometry, node locations, connectivity, and the nodal coordinate system are shown in Figure 192.1: INTER192 Geometry (p. 1047). The element geometry is defined by 4 nodes, which form bottom and top lines of the element. The bottom line is defined by nodes I, J; and the top line is defined by nodes K, L. The element connectivity is defined as I, J, K, L. This element has 2 integration points. The Gauss integration scheme is used for the numerical integration.

Temperatures may be input as element body loads at the nodes. The node I temperature T(I), defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

Input the nodal forces, if any, per unit of depth for a plane analysis (except for KEYOPT(3) = 3) and on a full 360° basis for an axisymmetric analysis.

By default, the element is capable of both through-thickness and transverse shear deformations (KEYOPT(2) = 1). The inclusion of transverse shear stiffness is generally required when the interfaces between

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the gasket and the mating parts are modeled as sliding contact. However, if the interfaces are modeled with a matching mesh method (that is, with coincident nodes), ANSYS, Inc. recommends using through-thickness deformation only (KEYOPT(2) = 0) to avoid unnecessary in-plane interaction between the gasket and the mating parts.

The following table summarizes the element input. See the Element Input (p. 43) section in the *Element Reference* for a general description of element input.

INTER192 Input Summary

Nodes
I, J, K, L

Degrees of Freedom
UX, UY

Real Constants

None, if KEYOPT(3) = 0, 1, or 2
THK - Plane stress with thickness, if KEYOPT(3) = 3

Material Properties
TB command: Gasket material

MP command: BETD, ALPX (or CTEX *or* THSX)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L)

Special Features

Linear perturbation

KEYOPT(2)
Element deformation:

0 --
Through-thickness deformation only

1 --
Through-thickness and transverse shear deformation (default)

KEYOPT(3)
Element behavior:

0 --
Plane stress

1 --
Axisymmetric

2 --
Plane strain (Z strain = 0.0)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3 --
Plane stress with thickness (THK) real constant input

INTER192 Output Data

The solution output associated with the element is in two forms:

- Nodal items such as nodal displacements are included in the overall nodal solution.
- Element items such as stresses and closures are element outputs as shown in Table 192.1: INTER192 Element Output Definitions (p. 1049).

The output directions for element items are parallel to the local element coordinate system based on the element midplane as illustrated in Figure 192.2: INTER192 Stress Output (p. 1049). See Gasket Material in the *Mechanical APDL Theory Reference* for details.

A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to review results.

Figure 192.2: INTER192 Stress Output

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 192.1: INTER192 Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Node connectivity - I, J, K, L	-	Y
MAT	Material number	-	Y
TEMP	Temperatures T(I), T(J), T(K), T(L)	-	Y
GKS:X, (XY)	Stress (also gasket pressure)	Y	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
GKD:X, (XY)	Total closure	Y	Y
GKDI:X, (XY)	Total inelastic closure	Y	Y
GKTH:X, (XY)	Thermal closure	Y	Y

INTER192 Assumptions and Restrictions

- This element is not supported for initial stress.
- Pressure as a type of surface load on element faces is not supported by this element.
- This element is based on the local coordinate system. **ESYS** is not permitted.
- This element is only available for static analyses.

INTER192 Product Restrictions

There are no product-specific restrictions on this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

INTER193

2-D 6-Node Gasket

MP ME ST <> <> <> <> <> PP EME MFS
Product Restrictions

INTER193 Element Description

INTER193 is a 2-D 6-node quadratic interface element used for 2-D structural assembly modeling. When used in conjunction with 2-D quadratic structural elements (such as PLANE183), INTER193 is used to simulate gasket joints. The element can be used either as a plane element (plane stress or plane strain) or as an axisymmetric element. It is defined by six nodes having two degrees of freedom at each node: translations in the nodal x and y directions.

See Gasket Material and INTER193 in the *Mechanical APDL Theory Reference* for more details about this element.

Also see Gasket Joints Simulation in the *Structural Analysis Guide* for more details on the gasket capability in ANSYS.

Figure 193.1: INTER193 Geometry

INTER193 Input Data

The element geometry, node locations, connectivity, and the nodal coordinate system are shown in Figure 193.1: INTER193 Geometry (p. 1051). The element geometry is defined by 6 nodes, which form bottom and top lines of the element. The bottom line is defined by nodes I, J, M; and the top line is defined by nodes K, L, O. The element connectivity is defined as I, J, K, L, M, O. This element has 2 integration points. Dropping mid side nodes M or O is not permitted.

Temperatures may be input as element body loads at the nodes. The node I temperature T(I), defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

Input the nodal forces, if any, per unit of depth for a plane analysis (except for KEYOPT(3) = 3) and on a full 360° basis for an axisymmetric analysis.

By default, the element adopts a full-integration scheme (KEYOPT(4) = 2), and is capable of both through-thickness and transverse shear deformations (KEYOPT(2) = 1). The full-integration scheme and the inclu-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

sion of transverse shear stiffness are generally required when the interfaces between the gasket and the mating parts are modeled as sliding contact. However, if the interfaces are modeled with a matching mesh method (that is, with coincident nodes), ANSYS, Inc. recommends using the reduced integration scheme (KEYOPT(4) = 0) and through-thickness deformation only (KEYOPT(2) = 0) to achieve better efficiency and to avoid unnecessary in-plane interaction between the gasket and the mating parts.

The next table summarizes the element input. See Element Input (p. 43) in the *Element Reference* for a general description of element input.

INTER193 Input Summary

Nodes
I, J, K, L, M, , O

Degrees of Freedom
UX, UY

Real Constants

None, if KEYOPT(3) = 0, 1, or 2
THK - Plane stress with thickness, if KEYOPT(3) = 3

Material Properties
TB command: Gasket material

MP command: BETD, ALPX (or CTEX *or* THSX)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(O)

Special Features

Linear perturbation

KEYOPT(2)
Element deformation:

0 --
Through-thickness deformation only

1 --
Through-thickness and transverse shear deformation (default)

KEYOPT(3)
Element behavior:

0 --
Plane stress

1 --
Axisymmetric

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2 --
Plane strain (Z strain = 0.0)

3 --
Plane stress with thickness (THK) real constant input

KEYOPT(4)
Element technology:

0 --
Uniform reduced integration

2 --
Full integration (default)

INTER193 Output Data

The solution output associated with the element is in two forms:

- Nodal items such as nodal displacements are included in the overall nodal solution.
- Element items such as stresses and closures are element outputs as shown in Table 193.1: INTER193 Element Output Definitions (p. 1054).

The output directions for element items are parallel to the local element coordinate system based on the element midplane as illustrated in Figure 193.2: INTER193 Stress Output (p. 1053). See Gasket Material in the *Mechanical APDL Theory Reference* for details.

A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to review results.

Figure 193.2: INTER193 Stress Output

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 193.1: INTER193 Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Node connectivity - I, J, K, L, M, O	-	Y
MAT	Material number	-	Y
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(O)	-	Y
GKS:X, (XY)	Stress (also gasket pressure)	Y	Y
GKD:X, (XY)	Total closure	Y	Y
GKDI:X, (XY)	Total inelastic closure	Y	Y
GKTH:X, (XY)	Thermal closure	Y	Y

INTER193 Assumptions and Restrictions

- This element is not supported for initial stress.
- Pressure as a type of surface load on element faces is not supported by this element.
- This element is based on the local coordinate system. **ESYS** is not permitted.
- This element is only available for static analyses.

INTER193 Product Restrictions

There are no product-specific restrictions on this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

INTER194

3-D 16-Node Gasket

MP ME ST <> <> <> <> <> PP EME MFS
Product Restrictions

INTER194 Element Description

INTER194 is a 3-D 16-node quadratic interface element. When used with 3-D quadratic structural elements (SOLID96, SOLID186, and SOLID187), INTER194 simulates gasket joints. It is defined by 16 nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions.

See Gasket Material and INTER194 in the *Mechanical APDL Theory Reference* for more details about this element.

Also see Gasket Joints Simulation in the *Structural Analysis Guide* for more details about gasket simulation.

Figure 194.1: INTER194 Geometry

INTER194 Input Data

The element geometry, node locations, connectivity, and the nodal coordinate system are shown in Figure 194.1: INTER194 Geometry (p. 1055). The element geometry is defined by 16 nodes, which form bottom and top surfaces of the element. The bottom surface is defined by nodes, I, J, K, L, Q, R, S, T; and the top surface is defined by nodes, M, N, O, P, U, V, W, X. As shown, the element connectivity is defined as I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X. The element is degenerated to a wedge (prism) element, when K=L=S and O=P=W, as shown in Figure 194.2: INTER194 3-D 16-Node Degenerated Quadratic Interface (p. 1056).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 194.2: INTER194 3-D 16-Node Degenerated Quadratic Interface

For the degenerated element, 3 integration points are used for numerical integration. The degenerated element can be used in conjunction with 10-node solid tetrahedral elements (such as SOLID187). Dropping any or some of midside nodes, Q, R, S, T, U, V, W, X is not permitted.

Temperatures may be input as element body loads at the nodes. The node I temperature T(I), defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

By default, the element adopts a full-integration scheme (KEYOPT(4) = 2) and is capable of both through-thickness and transverse shear deformations (KEYOPT(2) =1). The full-integration scheme and the inclusion of transverse shear stiffness are generally required when the interfaces between the gasket and the mating parts are modeled as sliding contact. However, if the interfaces are modeled with a matching mesh method (that is, with coincident nodes), ANSYS, Inc. recommends using the reduced-integration scheme (KEYOPT(4) = 0) and through-thickness deformation only (KEYOPT(2) = 0) to achieve better efficiency and to avoid unnecessary in-plane interaction between the gasket and the mating parts.

The following table summarizes the element input. See Element Input (p. 43) in the *Element Reference* for a general description of element input.

INTER194 Input Summary

Nodes

I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X

Degrees of Freedom

UX, UY, UZ

Real Constants

None

Material Properties

TB command: Gasket material

MP command: BETD, ALPX (or CTEX *or* THSX)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Body Loads

Temperatures --
T(I), T(J), T(K), T(L) T(M), T(N), T(O), T(P), T(Q), T(R), T(S), T(T), T(U), T(V), T(W), T(X)

Special Features

Linear perturbation

KEYOPT(2)
Element deformation:

0 --
Through-thickness deformation only

1 --
Through-thickness and transverse shear deformation (default)

KEYOPT(4)
Element technology:

0 --
Uniform reduced integration

2 --
Full integration (default)

INTER194 Output Data

The solution output associated with the element is in two forms:

- Nodal items such as nodal displacements are included in the overall nodal solution.
- Element items such as stresses and closures are element outputs as shown in Table 194.1: INTER194 Element Output Definitions (p. 1058).

The output directions for element items are parallel to the local element coordinate system based on the element midplane as illustrated in Figure 194.3: INTER194 Stress Output (p. 1058). See Gasket Material in the *Mechanical APDL Theory Reference* for details.

A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to review results.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 194.3: INTER194 Stress Output

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 194.1: INTER194 Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Node connectivity - I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X	-	Y
MAT	Material number	-	Y
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P), T(Q), T(R), T(S), T(T), T(U), T(V), T(W), T(X)	-	Y
GKS:X, (XY, XZ)	Stress (also gasket pressure)	Y	Y
GKD:X, (XY, XZ)	Total closure	Y	Y
GKDI:X, (XY, XZ)	Total inelastic closure	Y	Y
GKTH:X, (XY, XZ)	Thermal closure	Y	Y

INTER194 Assumptions and Restrictions

- This element is not supported for initial stress.
- Pressure as a type of surface load on element faces is not supported by this element.
- This element is based on the local coordinate system. **ESYS** is not permitted.
- This element is only available for static analyses.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

INTER194 Product Restrictions

There are no product-specific restrictions on this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

INTER195

3-D 8-Node Gasket

MP ME ST <> <> <> <> <> PP EME MFS
Product Restrictions

INTER195 Element Description

INTER195 is a 3-D 8-node linear interface element. When used with 3-D linear structural elements (SOLID65, SOLID185, SOLSH190, SOLID272, SOLID273, and SOLID285), INTER195 simulates gasket joints. It is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions.

See Gasket Material and INTER195 in the *Mechanical APDL Theory Reference* for more details about this element.

Also see Gasket Joints Simulation in the *Structural Analysis Guide* for more details on this ANSYS capability.

Figure 195.1: INTER195 Geometry

INTER195 Input Data

The element geometry, node locations, connectivity, and the nodal coordinate system are shown in Figure 195.1: INTER195 Geometry (p. 1061). The element geometry is defined by 8 nodes, which form bottom and top surfaces of the element. The bottom surface is defined by nodes, I, J, K, L; and the top surface is defined by nodes, M, N, O, P. As shown, the element connectivity is defined as I, J, K, L, M, N, O, P.

Temperatures may be input as element body loads at the nodes. The node I temperature T(I), defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

By default, the element is capable of both through-thickness and transverse shear deformations (KEYOPT(2) = 1). The inclusion of transverse shear stiffness is generally required when the interfaces between the gasket and the mating parts are modeled as sliding contact. However, if the interfaces are modeled with a matching mesh method (that is, with coincident nodes), ANSYS, Inc. recommends using through-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

thickness deformation only (KEYOPT(2) = 0) to avoid unnecessary in-plane interaction between the gasket and the mating parts.

The next table summarizes the element input. See Element Input (p. 43) in the *Element Reference* for a general description of element input.

INTER195 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
UX, UY, UZ

Real Constants
None

Material Properties
TB command: Gasket material

MP command: BETD, ALPX (or CTEX *or* THSX)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

Special Features

Linear perturbation

KEYOPT(2)
Element deformation:

0 --
Through-thickness deformation only

1 --
Through-thickness and transverse shear deformation (default)

INTER195 Output Data

The solution output associated with the element is in two forms:

- Nodal items such as nodal displacements are included in the overall nodal solution.
- Element items such as stresses and closures are element outputs as shown in Table 195.1: INTER195 Element Output Definitions (p. 1063).

The output directions for element items are parallel to the local element coordinate system based on the element midplane as illustrated in Figure 195.2: INTER195 Stress Output (p. 1063). See Gasket Material in the *Mechanical APDL Theory Reference* for details.

A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to review results.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 195.2: INTER195 Stress Output

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 195.1: INTER195 Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Node connectivity - I, J, K, L, M, N, O, P	-	Y
MAT	Material number	-	Y
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	-	Y
GKS:X, (XY, XZ)	Stress (also gasket pressure)	Y	Y
GKD:X, (XY, XZ)	Total closure	Y	Y
GKDI:X, (XY, XZ)	Total inelastic closure	Y	Y
GKTH:X, (XY, XZ)	Thermal closure	Y	Y

INTER195 Assumptions and Restrictions

- This element is not supported for initial stress.
- Pressure as a type of surface load on element faces is not supported by this element.
- This element is based on the local coordinate system. **ESYS** is not permitted.
- This element is only available for static analyses.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

INTER195 Product Restrictions

There are no product-specific restrictions on this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MESH200

Meshing Facet

MP ME ST PR PRN <> EM DY PP EME MFS
Product Restrictions

MESH200 Element Description

MESH200 is a "mesh-only" element, contributing nothing to the solution. This element can be used for the following types of operations:

- Multistep meshing operations, such as extrusion, that require a lower dimensionality mesh be used for the creation of a higher dimensionality mesh
- Line-meshing in 2-D or 3-D space with or without midside nodes,
- Area-meshing or volume-meshing in 3-D space with triangles, quadrilaterals, tetrahedra, or bricks, with or without midside nodes.
- Temporary storage of elements when the analysis physics has not yet been specified.

MESH200 may be used in conjunction with any other ANSYS element types. After it is no longer needed, it can be deleted (cleared), or can be left in place. Its presence will not affect solution results.

MESH200 elements can be changed into other element types using **EMODIF**.

Figure 200.1: MESH200 Geometry

KEYOPT (1) = 0
2-D line with 2 nodes

KEYOPT (1) = 1
2-D line with 3 nodes

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT (1) = 2
3-D line with 2 nodes

KEYOPT (1) = 3
3-D line with 3 nodes

KEYOPT (1) = 4
3-D triangle with 3 nodes

KEYOPT (1) = 5
3-D triangle with 6 nodes

KEYOPT (1) = 6
3-D quadrilateral with 4 nodes

KEYOPT (1) = 7
3-D quadrilateral with 8 nodes

KEYOPT (1) = 8
tetrahedron with 4 nodes

KEYOPT (1) = 9
tetrahedron with 10 nodes

KEYOPT (1) = 10
brick with 8 nodes

KEYOPT (1) = 11
brick with 20 nodes

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MESH200 Input Data

The permissible geometry and node locations for this element are shown in Figure 200.1: MESH200 Geometry (p. 1065). The element is defined by two to twenty nodes. It has no degrees of freedom, material properties, real constants, or loadings.

"MESH200 Input Summary" (p. 1067) summarizes the element input. See Element Input (p. 43) in the *Element Reference* for a general description of element input.

MESH200 Input Summary

Nodes

I, J if KEYOPT (1) = 0, 2-D line with 2 nodes
I, J, K if KEYOPT (1) = 1, 2-D line with 3 nodes
I, J if KEYOPT (1) = 2, 3-D line with 2 nodes
I, J, K if KEYOPT (1) = 3, 3-D line with 3 nodes
I, J, K if KEYOPT (1) = 4, 3-D triangle with 3 nodes
I, J, K, L, M, N if KEYOPT (1) = 5, 3-D triangle with 6 nodes
I, J, K, L if KEYOPT (1) = 6, 3-D quadrilateral with 4 nodes
I, J, K, L, M, N, O, P if KEYOPT (1) = 7, 3-D quadrilateral with 8 nodes
I, J, K, L if KEYOPT (1) = 8, tetrahedron with 4 nodes
I, J, K, L, M, N, O, P, Q, R if KEYOPT (1) = 9, tetrahedron with 10 nodes
I, J, K, L, M, N, O, P if KEYOPT (1) = 10, brick with 8 nodes
I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B if KEYOPT (1) = 11, brick with 20 nodes

Degrees of Freedom
None

Real Constants
None

Material Properties
None

Surface Loads
None

Body Loads
None

Special Features
Linear perturbation

KEYOPT(1)
Element shape and number of nodes:

0 --
2-D line with 2 nodes

1 --
2-D line with 3 nodes

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2 --
3-D line with 2 nodes

3 --
3-D line with 3 nodes

4 --
3-D triangle with 3 nodes

5 --
3-D triangle with 6 nodes

6 --
3-D quadrilateral with 4 nodes

7 --
3-D quadrilateral with 8 nodes

8 --
tetrahedron with 4 nodes

9 --
tetrahedron with 10 nodes

10 --
brick with 8 nodes

11 --
brick with 20 nodes

KEYOPT(2)
Element shape testing:

0 --
Shape testing is done (default)

1 --
No shape testing is done for this element

MESH200 Output Data

This element has no output data.

MESH200 Assumptions and Restrictions

- When this element is a triangle or quadrilateral, it is shape-tested in the same manner as an equivalent "non-structural shell". When it is a tetrahedron or brick, it is shape-tested like a SOLID185. This is so that meshing will work to create well-shaped elements. If KEYOPT(2) = 1, no shape testing is done for this element type.
- MESH200 elements may not be active during result contour plotting (**/POST1**, **PLNSOL**, or **PLESOL**). The elements are automatically unselected during either operation.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MESH200 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

FOLLW201

Follower Load

MP ME ST PR PRN DS <> <> PP EME MFS
Product Restrictions

FOLLW201 Element Description

FOLLW201 is a one-node 3-D element that can be overlaid onto an existing node with physical rotation degrees of freedom. The element specifies external forces and moments which follow the deformation of a structure in a nonlinear analysis. FOLLW201 contributes follower load stiffness terms in a geometrically nonlinear analysis (**NLGEOM**,ON).

Figure 201.1: FOLLW201 Geometry

FOLLW201 overlaid on a node shared by shell or beam elements. The element has two faces: face 1 for specifying magnitude of force and face 2 for specifying magnitude of moment.

FOLLW201 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 201.1: FOLLW201 Geometry (p. 1071). The element is defined by a single node. The node has three translational and rotational degrees of freedom each. The element may be defined only at those nodes which are associated with structural elements having three translational and rotational degrees of freedom; a singularity will result if the element is used in any other way.

Real constants of the element specify the direction of the force/moment vectors, and the element load command **SFE** specifies the magnitude of force/moment.

Element loads are described in Nodal Loading (p. 49). The vectors defined by real constants will evolve with deformation (follow the displacements) in a geometrically nonlinear analysis.

KEYOPT(1) = 1 provides a means of specifying complex (real and imaginary) point loads via an element specification. You can consider it as a nodal point equivalent of surface-effect elements (such as SURF154). When KEYOPT(1) = 1 (intended primarily for use in the ANSYS Workbench interface), the direction of the load is not updated for geometrically nonlinear analyses, and the element is renamed to (and appears

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in the output as) CLOAD201. In this case, the applied load is the same as that generated via the **F** command, except that loads can be simultaneously applied in multiple directions to one node. The KEYOPT(2) = 1 setting, which can only be used with KEYOPT(1) = 1, enables use of the CLOAD201 element on nodes that may or may not have active rotational degrees of freedom.

With the exception of follower load effects, the element contributes nothing to the stiffness matrix. By default, follower (pressure) load stiffness effects are included in a geometrically nonlinear analysis. The stiffness contribution is usually unsymmetrical and may require an unsymmetrical solution option (**NROPT**,UNSYM).

"FOLLW201 Input Summary" (p. 1072) contains a summary of the element input. See Element Input (p. 43) in the *Element Reference* for a general description of element input.

FOLLW201 Input Summary

Nodes

I

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants

FX - Cosine of the angle between force vector and global X direction
FY - Cosine of the angle between force vector and global Y direction
FZ - Cosine of the angle between force vector and global Z direction
MX - Cosine of the angle between moment vector and global X direction
MY - Cosine of the angle between moment vector and global Y direction
MZ - Cosine of the angle between moment vector and global Z direction

Material Properties

None

Surface Loads

face 1 (force magnitude)
face 2 (moment magnitude)

Body Loads

None

Special Features

Birth and death
Large deflection
Linear perturbation

KEYOPT(1)

Direction load:

0 --

Updated direction load (default)

1 --

Constant direction load (intended primarily for use in the ANSYS Workbench interface)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(2)
Degrees of freedom control:

0 --
Use all (UX, UY, UZ, ROTX, ROTY, ROTZ) degrees of freedom (default)

1 --
Use UX, UY, and UZ degrees of freedom only (valid only when KEYOPT(1) = 1)

FOLLW201 Output Data

The Element Outputs consist of updated direction cosines of the force/moment vectors as Miscellaneous quantities (SMISC). No other output is provided.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

The following table lists output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information.

Name
output quantity as defined above

Item
predetermined item label for **ETABLE** command

I
sequence number for data at node I

Table 201.1: FOLLW201 Item and Sequence Numbers

Name	Item	Location
FX	SMISC	1
FY	SMISC	2
FZ	SMISC	3
MX	SMISC	4
MY	SMISC	5
MZ	SMISC	6

FOLLW201 Assumptions and Restrictions

- Follower load stiffening is ignored in geometrically linear analyses (**NLGEOM**,OFF), which is equivalent to the normal specification of forces and moments (**F**).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The element must be overlaid on a node having existing physical stiffness contributions (from other shell or beam elements).
- Follower load effects are nonconservative. They often introduce dynamics instability issues (such as flutter) which may cause convergence difficulties.

FOLLW201 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

INTER202

2-D 4-Node Cohesive

MP ME ST <> <> <> <> <> PP EME MFS
Product Restrictions

INTER202 Element Description

INTER202 is a 2-D 4-node linear interface element used for 2-D structural assembly modeling. When used with 2-D linear structural elements (such as PLANE182), INTER202 simulates the interface surfaces and the subsequent delamination process, where the separation is represented by an increasing displacement between nodes, within the interface element itself, that are initially coincident. The element can be used as either a plane element (plane stress or plane strain) or as an axisymmetric element. It is defined by four nodes having two degrees of freedom at each node: translations in the nodal x and y directions.

See Cohesive Zone Material (CZM) Model and INTER202 in the *Mechanical APDL Theory Reference* for more information about this element. Also see Cohesive Material Law in the *Material Reference*.

For more information about the interface failure/delamination capability, see Interface Delamination and Failure Simulation in the *Structural Analysis Guide* .

Figure 202.1: INTER202 Geometry

INTER202 Input Data

The element geometry, node locations, connectivity, and the nodal coordinate system are shown in Figure 202.1: INTER202 Geometry (p. 1075). The element geometry is defined by 4 nodes, which form bottom and top lines of the element. The bottom line is defined by nodes I, J; and the top line is defined by nodes K, L. The element connectivity is defined as I, J, K, L. This element has 2 integration points. The Gauss integration scheme is used for the numerical integration.

INTER202 is used to simulate interfacial decohesion with the cohesive zone model along an interface defined by this element. At the outset of the simulation, nodes I,L and J,K are coincident, both with each other and with the corresponding nodes in the adjacent structural elements. The subsequent separation of the adjacent elements (usually defined contiguously as components) is represented by an increasing displacement between the nodes within this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

INTER202 can also be used to simulate interfacial delamination of laminate composite and general crack growth with VCCT. For more information, see VCCT-Based Crack Growth Simulation in the *Structural Analysis Guide*.

Temperatures may be input as element body loads at the nodes. The node I temperature T(I), defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

Input the nodal forces, if any, per unit of depth for a plane analysis (except for KEYOPT(3) = 3) and on a full 360° basis for an axisymmetric analysis.

The next table summarizes the element input. See the Element Input (p. 43) section in the *Element Reference* for a general description of element input.

INTER202 Input Summary

Nodes
I, J, K, L

Degrees of Freedom
UX, UY

Real Constants

None, if KEYOPT(3) = 0, 1, or 2
THK - Plane stress with thickness, if KEYOPT(3) = 3

Material Properties
TB command: Cohesive zone material

Body Loads

Temperatures --
T(I), T(J), T(K), T(L)

The temperature is used only to evaluate the material properties.

Special Features
None

KEYOPT(2)
Element option:

0 --
Used with cohesive zone material (**TB**,CZM)

1 --
Multipoint constraint (MPC) option, used for crack growth simulation with VCCT technology.

KEYOPT(3)
Element behavior:

0 --
Plane stress

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
Axisymmetric

2 --
Plane strain (Z strain = 0.0)

3 --
Plane stress with thickness (THK) real constant input

INTER202 Output Data

The solution output associated with the element is in two forms:

- Nodal items such as nodal displacements are included in the overall nodal solution.
- Element items such as tractions and separations are element outputs as shown in Table 202.1: INTER202 Element Output Definitions (p. 1077).

The output directions for element items are parallel to the local element coordinate system based on the element midplane, as illustrated in Figure 202.2: INTER202 Stress Output (p. 1077). See Cohesive Zone Model in the *Mechanical APDL Theory Reference* for details.

A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to review results.

Figure 202.2: INTER202 Stress Output

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 202.1: INTER202 Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Node connectivity - I, J, K, L	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
MAT	Material number	-	Y
TEMP	Temperatures T(I), T(J), T(K), T(L)	-	Y
SS:X, (XY)	Interface Traction (stress)	Y	Y
SD:X, (XY)	Interface Separation (displacement)	Y	Y

INTER202 Assumptions and Restrictions

- This element is not supported for initial stress.
- Pressure as a type of surface load on element faces is not supported by this element.
- This element is based on the local coordinate system. **ESYS** is not permitted.
- This element is only available for static analyses.
- This element cannot be used in a distributed solution.

INTER202 Product Restrictions

There are no product-specific restrictions on this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

INTER203

2-D 6-Node Cohesive

MP ME ST <> <> <> <> <> PP EME MFS
Product Restrictions

INTER203 Element Description

INTER203 is a 2-D 6-node quadratic interface element used for 2-D structural assembly modeling. When used with 2-D quadratic structural elements (such as PLANE183), INTER203 simulates the interface surfaces and the subsequent delamination process, where the separation is represented by an increasing displacement between nodes, within the interface element itself, that are initially coincident. The element can be used either as a plane element (plane stress or plane strain) or as an axisymmetric element. It is defined by six nodes having two degrees of freedom at each node: translations in the nodal x and y directions.

See Cohesive Zone Material Model and INTER203 in the *Mechanical APDL Theory Reference* for more details about this element.

Also see Interface Delaminaton and Failure Simulation in the *Structural Analysis Guide* for more details on the interface failure/delamination capability in ANSYS.

Figure 203.1: INTER203 Geometry

INTER203 Input Data

The element geometry, node locations, connectivity, and the nodal coordinate system are shown in Figure 203.1: INTER203 Geometry (p. 1079). The element geometry is defined by 6 nodes, which form bottom and top lines of the element. The bottom line is defined by nodes I, J, M; and the top line is defined by nodes K, L, O. This element has 2 integration points. Dropping mid side nodes M or O is not permitted.

INTER203 is used to simulate a separation along an interface defined by this element. At the outset of your simulation, nodes I,L, nodes M,O and nodes J,K are coincident, with each other, and with the corresponding nodes in the adjacent structural elements. The subsequent separation of the adjacent elements (usually defined contiguously as components) is represented by an increasing displacement between the initially coincident nodes within this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Temperatures may be input as element body loads at the nodes. The node I temperature T(I), defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

Input the nodal forces, if any, per unit of depth for a plane analysis (except for KEYOPT(3) = 3) and on a full 360° basis for an axisymmetric analysis.

The next table summarizes the element input. See Element Input (p. 43) in the *Element Reference* for a general description of element input.

INTER203 Input Summary

Nodes
I, J, K, L, M, , O

Degrees of Freedom
UX, UY

Real Constants

None, if KEYOPT(3) = 0, 1, or 2
THK - Plane stress with thickness, if KEYOPT(3) = 3

Material Properties
TB command: Cohesive zone material

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(O)

Temperature is used only to evaluate the material properties.

Special Features
None

KEYOPT(3)
Element behavior:

0 --
Plane stress

1 --
Axisymmetric

2 --
Plane strain (Z strain = 0.0)

3 --
Plane stress with thickness (THK) real constant input

INTER203 Output Data

The solution output associated with the element is in two forms:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Nodal items such as nodal displacements are included in the overall nodal solution.
- Element items such as tractions and separations are element outputs as shown in Table 203.1: INTER203 Element Output Definitions (p. 1081).

The output directions for element items are parallel to the local element coordinate system based on the element midplane as illustrated in Figure 203.2: INTER203 Stress Output (p. 1081). See Cohesive Zone Material Model and INTER203 in the *Mechanical APDL Theory Reference* for details.

A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to review results.

Figure 203.2: INTER203 Stress Output

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 203.1: INTER203 Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Node connectivity - I, J, K, L, M, O	-	Y
MAT	Material number	-	Y
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(O)	-	Y
SS:X, (XY)	Interface Traction (Stress)	Y	Y
SD:X, (XY)	Interface Separation	Y	Y

INTER203 Assumptions and Restrictions

- This element is not supported for initial stress.
- Pressure as a type of surface load on element faces is not supported by this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- This element is based on the local coordinate system. **ESYS** is not permitted.
- This element is only available for static analyses.
- This element cannot be used in a distributed solution.

INTER203 Product Restrictions

There are no product-specific restrictions on this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

INTER204

3-D 16-Node Cohesive

MP ME ST <> <> <> <> <> PP EME MFS
Product Restrictions

INTER204 Element Description

INTER204 is a 3-D 16-node quadratic interface element. When used with 3-D quadratic structural elements (SOLID186, and SOLID187), INTER204 simulates an interface between two surfaces and the subsequent delamination process, where the separation is represented by an increasing displacement between nodes, within the interface element itself. The nodes are initially coincident.

INTER204 is defined by 16 nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions.

See Cohesive Zone Material Model and INTER204 in the *Mechanical APDL Theory Reference* for more details about this element.

Also see Interface Delaminaton and Failure Simulation in the *Structural Analysis Guide* for more details about the interface failure/delamination capability.

Figure 204.1: INTER204 Geometry

INTER204 Input Data

The element geometry, node locations, connectivity, and the nodal coordinate system are shown in Figure 204.1: INTER204 Geometry (p. 1083). The element geometry is defined by 16 nodes, which form bottom and top surfaces of the element. The bottom surface is defined by nodes, I, J, K, L, Q, R, S, T; and the top surface is defined by nodes, M, N, O, P, U, V, W, X. The element may be degenerated to a wedge (prism) element, by setting nodes K=L=S and O=P=W, as shown in Figure 204.2: INTER204 3-D 16-Node Degenerated Quadratic Interface (p. 1084).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 204.2: INTER204 3-D 16-Node Degenerated Quadratic Interface

For the degenerated element, 3 integration points are used for numerical integration. The degenerated element can be used in conjunction with 10-node solid tetrahedral elements (SOLID187). Dropping any or some of the midside nodes, Q, R, S, T, U, V, W, X is not permitted.

Temperatures may be input as element body loads at the nodes. The node I temperature T(I), defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

The next table summarizes the element input. See Element Input (p. 43) in the *Element Reference* for a general description of element input.

INTER204 Input Summary

Nodes

I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X

Degrees of Freedom

UX, UY, UZ

Real Constants

None

Material Properties

TB command: Cohesive zone material

Body Loads

Temperatures --

T(I), T(J), T(K), T(L) T(M), T(N), T(O), T(P), T(Q), T(R), T(S), T(T), T(U), T(V), T(W), T(X)

Special Features

None

KEYOPTS

None

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

INTER204 Output Data

The solution output associated with the element is in two forms:

- Nodal items such as nodal displacements are included in the overall nodal solution.
- Element items such as tractions and separations are element outputs as shown in Table 204.1: INTER204 Element Output Definitions (p. 1085).

The output directions for element items are parallel to the local element coordinate system based on the element midplane as illustrated in Figure 204.3: INTER204 Stress Output (p. 1085). See Cohesive Zone Material in the *Mechanical APDL Theory Reference* for details.

A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to review results.

Figure 204.3: INTER204 Stress Output

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 204.1: INTER204 Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Node connectivity - I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X	-	Y
MAT	Material number	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P), T(Q), T(R), T(S), T(T), T(U), T(V), T(W), T(X)	-	Y
SS:X, (XY, XZ)	Interface traction (stress)	Y	Y
SD:X, (XY, XZ)	Interface Separation	Y	Y

INTER204 Assumptions and Restrictions

- This element is not supported for initial stress.
- Pressure as a type of surface load on element faces is not supported by this element.
- This element is based on the local coordinate system. **ESYS** is not permitted.
- This element is only available for static analyses.
- This element cannot be used in a distributed solution.

INTER204 Product Restrictions

There are no product-specific restrictions on this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

INTER205

3-D 8-Node Cohesive

MP ME ST <> <> <> <> <> PP EME MFS
Product Restrictions

INTER205 Element Description

INTER205 is a 3-D 8-node linear interface element. When used with 3-D linear structural elements (SOLID185, SOLSH190, SOLID272, SOLID273, and SOLID285), INTER205 simulates an interface between two surfaces and the subsequent delamination process, where the separation is represented by an increasing displacement between nodes, within the interface element itself. The nodes are initially coincident.

The element is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions.

See Cohesive Zone Material (CZM) Model and INTER205 in the *Mechanical APDL Theory Reference* for more information about this element. Also see Cohesive Material Law in the *Material Reference*.

For more information about the interface failure/delamination capability, see Interface Delamination and Failure Simulation in the *Structural Analysis Guide*.

Figure 205.1: INTER205 Geometry

INTER205 Input Data

The element geometry, node locations, connectivity, and the nodal coordinate system are shown in Figure 205.1: INTER205 Geometry (p. 1087). The element geometry is defined by 8 nodes, which form bottom and top surfaces of the element. The bottom surface is defined by nodes, I, J, K, L; and the top surface is defined by nodes, M, N, O, P.

INTER205 is used to simulate interfacial decohesion with the cohesive zone model along an interface defined by this element. At the outset of the simulation, nodes pairs I and M, J and N, K and O, and L and P are coincident, both with each other and with the corresponding nodes in the adjacent structural

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

elements. The subsequent separation of the adjacent elements (usually defined contiguously as components) is represented by an increasing displacement between the nodes within this element.

INTER205 can also be used to simulate interfacial delamination of laminate composite and general crack growth with VCCT. For more information, see VCCT-Based Crack Growth Simulation in the *Structural Analysis Guide*.

Temperatures may be input as element body loads at the nodes. The node I temperature T(I), defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input pattern, unspecified temperatures default to TUNIF.

The next table summarizes the element input. See Element Input (p. 43) in the *Element Reference* for a general description of element input.

INTER205 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
UX, UY, UZ

Real Constants
None

Material Properties
TB command: Cohesive zone material

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

The temperature is used only to evaluate the material properties.

Special Features
None

KEYOPT(2)
Element option:

0 --
Used with cohesive zone material (**TB**,CZM)

1 --
Multipoint constraint (MPC) option, used for crack growth simulation with VCCT technology.

INTER205 Output Data

The solution output associated with the element is in two forms:

- Nodal items such as nodal displacements are included in the overall nodal solution.
- Element items such as tractions and separations are element outputs as shown in Table 205.1: INTER205 Element Output Definitions (p. 1089).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The output directions for element items are parallel to the local element coordinate system based on the element midplane as illustrated in Figure 205.2: INTER205 Stress Output (p. 1089). See Gasket Material in the *Mechanical APDL Theory Reference* for details.

A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to review results.

Figure 205.2: INTER205 Stress Output

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 205.1: INTER205 Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Node connectivity - I, J, K, L, M, N, O, P	-	Y
MAT	Material number	-	Y
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	-	Y
SS:X, (XY, XZ)	Interface traction (stress)	Y	Y
SD:X, (XY, XZ)	Interface separation	Y	Y

INTER205 Assumptions and Restrictions

- This element is not supported for initial stress.
- Pressure as a type of surface load on element faces is not supported by this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- This element is based on the local coordinate system. **ESYS** is not permitted.
- This element is only available for static analyses.
- This element cannot be used in a distributed solution.

INTER205 Product Restrictions

There are no product-specific restrictions on this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SHELL208

2-Node Axisymmetric Shell

MP ME ST PR PRN <> <> <> PP EME MFS
Product Restrictions

SHELL208 Element Description

The SHELL208 element is suitable for modeling thin to moderately thick axisymmetric shell structures, such as oil tanks, pipes, and cooling towers. It is a two-node element with three degrees of freedom at each node: translations in the x, and y directions, and rotation about the z-axis. A fourth translational degree of freedom in z direction can be included to model uniform torsion (KEYOPT(2) = 1). When the membrane option is used, the rotational degree of freedom is excluded. An extra internal node is available via KEYOPT(3) = 2. (SHELL209 incorporates this extra node by default.)

SHELL208 allows you to account for large strain effects, transverse shear deformation, hyperelasticity and layers in your models. The element is intended to model finite strain with pure axisymmetric displacements; transverse shear strains are assumed to be small.

SHELL208 can be used for layered applications for modeling laminated composite shells or sandwich construction. See SHELL208 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 208.1: SHELL208 Geometry

SHELL208 Input Data

Figure 208.1: SHELL208 Geometry (p. 1091) shows the geometry, node locations, and element coordinate system for SHELL208. The element is defined by two nodes. For material property labels, the local x-direction corresponds to the meridional direction of the shell element. The local y-direction is the circumferential. The local z-direction corresponds to the through-the-thickness direction. Element formulation is based on logarithmic strain and true stress measures. Element kinematics allows for finite membrane strains (stretching). However, the curvature changes within an increment are assumed to be small.

The shell thickness and more general properties (such as material and number of integration points through the thickness) are specified via section commands (**SECTYPE**, **SECDATA** and **SECCONTROL**).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Shell section commands allow for both single-layered and composite shell definitions. You can designate the number of integration points (1, 3, 5, 7, or 9) located through the thickness of each layer. If only one, the integration point is always located midway between the top and the bottom surfaces. If three or more, two points are located on the top and the bottom surfaces respectively and the remaining points are distributed evenly between these two points. The default for each layer is three integration points. The element can have variable thickness, as a tabular function of global/local coordinates or node numbers (**SECFUNCTION**).

Element loads are described in Nodal Loading (p. 49). Pressure may be input as surface loads on the element faces as shown by the circled numbers on Figure 208.1: SHELL208 Geometry (p. 1091). Positive pressures act into the element.

Temperatures may be input as element body loads at the corners of the outside faces of the element and the corners of the interfaces between layers. The first corner temperature T1 defaults to TUNIF. If all other temperatures are unspecified, they default to T1. If KEYOPT(1) = 0 and exactly NL+1 (where NL is the number of layers in the shell section) temperatures are input, one temperature is used for the bottom corners of each layer, and the last temperature is for the top corners of the top layer. If KEYOPT(1) = 1 and if exactly NL temperatures are input, one temperature is used for the two corners of each layer; that is, T1 is used for T1 and T2; T2 (as input) is used for T3 and T4, etc. For any other input patterns, unspecified temperatures default to TUNIF.

Nodal forces, if any, should be input on a full 360° basis.

KEYOPT(1) is the membrane option. When KEYOPT(1) = 1, the element uses one integration point through-the-thickness and accounts for only membrane stiffness (that is, the bending and transverse shear stiffness are ignored).

KEYOPT(2) controls the torsion capability. When KEYOPT(2) = 1, the element allows constant torsion by allowing a translational degree of freedom UZ in the circumferential direction.

KEYOPT(3) is used to include or suppress internal nodes. When KEYOPT(3) = 2, the element contains an extra internal node and adopts a two-point integration rule. By default, the element uses one-point integration scheme (see Figure 208.1: SHELL208 Geometry (p. 1091)). Internal nodes are not accessible to users. Therefore, boundary conditions/loading can not be specified on those nodes.

SHELL208 includes the effects of transverse shear deformation. The transverse shear stiffness E11 can be specified using **SECCONTROL**. For a single-layered shell with isotropic material, default transverse shear stiffness is kGh, in which k = 5/6, G is the shear modulus, and h is the thickness of the shell.

SHELL208 can be associated with linear elastic, elastoplastic, creep, or hyperelastic material properties.

Set KEYOPT(8) = 2 to store midsurface results in the results file for single- or multi-layer shell elements. If you use **SHELL**,MID, you will see these calculated values, rather than the average of the TOP and BOTTOM results. You should use this option to access these correct midsurface results (membrane results) for those analyses where averaging TOP and BOTTOM results is inappropriate. Examples include midsurface stresses and strains with nonlinear material behavior, and midsurface results after mode combinations that involve squaring operations such as in spectrum analyses.

Set KEYOPT(9) = 1 to read initial thickness data from a user subroutine.

You can apply an initial stress state to this element via the **INISTATE** command. For more information, see Initial State in the *Basic Analysis Guide*.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

A summary of the element input is given in "SHELL208 Input Summary" (p. 1093). A general description of element input is given in Element Input (p. 43).

SHELL208 Input Summary

Nodes

I, J

Degrees of Freedom

UX, UY, ROTZ -- If KEYOPT(1) = 0 and KEYOPT(2) = 0
UX, UY -- If KEYOPT(1) = 1 and KEYOPT(2) = 0
UX, UY, UZ, ROTZ -- If KEYOPT(1) = 0 and KEYOPT(2) = 1
UX, UY, UZ -- If KEYOPT(1) = 1 and KEYOPT(2) = 1

Real Constants

None

Section Controls

E11, ADMSUA

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, EY, EZ, PRXY, PRYZ, PRXZ (*or* NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ),
DENS, GXY, GYZ, GXZ, ALPD, BETD

Surface Loads

Pressures --

face 1 (I-J) (top, in -N direction),
face 2 (I-J) (bottom, in +N direction)

Body Loads

Temperatures --

For KEYOPT(1) = 0:
T1, T2 (corresponding to nodes I and J) at bottom of layer 1, and T3, T4 (corresponding to nodes I and J) between layers 1-2. A similar relationship exists for all layers, ending with temperatures at the top of layer NL. Hence, for one-layer elements, four temperatures are used.
For KEYOPT(1) = 1:
T1, T2 for layer 1; T3, T4 for layer 2; similarly for all layers (2 * NL maximum). Hence, for one-layer elements, two temperatures are used.

Special Features

Birth and death
Element technology autoselect

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Initial state
Large deflection
Large strain
Linear perturbation
Nonlinear stabilization
Stress stiffening

KEYOPT(1)
Element stiffness:

0 --
Bending and membrane stiffness (default).

1 --
Membrane stiffness only.

KEYOPT(2)
Torsion capability:

0 --
Excluded (default).

1 --
Included.

KEYOPT(3)
Extra internal node option:

0 --
Suppress extra internal node (default).

2 --
Include extra internal node.

KEYOPT(8)
Storage of layer data:

0 --
Store data for BOTTOM of bottom layer and TOP of top layer (default).

1 --
Store data for TOP and BOTTOM for all layers.

2 --
Store data for TOP, BOTTOM, and MID for all layers. (The volume of data may be excessive.)

KEYOPT(9)
User-defined thickness:

0 --
No user subroutine to provide initial thickness (default).

1 --
Read initial thickness data from user subroutine UTHICK

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See the *Guide to User-Programmable Features* for information about user-written subroutines

SHELL208 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution.
- Additional element output as shown in Table 208.1: SHELL208 Element Output Definitions (p. 1096)

Several items are illustrated in Figure 208.2: SHELL208 Element Stress Output (p. 1096).

KEYOPT(8) controls the amount of data output on the result file for processing with the **LAYER** command. Interlaminar shear stress is available at the layer interfaces. Setting KEYOPT(8) = 1 or 2 is necessary for these stresses to be output in POST1. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The element stress resultants (N11, M11, Q13, etc.) are parallel to the element coordinate system, as are the membrane strains and curvatures of the element. Such generalized strains are available through the SMISC option at the element centroid only. The transverse shear force Q13 is available only in resultant form: that is, use SMISC,5. Likewise, the transverse shear strain γ_{13} is constant through the thickness and only available as a SMISC item (SMISC,10).

ANSYS computes moments (M11, M22) with respect to the shell reference plane. By default, ANSYS adopts the shell midplane as the reference plane. To offset the reference plane to any other specified location, issue the **SECOFFSET** command. When there is a nonzero offset (L) from the reference plane to the midplane, moments with respect to the midplane ($\overline{M11}, \overline{M22}$) can be recovered from stress resultants with respect to the reference plane as follows:

$$\overline{M11} = M11 - L \times N11$$

$$\overline{M22} = M22 - L \times N22$$

SHELL208 does not support extensive basic element printout. POST1 provides more comprehensive output processing tools; you should use the **OUTRES** command to ensure that the required results are stored in the database.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 208.2: SHELL208 Element Stress Output

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 208.1: SHELL208 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	-	Y
MAT	Material number	-	Y
THICK	Average thickness	-	Y
VOLU:	Volume	-	Y
XC, YC	Location where results are reported	Y	4
PRES	Pressures P1 (top) at NODES I, J; P2 (bottom) at NODES I, J	-	Y
TEMP	Temperatures T1, T2 at bottom of layer 1, T3, T4 between layers 1-2, similarly for between next layers, ending with temperatures at top of layer NL (2*(NL+1) maximum)	-	Y
LOC	TOP, MID, BOT, or integration point location	-	1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
S:X, Y, Z, XY, YZ, XZ	Stresses	3	1
S:1, 2, 3	Principal stresses	-	1
S:INT	Stress intensity	-	1
S:EQV	Equivalent stress	-	1
EPEL:X, Y, Z, XY,YZ,XZ	Elastic strains	3	1
EPEL:EQV	Equivalent elastic strain	-	1
EPTH:X, Y, Z, XY,YZ,XZ	Thermal strains	3	1
EPTH:EQV	Equivalent thermal strain	-	1
EPPL:X, Y, Z, XY,YZ,XZ	Average plastic strains	3	2
EPPL:EQV	Equivalent plastic strain	-	2
EPCR:X, Y, Z, XY ,YZ,XZ	Average creep strains	3	2
EPCR:EQV	Equivalent creep strain	-	2
EPTO:X, Y, Z ,XY,YZ,XZ	Total mechanical strains (EPEL+EPPL+EPCR)	3	-
EPTO:EQV	Total equivalent mechanical strains	-	-
NL:EPEQ	Accumulated equivalent plastic strain	-	2
NL:CREQ	Accumulated equivalent creep strain	-	2
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	-	2
NL:PLWK	Plastic work/volume	-	2
NL:HPRES	Hydrostatic pressure	-	2
SEND:Elastic, Plastic, Creep	Strain energy densities	-	2
N11, N22, N12	In-plane forces (per unit length)	-	Y
M11, M22	Out-of-plane moments (per unit length)	-	Y
Q13	Transverse shear forces (per unit length)	-	Y
E11, E22, E12	Membrane strains	-	Y
K11, K22	Curvatures	-	Y
γ_{13}	Transverse shear strain	-	Y
LOCI:X, Y, Z	Integration point locations	-	5
SVAR:1, 2, ... , N	State variables	-	6
ILSXZ	SXZ interlaminar shear stress	-	Y
ILSYZ	SYZ interlaminar shear stress	-	Y
ILSUM	Magnitude of the interlaminar shear stress vector	-	Y
ILANG	Angle of interlaminar shear stress vector (measured from the element x-axis toward the element y-axis in degrees)	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
Sm: 11, 22, 12	Membrane stresses	-	Y
Sb: 11, 22	Bending stresses	-	Y
Sp: 11, 22, 12	Peak stresses	-	Y
St: 13	Averaged transverse shear stresses	-	Y

1. The following stress solution repeats for top, middle, and bottom surfaces.
2. Nonlinear solution output for top, middle, and bottom surfaces, if the element has a nonlinear material, or if large-deflection effects are enabled (**NLGEOM**,ON) for SEND.
3. Stresses, total strains, plastic strains, elastic strains, creep strains, and thermal strains in the element coordinate system are available for output (at all section points through thickness). If layers are in use, the results are in the layer coordinate system.
4. Available only at centroid as a ***GET** item.
5. Available only if **OUTRES**,LOCI is used.
6. Available only if the `UserMat` subroutine and **TB**,STATE command are used.

Table 208.2: SHELL208 Item and Sequence Numbers (p. 1098) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) of this manual for more information. The following notation is used in Table 208.2: SHELL208 Item and Sequence Numbers (p. 1098):

Name
output quantity as defined in the Table 208.1: SHELL208 Element Output Definitions (p. 1096)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I, J
sequence number for data at nodes I, J.

Table 208.2: SHELL208 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
N11	SMISC	1	-	-
N22	SMISC	2	-	-
N12	SMISC	3	-	-
M11	SMISC	4	-	-
M22	SMISC	5	-	-
Q13	SMISC	6	-	-
ε_{11}	SMISC	7	-	-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
ε_{22}	SMISC	8	-	-
ε_{12}	SMISC	9	-	-
k_{11}	SMISC	10	-	-
k_{22}	SMISC	11	-	-
γ_{13}	SMISC	12	-	-
THICK	SMISC	13	-	-
P1	SMISC	-	14	15
P2	SMISC	-	16	17
Sm: 11	SMISC	18	-	-
Sm: 22	SMISC	19	-	-
Sm: 12	SMISC	20	-	-
Sb: 11	SMISC	21	-	-
Sb: 22	SMISC	22	-	-
Sp: 11 (at shell bottom)	SMISC	23	-	-
Sp: 22 (at shell bottom)	SMISC	24	-	-
Sp: 12 (at shell bottom)	SMISC	25	-	-
Sp: 11 (at shell top)	SMISC	26	-	-
Sp: 22 (at shell top)	SMISC	27	-	-
Sp: 12 (at shell top)	SMISC	28	-	-
St: 13	SMISC	29	-	-

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	Bottom of Layer i	Top of Layer NL
ILSXZ	SMISC	8 * (i - 1) + 31	8 * (NL - 1) + 32
ILSYZ	SMISC	8 * (i - 1) + 33	8 * (NL - 1) + 34
ILSUM	SMISC	8 * (i - 1) + 35	8 * (NL - 1) + 36
ILANG	SMISC	8 * (i - 1) + 37	8 * (NL - 1) + 38

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SHELL208 Assumptions and Restrictions

- The axisymmetric shell element must be defined in the global X-Y plane with the Y-axis the axis of symmetry.
- The element cannot have a zero length.
- Zero thickness elements, or elements tapering down to a zero thickness at any corner, are not allowed (but zero thickness *layers* are allowed).
- If multiple load steps are used, the number of layers may not change between load steps.
- No slippage is assumed between the element layers. Shear deflections are included in the element; however, normals to the center plane before deformation are assumed to remain straight after deformation.
- Transverse shear stiffness of the shell section is estimated by an energy equivalence procedure (of the generalized section forces & strains vs. the material point stresses and strains). The accuracy of this calculation may be adversely affected if the ratio of material stiffnesses (Young's moduli) between adjacent layers is very high.
- The calculation of interlaminar shear stresses is based on simplifying assumptions of unidirectional, uncoupled bending in each direction. If accurate edge interlaminar shear stresses are required, shell-to-solid submodeling should be used.
- The section definition permits use of hyperelastic material models and elastoplastic material models in laminate definition. However, the accuracy of the solution is primarily governed by fundamental assumptions of shell theory. The applicability of shell theory in such cases is best understood by using a comparable solid model.
- For nonlinear applications, this element works best with full Newton-Raphson solution scheme (**NROPT**,FULL,ON).
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). Prestress effects can be activated by the **PSTRES** command.
- In a nonlinear analysis, the solution process terminates if the thickness at any integration point that was defined with a nonzero thickness vanishes (within a small numerical tolerance).
- SHELL208 with an internal node cannot be used in substructures.

SHELL208 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The only special features allowed are stress stiffening and large deflections.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SHELL209

3-Node Axisymmetric Shell

MP ME ST PR PRN <> <> <> PP EME MFS
Product Restrictions

SHELL209 Element Description

The SHELL209 element is suitable for analyzing thin to moderately thick axisymmetric shell structures. It is a three-node element with three degrees of freedom at each node: translations in the X, Y directions, and a rotation about the Z-axis. A fourth translational degree of freedom in z direction can be included to model uniform torsion (KEYOPT(2) = 1). When the membrane option is used, the rotational degree of freedom is excluded. (For higher efficiency, the two-node element SHELL208 may be more suitable.)

The element is well suited for linear, large rotation, and/or large strain nonlinear applications. Changes in shell thickness and follower effects of distributed pressures are accounted for in nonlinear analyses, and it can be used for layered applications for modeling laminated composite shells or sandwich construction. See SHELL209 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 209.1: SHELL209 Geometry

SHELL209 Input Data

Figure 209.1: SHELL209 Geometry (p. 1101) shows the geometry, node locations, and element coordinate system for this element. The element is defined by three nodes. For material property labels, the local x-direction corresponds to the meridional direction of the shell element. The local y-direction is the circumferential. The local z-direction corresponds to the through-the-thickness direction. Element formulation is based on logarithmic strain and true stress measures. Element kinematics allows for finite membrane strains (stretching). However, the curvature changes within an increment are assumed to be small.

The shell thickness and more general properties (such as material and number of integration points through the thickness) are specified via section commands (**SECTYPE**, **SECDATA** and **SECCONTROL**). Shell section commands allow for both single-layered and composite shell definitions. You can designate the number of integration points (1, 3, 5, 7, or 9) located through the thickness of each layer. If only one, the integration point is always located midway between the top and the bottom surfaces. If three

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

or more, two points are located on the top and the bottom surfaces respectively and the remaining points are distributed evenly between these two points. The default for each layer is three integration points. The element can have variable thickness, as a tabular function of global/local coordinates or node numbers (**SECFUNCTION**).

Element loads are described in Nodal Loading (p. 49). Pressure may be input as surface loads on the element faces as shown by the circled numbers on Figure 209.1: SHELL209 Geometry (p. 1101). Positive pressures act into the element.

Temperatures can be input as element body loads at the corners of the outside faces of the element and the corners of the interfaces between layers. The first corner temperature T1 defaults to TUNIF. If all other temperatures are unspecified, they default to T1. If KEYOPT(1) = 0 and exactly NL+1 (where NL is the number of layers in the shell section) temperatures are input, one temperature is used for the bottom corners of each layer, and the last temperature is for the top corners of the top layer. If KEYOPT(1) = 1 and if exactly NL temperatures are input, one temperature is used for the two corners of each layer. That is, T1 is used for T1, T2, and T3; T2 (as input) is used for T4, T5, and T6, etc. For any other input patterns, unspecified temperatures default to TUNIF.

Nodal forces, if any, should be input on a full 360° basis.

KEYOPT(1) is the membrane option. When KEYOPT(1) = 1, the element uses one integration point through-the-thickness and accounts for only membrane stiffness (that is, the bending and transverse shear stiffness are ignored).

KEYOPT(2) controls the torsion capability. When KEYOPT(2) = 1, the element allows constant torsion by allowing a translational degree of freedom UZ in the circumferential direction.

SHELL209 includes the effects of transverse shear deformation. The transverse shear stiffness E11 can be specified with **SECCONTROL**. For a single-layered shell with isotropic material, default transverse shear stiffness is kGh, in which k = 5/6, G is the shear modulus, and h is the thickness of the shell.

SHELL209 can be associated with linear elastic, elastoplastic, creep, or hyperelastic material properties.

Set KEYOPT(8) = 2 to store midsurface results in the results file for single or multi-layer shell elements. If you use **SHELL**,MID, you will see these calculated values, rather than the average of the TOP and BOTTOM results. You should use this option to access these correct midsurface results (membrane results) for those analyses where averaging TOP and BOTTOM results is inappropriate. Examples include midsurface stresses and strains with nonlinear material behavior, and midsurface results after mode combinations that involve squaring operations such as in spectrum analyses.

Set KEYOPT(9) = 1 to read initial thickness data from a user subroutine.

You can apply an initial stress state to this element via the **INISTATE** command. For more information, see Initial State in the *Basic Analysis Guide*.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

"SHELL209 Input Summary" (p. 1103) gives a summary of the element input. A general description of element input is given in Element Input (p. 43)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SHELL209 Input Summary

Nodes

I, J, K

Degrees of Freedom

UX, UY, ROTZ -- If KEYOPT(1) = 0 and KEYOPT(2) = 0
UX, UY -- If KEYOPT(1) = 1 and KEYOPT(2) = 0
UX, UY, UZ, ROTZ -- If KEYOPT(1) = 0 and KEYOPT(2) = 1
UX, UY, UZ -- If KEYOPT(1) = 1 and KEYOPT(2) = 1

Real Constants

None

Section Controls

E11, ADMSUA

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, EY, EZ, PRXY, PRYZ, PRXZ (*or* NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ),
DENS, GXY, GYZ, GXZ, ALPD, BETD

Surface Loads

Pressures --

face 1 (I-J-K) (top, in -N direction),
face 2 (I-J-K) (bottom, in +N direction)

Body Loads

Temperatures --

For KEYOPT(1) = 0:
T1, T2 T3 (corresponding to nodes I, J, and K) at bottom of layer 1, and T4, T5, T6 (corresponding to nodes I, J, and K) between layers 1-2. A similar relationship exists for all layers, ending with temperatures at the top of layer NL. For one-layer elements, therefore, six temperatures are used.
For KEYOPT(1) = 1:
T1, T2, T3 for layer 1; T4, T5, T6 for layer 2; similarly for all layers (3 * NL maximum). Hence, for one-layer elements, three temperatures are used.

Special Features

Birth and death
Element technology autoselect
Initial state
Large deflection
Large strain
Linear perturbation
Nonlinear stabilization
Stress stiffening

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(1)
Element stiffness:

0 --
Bending and membrane stiffness (default).

1 --
Membrane stiffness only.

KEYOPT(2)
Torsion capability:

0 --
Excluded (default).

1 --
Included.

KEYOPT(8)
Storage of layer data:

0 --
Store data for BOTTOM of bottom layer and TOP of top layer (default).

1 --
Store data for TOP and BOTTOM for all layers.

2 --
Store data for TOP, BOTTOM, and MID for all layers. (The volume of data may be excessive.)

KEYOPT(9)
User-defined thickness:

0 --
No user subroutine to provide initial thickness (default).

1 --
Read initial thickness data from user subroutine UTHICK

See the *Guide to User-Programmable Features* for information about user-written subroutines.

SHELL209 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution.
- Additional element output as shown in Table 209.1: SHELL209 Element Output Definitions (p. 1106)

Several items are illustrated in Figure 209.2: SHELL209 Element Stress Output (p. 1105).

KEYOPT(8) controls the amount of data output on the result file for processing with the **LAYER** command. Interlaminar shear stress is available at the layer interfaces. Setting KEYOPT(8) = 1 or 2 is necessary for these stresses to be output in POST1. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The element stress resultants (N11, M11, Q13, etc.) are parallel to the element coordinate system, as are the membrane strains and curvatures of the element. Such generalized strains are available through the SMISC option at the element centroid only. The transverse shear force Q13 are available only in resultant form: that is, use SMISC,5. Likewise, the transverse shear strain.

γ_{13} is constant through the thickness and only available as a SMISC item (SMISC,10).

ANSYS computes moments (M11, M22) with respect to the shell reference plane. By default, ANSYS adopts the shell midplane as the reference plane. To offset the reference plane to any other specified location, issue the **SECOFFSET** command. When there is a nonzero offset (L) from the reference plane to the midplane, moments with respect to the midplane ($\overline{M11}, \overline{M22}$) can be recovered from stress resultants with respect to the reference plane as follows:

$$\overline{M11} = M11 - L \times N11$$

$$\overline{M22} = M22 - L \times N22$$

SHELL209 does not support extensive basic element printout. POST1 provides more comprehensive output processing tools; therefore, we suggest using **OUTRES** command to ensure that the required results are stored in the database.

Figure 209.2: SHELL209 Element Stress Output

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 209.1: SHELL209 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K	-	Y
MAT	Material number	-	Y
THICK	Average thickness	-	Y
VOLU:	Volume	-	Y
XC, YC	Location where results are reported	Y	4
PRES	Pressures P1 (top) at NODES I, J; P2 (bottom) at NODES I, J	-	Y
TEMP	Temperatures T1, T2 at bottom of layer 1, T3, T4 between layers 1-2, similarly for between next layers, ending with temperatures at top of layer NL (2*(NL+1) maximum)	-	Y
LOC	TOP, MID, BOT, or integration point location	-	1
S:X, Y, Z, XY, YZ, XZ	Stresses	3	1
S:1, 2, 3	Principal stresses	-	1
S:INT	Stress intensity	-	1
S:EQV	Equivalent stress	-	1
EPEL:X, Y, Z, XY,YZ,XZ	Elastic strains	3	1
EPEL:EQV	Equivalent elastic strain	-	1
EPTH:X, Y, Z, XY,YZ,XZ	Thermal strains	3	1
EPTH:EQV	Equivalent thermal strain	-	1
EPPL:X, Y, Z, XY,YZ,XZ	Average plastic strains	3	2
EPPL:EQV	Equivalent plastic strain	-	2
EPCR:X, Y, Z, XY,YZ,XZ	Average creep strains	3	2
EPCR:EQV	Equivalent creep strain	-	2
EPTO:X, Y, Z ,XY,YZ,XZ	Total mechanical strains (EPEL+EPPL+EPCR)	3	-
EPTO:EQV	Total equivalent mechanical strains	-	-
NL:EPEQ	Accumulated equivalent plastic strain	-	2
NL:CREQ	Accumulated equivalent creep strain	-	2
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	-	2
NL:PLWK	Plastic work/volume	-	2

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
NL:HPRES	Hydrostatic pressure	-	2
SEND:Elastic, Plastic, Creep	Strain energy densities	-	2
N11, N22, N12	In-plane forces (per unit length)	-	Y
M11, M22	Out-of-plane moments (per unit length)	-	Y
Q13	Transverse shear forces (per unit length)	-	Y
E11, E22, E12	Membrane strains	-	Y
K11, K22	Curvatures	-	Y
γ_{13}	Transverse shear strain	-	Y
LOCI:X, Y, Z	Integration point locations	-	5
SVAR:1, 2, ... , N	State variables	-	6
ILSXZ	SXZ interlaminar shear stress	-	Y
ILSYZ	SYZ interlaminar shear stress	-	Y
ILSUM	Magnitude of the interlaminar shear stress vector	-	Y
ILANG	Angle of interlaminar shear stress vector (measured from the element x-axis toward the element y-axis in degrees)	-	Y
Sm: 11, 22, 12	Membrane stresses	-	Y
Sb: 11, 22	Bending stresses	-	Y
Sp: 11, 22, 12	Peak stresses	-	Y
St: 13	Averaged transverse shear stresses	-	Y

1. The following stress solution repeats for top, middle, and bottom surfaces.

2. Nonlinear solution output for top, middle, and bottom surfaces, if the element has a nonlinear material, or if large-deflection effects are enabled (**NLGEOM**,ON) for SEND.

3. Stresses, total strains, plastic strains, elastic strains, creep strains, and thermal strains in the element co-ordinate system are available for output (at all section points through thickness). If layers are in use, the results are in the layer coordinate system.

4. Available only at centroid as a ***GET** item.

5. Available only if **OUTRES**,LOCI is used.

6. Available only if the `UserMat` subroutine and **TB**,STATE command are used.

Table 209.2: SHELL209 Item and Sequence Numbers (p. 1108) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) of this manual for more information. The following notation is used in Table 209.2: SHELL209 Item and Sequence Numbers (p. 1108):

Name

output quantity as defined in the Table 208.1: SHELL208 Element Output Definitions (p. 1096)

Item

predetermined Item label for **ETABLE** command

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

E

sequence number for single-valued or constant element data

I, J, K

sequence number for data at nodes I, J, K.

Table 209.2: SHELL209 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	E	I	J	K
N11	SMISC	1	-	-	-
N22	SMISC	2	-	-	-
N12	SMISC	3	-	-	-
M11	SMISC	4	-	-	-
M22	SMISC	5	-	-	-
Q13	SMISC	6	-	-	-
ε_{11}	SMISC	7	-	-	-
ε_{22}	SMISC	8	-	-	-
ε_{12}	SMISC	9	-	-	-
k_{11}	SMISC	10	-	-	-
k_{22}	SMISC	11	-	-	-
γ_{13}	SMISC	12	-	-	-
THICK	SMISC	13	-	-	-
P1	SMISC	-	14	15	16
P2	SMISC	-	17	18	19
Sm: 11	SMISC	18	-	-	-
Sm: 22	SMISC	19	-	-	-
Sm: 12	SMISC	20	-	-	-
Sb: 11	SMISC	21	-	-	-
Sb: 22	SMISC	22	-	-	-
Sp: 11 (at shell bottom)	SMISC	23	-	-	-
Sp: 22 (at shell bottom)	SMISC	24	-	-	-
Sp: 12 (at shell bottom)	SMISC	25	-	-	-
Sp: 11 (at shell top)	SMISC	26	-	-	-
Sp: 22 (at shell top)	SMISC	27	-	-	-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input				
	Item	E	I	J	K
Sp: 12 (at shell top)	SMISC	28	-	-	-
St: 13	SMISC	29	-	-	-

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	Bottom of Layer i	Top of Layer NL
ILSXZ	SMISC	8 * (i - 1) + 31	8 * (NL - 1) + 32
ILSYZ	SMISC	8 * (i - 1) + 33	8 * (NL - 1) + 34
ILSUM	SMISC	8 * (i - 1) + 35	8 * (NL - 1) + 36
ILANG	SMISC	8 * (i - 1) + 37	8 * (NL - 1) + 38

SHELL209 Assumptions and Restrictions

- The axisymmetric shell element must be defined in the global X-Y plane with the Y-axis the axis of symmetry.
- The element cannot have a zero length.
- Zero thickness elements, or elements tapering down to a zero thickness at any corner, are not allowed (but zero thickness *layers* are allowed).
- If multiple load steps are used, the number of layers may not change between load steps.
- No slippage is assumed between the element layers. Shear deflections are included in the element; however, normals to the center plane before deformation are assumed to remain straight after deformation.
- Transverse shear stiffness of the shell section is estimated by an energy equivalence procedure (of the generalized section forces & strains vs. the material point stresses and strains). The accuracy of this calculation may be adversely affected if the ratio of material stiffnesses (Young's moduli) between adjacent layers is very high.
- The calculation of interlaminar shear stresses is based on simplifying assumptions of unidirectional, uncoupled bending in each direction. If accurate edge interlaminar shear stresses are required, shell-to-solid submodeling should be used.
- The section definition permits use of hyperelastic material models and elastoplastic material models in laminate definition. However, the accuracy of the solution is primarily governed by fundamental assumptions of shell theory. The applicability of shell theory in such cases is best understood by using a comparable solid model.
- For nonlinear applications, this element works best with full Newton-Raphson solution scheme (**NROPT**,FULL,ON).
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). Prestress effects can be activated by the **PSTRES** command.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- In a nonlinear analysis, the solution process terminates if the thickness at any integration point that was defined with a nonzero thickness vanishes (within a small numerical tolerance).

SHELL209 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The only special features allowed are stress stiffening and large deflections.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CPT212

2-D 4-Node Coupled Pore-Pressure Mechanical Solid

MP ME <> <> <> <> <> <> PP EME MFS
Product Restrictions

CPT212 Element Description

CPT212 is a 2-D four-node coupled pore-pressure mechanical solid element. The element has bilinear displacement behavior.

The element is defined by four nodes having three degrees of freedom at each node: translations in the nodal x and y directions, and one pore-pressure degree of freedom at each node.

CPT212 can be used as a plane strain or axisymmetric element. The element has stress stiffening, large deflection, and large strain capabilities. Various printout options are also available.

See CPT212 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 212.1: CPT212 Geometry

CPT212 Input Data

The geometry and node locations for this element are shown in Figure 212.1: CPT212 Geometry (p. 1111).

A degenerated triangular-shaped element can be formed by defining the same node number for nodes K and L. In addition to the nodes, the element input data includes the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. (The element coordinate system orientation is described in Coordinate Systems (p. 55).)

Element loads are described in Nodal Loading (p. 49). Pressures can be input as surface loads on the element faces as shown by the circled numbers in Figure 212.1: CPT212 Geometry (p. 1111). Positive pressures act into the element. Temperatures can be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input temperature pattern, unspecified temperatures default to TUNIF.

Input the nodal forces, if any, per unit of depth for a plane analysis and on a full 360° basis for an axisymmetric analysis.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

As described in Coordinate Systems (p. 55), you can use the **ESYS** command to orient the material properties and strain/stress output. Use the **RSYS** command to choose output that follows the material coordinate system or the global coordinate system.

The effects of pressure load stiffness are automatically included for this element, and the element generally produces an unsymmetric matrix. To avoid convergence difficulty, issue the **NROPT**,UNSYM command to use the unsymmetric solver.

The following table summarizes the element input. For a general description of element input, see Element Input (p. 43).

CPT212 Input Summary

Nodes
I, J, K, L

Degrees of Freedom
UX, UY, PRES

Real Constants

None

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ),
DENS, GXY, GYZ, GXZ, ALPD, BETD

Surface Loads

Pressures --
face 1 (J-I), face 2 (K-J), face 3 (I-K), face 4 (I-L)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L)

Special Features --

Initial state
Large deflection
Large strain
Stress stiffening

KEYOPT(3)
Element behavior:

1 --
Axisymmetric

2 --
Plane strain (Z strain = 0.0) (default)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CPT212 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements and pore pressure included in the overall nodal solution
- Additional element output as shown in Table 212.1: CPT212 Element Output Definitions (p. 1113)

The element stress directions are parallel to the element coordinate system, as shown in Figure 212.2: CPT212 Stress Output (p. 1113). A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

Figure 212.2: CPT212 Stress Output

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 212.1: CPT212 Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Nodes - I, J, K, L	-	Y
MAT	Material number	-	Y
THICK	Thickness	-	Y
VOLU	Volume	-	Y
XC, YC	Location where results are reported	Y	2
TEMP	Temperatures T(I), T(J), T(K), T(L)	-	`Y
S:X, Y, Z, XY	Stresses	Y	Y
S:1, 2, 3	Principal stresses	-	Y
S:INT	Stress intensity	-	Y
S:EQV	Equivalent stress	Y	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
EPEL:X, Y, Z, XY	Elastic strains	Y	Y
EPEL:1, 2, 3	Principal elastic strains	-	Y
EPEL:EQV	Equivalent elastic strain	Y	Y
EPTH:X, Y, Z, XY	Thermal strains	1	1
ESIG:X, Y, Z, XY	Effective stresses	-	Y

1. Output only if element has a thermal load.

2. Available only at centroid as a ***GET** item.

For axisymmetric solutions in a global coordinate system, the X, Y, XY, and Z stress and strain outputs correspond to the radial, axial, in-plane shear, and hoop stresses and strains, respectively.

Table 212.2: CPT212 Item and Sequence Numbers (p. 1114) lists output available via the **ETABLE** command using the Sequence Number method. For more information, see Creating an Element Table in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this document. The table uses the following notation:

Name
output quantity as defined in the Table 212.1: CPT212 Element Output Definitions (p. 1113)

Item
predetermined Item label for **ETABLE**

E
sequence number for single-valued or constant element data

I,J,K,L
sequence number for data at nodes I, J, K, L

Table 212.2: CPT212 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input									
	Item	E	I	J	K	L	M	N	O	P
P1	SMISC	-	2	1	-	-	-	-	-	-
P2	SMISC	-	-	4	3	-	-	-	-	-
P3	SMISC	-	-	-	6	5	-	-	-	-
P4	SMISC	-	7	-	-	8	-	-	-	-

CPT212 Assumptions and Restrictions

- The area of the element must be positive.

- The element must lie in a global X-Y plane as shown in Figure 212.1: CPT212 Geometry (p. 1111) and the Y-axis must be the axis of symmetry for axisymmetric analyses. An axisymmetric structure should be modeled in the +X quadrants.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You can form a triangular element by defining duplicate K and L node numbers. (For more information, see Degenerated Shape Elements (p. 39).)
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). It is ignored in geometrically linear analyses (**NLGEOM**,OFF). Prestress effects can be activated by the **PSTRES** command.

CPT212 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

CPT213

2-D 8-Node Coupled Pore-Pressure Mechanical Solid

MP ME <> <> <> <> <> <> PP EME MFS
Product Restrictions

CPT213 Element Description

CPT213 is a higher-order 2-D eight-node coupled pore-pressure mechanical solid element. The element has quadratic displacement behavior and is well suited to modeling curved boundaries.

The element is defined by eight nodes having three degrees of freedom at each corner node:

- The translations in the nodal x and y directions
- One pore-pressure degree of freedom and two degrees of freedom at the midside nodes: the translations in the nodal x and y directions.

CPT213 can be used as a plane strain or axisymmetric element. The element has stress stiffening, large deflection, and large strain capabilities. Various printout options are also available. See CPT213 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 213.1: CPT213 Geometry

A higher-order version of this element is CPT217.

CPT213 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 213.1: CPT213 Geometry (p. 1117).

A degenerated triangular-shaped element can be formed by defining the same node number for nodes K, L and O. In addition to the nodes, the element input data includes the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. (The element coordinate system orientation is described in Coordinate Systems (p. 55).)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Element loads are described in Nodal Loading (p. 49). Pressures can be input as surface loads on the element faces as shown by the circled numbers in Figure 213.1: CPT213 Geometry (p. 1117). Positive pressures act into the element. Temperatures can be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). If all corner node temperatures are specified, each midside node temperature defaults to the average temperature of its adjacent corner nodes. For any other input temperature pattern, unspecified temperatures default to TUNIF.

The nodal forces, if any, should be input per unit of depth for a plane analysis and on a full 360° basis for an axisymmetric analysis.

As described in Coordinate Systems (p. 55), you can use the **ESYS** command to orient the material properties and strain/stress output. Use **ESYS** to choose output that follows the material coordinate system or the global coordinate system.

The effects of pressure load stiffness are automatically included for this element, and the element generally produces an unsymmetric matrix. To avoid convergence difficulty, issue the **NROPT**,UNSYM command to use the unsymmetric solver.

The following table summarizes the element input. Element Input (p. 43) gives a general description of element input.

CPT213 Input Summary

Nodes

I, J, K, L, M, N, O, P

Degrees of Freedom

UX, UY, PRES

Real Constants

None

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ),
DENS, GXY, GYZ, GXZ, ALPD, BETD

Surface Loads

Pressures --

face 1 (J-I), face 2 (K-J), face 3 (I-K), face 4 (I-L)

Body Loads

Temperatures --

T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

Special Features

Initial state
Large deflection

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Large strain
Stress stiffening

KEYOPT(3)
Element behavior:

1 --
Axisymmetric

2 --
Plane strain (Z strain = 0.0) (default)

CPT213 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements and pore pressure included in the overall nodal solution
- Additional element output as shown in Table 213.1: CPT213 Element Output Definitions (p. 1119).

As illustrated in Figure 213.2: CPT213 Stress Output (p. 1119), the element stress directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

Figure 213.2: CPT213 Stress Output

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 213.1: CPT213 Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Nodes - I, J, K, L	-	Y
MAT	Material number	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
THICK	Thickness	-	Y
VOLU	Volume	-	Y
XC, YC	Location where results are reported	Y	1
TEMP	Temperatures T(I), T(J), T(K), T(L)	-	Y
S:X, Y, Z, XY	Stresses	Y	Y
S:1, 2, 3	Principal stresses	-	Y
S: INT	Stress intensity	-	Y
S:EQV	Equivalent stress	-	Y
EPEL:X, Y, Z, XY	Elastic strains	Y	Y
EPEL:1, 2, 3	Principal elastic strains	Y	-
EPEL:EQV	Equivalent elastic strain [2]	-	Y
EPTH:X, Y, Z, XY	Thermal strains	3	3
EPTH:EQV	Equivalent thermal strain [2]	-	3
ESIG:X, Y, Z, XY	Effective stresses	-	Y

1. Available only at centroid as a ***GET** item.

2. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.

3. Output only if element has a thermal load.

For axisymmetric solutions, the X, Y, XY, and Z stress and strain outputs correspond to the radial, axial, in-plane shear, and hoop stresses and strains.

Table 213.2: CPT213 Item and Sequence Numbers (p. 1121) lists output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this document for more information. The following notation is used in Table 213.2: CPT213 Item and Sequence Numbers (p. 1121):

Name
output quantity as defined in Table 213.1: CPT213 Element Output Definitions (p. 1119)

Item
predetermined Item label for **ETABLE**

E
sequence number for single-valued or constant element data

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

I,J,...,P
sequence number for data at nodes I, J, ..., P

Table 213.2: CPT213 Item and Sequence Numbers

Output Quant-ity Name	ETABLE and ESOL Command Input									
	Item	E	I	J	K	L	M	N	O	P
P1	SMISC	-	2	1	-	-	-	-	-	-
P2	SMISC	-	-	4	3	-	-	-	-	-
P3	SMISC	-	-	-	6	5	-	-	-	-
P4	SMISC	-	7	-	-	8	-	-	-	-

CPT213 Assumptions and Restrictions

- The area of the element must be positive.
- The element must lie in a global X-Y plane as shown in Figure 213.1: CPT213 Geometry (p. 1117) and the Y-axis must be the axis of symmetry for axisymmetric analyses. An axisymmetric structure should be modeled in the +X quadrants.
- An edge with a removed midside node implies that the displacement varies linearly, rather than parabolically, along that face. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.
- A triangular element can be formed by defining duplicate K-L-O node numbers. (See Degenerated Shape Elements (p. 39).) For these degenerated elements, the triangular shape function is used and the solution is the same as for the regular triangular 6-node elements.
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). It is ignored in geometrically linear analyses (**NLGEOM**,OFF). Prestress effects can be activated via the **PSTRES** command.

CPT213 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

COMBI214

2-D Spring-Damper Bearing

MP ME ST PR PRN DS <> <> PP EME MFS
Product Restrictions

COMBI214 Element Description

COMBI214 has longitudinal as well as cross-coupling capability in 2-D applications. It is a tension-compression element with up to two degrees of freedom at each node: translations in any *two* nodal directions (x, y, or z). COMBI214 has two nodes plus one optional orientation node. No bending or torsion is considered.

The spring-damper element has no mass. Masses can be added by using the appropriate mass element (MASS21). The spring or the damping capability may be removed from the element.

A longitudinal spring/damper with torsion capabilities is available via the COMBIN14 element. A general spring or damper is also available in the stiffness, damping or mass matrix element MATRIX27. Another spring-damper element having its direction of action determined by the nodal coordinate directions is COMBIN40.

For more information about this element, see COMBI214 - 2-D Spring-Damper Bearing in the *Mechanical APDL Theory Reference*.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 214.1: COMBI214 Geometry

For linear analyses, I and J can be coincident.

COMBI214 Input Data

The geometry, node locations, and coordinate system for this element are shown in Figure 214.1: COMBI214 Geometry (p. 1124). The element is defined by two nodes. It has stiffness characteristics K_{11}, K_{22}, K_{12} and K_{21} and damping characteristics C_{11}, C_{22}, C_{12} and C_{21}. The stiffness coefficients should have units of Force/Length, and the damping coefficient units are Force*Time/Length. (The damping capability is not used for static or undamped modal analyses.)

The third node is for orientation and applies to nonlinear analyses only.

For stiffness and damping real constants, either numerical values or tabular array inputs can be specified. If specifying tabular inputs, enclose the table name within "%" characters (%`tabname`%).

These real constants can vary with the amplitude of the rotational velocity vector (defined via the **OMEGA** or **CMOMEGA** command). Use the ***DIM** command and the primary variable OMEGS to dimension the table and identify the variable. Because the amplitude of the rotational velocity vector is an absolute value, only positive values of OMEGS in the table parameter are valid.

These real constants can also vary with the eccentricity (the position of the center line of the rotor) and/or the phase shift between the displacements in the two nodal directions. Use the ***DIM** command and the primary variable ECCENT and/or THETA to dimension the table and identify the variables. The phase shift values unit (THETA) is the degree. ECCENT and THETA only apply to nonlinear analyses.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For more information about using tabular inputs, see Array Parameters in the *ANSYS Parametric Design Language Guide*, Applying Loads Using TABLE Type Array Parameters in the *Basic Analysis Guide*, and Performing a Thermal Analysis Using Tabular Boundary Conditions in the *Thermal Analysis Guide*.

KEYOPT(2) = 0 through 2 options define the element plane. The element operates in the nodal coordinate system.

The KEYOPT(3) = 0 and 1 options specify whether or not the element is symmetric. When symmetric, cross-coupling terms in stiffness and damping coefficients are equal (that is, $K_{12} = K_{21}$ and $C_{12} = C_{21}$).

A summary of the element input is given in "COMBI214 Input Summary" (p. 1125). A general description of element input is given in Element Input (p. 43).

COMBI214 Input Summary

Nodes

I, J, K (The K orientation node is optional and for nonlinear analyses only.)

Degrees of Freedom

UX, UY (KEYOPT (2) = 0)
UY, UZ (KEYOPT (2) = 1)
UX, UZ (KEYOPT (2) = 2)

Real Constants

K_{11}, K_{22}, K_{12}, K_{21} , C_{11}, C_{22}, C_{12}, C_{21}
K_{ij} - (i=1,2 j=1,2) Stiffness coefficients
C_{ij} - (i=1,2 j=1,2) Damping coefficients

Note

Real constants may be defined as table parameters as a function of the rotational velocity (using primary variable OMEGS), the eccentricity (using primary variable ECCENT), and/or the phase shift (using primary variable THETA). If the rotational velocity is the only parameter, the characteristics can be imported directly from an ASCII text file using the APDL macro `importbearing1.mac`. See Selecting Parts and Bearings for more details.

Material Properties

None

Surface Loads

None

Body Loads

None

Special Features

Birth and death
Large deflection
Linear perturbation
Stress stiffening

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(2)

Degrees of freedom selection:

0 --

Element lies in a plane parallel to the XY plane. The degrees of freedom are UX and UY. This value option is the default.

1 --

Element lies in a plane parallel to the YZ plane. The degrees of freedom are UY and UZ.

2 --

Element lies in a plane parallel to the XZ plane. The degrees of freedom are UX and UZ.

KEYOPT(3)

Symmetry:

0 --

Element is symmetric: $K_{12} = K_{21}$ and $C_{12} = C_{21}$. This option is the default.

1 --

Element is not symmetric.

KEYOPT(4)

Element stiffness and damping matrices output:

0 --

Do not print element matrices.

1 --

Print element matrices at beginning of solution phase.

COMBI214 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 214.1: COMBI214 Element Output Definitions (p. 1127).

The Element Output Definitions table uses the following notation:

(1) and (2) indicate the first and second axis of the element plane defined by KEYOPT(2). For example, if KEYOPT(2) equals 0, then (1) is the X axis and (2) is the Y axis.

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 214.1: COMBI214 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J	Y	Y
XC, YC, ZC	Location where results are reported	Y	1
FORC1	Spring force along (1)	Y	Y
FORC2	Spring force along (2)	Y	Y
STRETCH1	Stretch of spring along (1)	Y	Y
STRETCH2	Stretch of spring along (2)	Y	Y
VELOCITY1	Velocity along (1)	-	Y
VELOCITY2	Velocity along (2)	-	Y
DAMPING FORCE1	Damping force along (1) -- Zero unless this is a transient analysis (**ANTYPE**,TRANS) *and* damping is present	Y	Y
DAMPING FORCE2	Damping force along (2) -- Zero unless this is a transient analysis (**ANTYPE**,TRANS) *and* damping is present	Y	Y

1. Available only at centroid as a ***GET** item.

Table 214.2: COMBI214 Item and Sequence Numbers (p. 1127) lists output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this document for more information. The following notation is used in Table 214.2: COMBI214 Item and Sequence Numbers (p. 1127):

Name
Output quantity as defined in Table 214.1: COMBI214 Element Output Definitions (p. 1127)

Item
Predetermined Item label for the **ETABLE** command

E
Sequence number for single-valued or constant element data

Output data for COMBI214 consists of the following:

Table 214.2: COMBI214 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
FORC1	SMISC	1
FORC2	SMISC	2
STRETCH1	NMISC	1
STRETCH2	NMISC	2

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
VELOCITY1	NMISC	3
VELOCITY2	NMISC	4
DAMPING FORCE1	NMISC	5
DAMPING FORCE2	NMISC	6

COMBI214 Assumptions and Restrictions

- Nodes must lie in the plane defined by KEYOPT(2).
- The following applies to a nonlinear analysis:

  -- The orientation node K is required.

  -- The length of the spring-damper element must not be zero (that is, nodes I, J and K should not be coincident because the node locations determine the spring orientation).
- Vector (IJ) must be along axis (1). Vector (KJ) must be along axis (2). Directions are considered.
- The element allows only a uniform stress in the springs.
- K_{11} and K_{22} must be non zero to activate the spring capability.
- C_{11} and C_{22} must be non zero to activate the damper capability.
- The following applies when KEYOPT(3) = 0 (symmetric):

  -- If K_{12} is non-zero and K_{21} is zero, then K_{21} is set to K_{12}.

  -- If C_{12} is non-zero and C_{21} is zero, then C_{21} is set to C_{12}.
- The spring or the damping capability may be deleted from the element by setting all K_{ij} (i=1,2 j=1,2) or all C_{ij} (i=1,2 j=1,2) equal to zero, respectively.
- The degrees of freedom are specified in the nodal coordinate system and are the same for both nodes. (For more information, see Elements That Operate in the Nodal Coordinate System (p. 57).) If the nodal coordinate systems are rotated relative to each other, the same degree of freedom may be in different directions (thereby giving possibly unexpected results).
- No moment effects are included; that is, if the nodes are offset from the lines of action, moment equilibrium may not be satisfied.
- The element is defined such that a positive displacement of node J relative to node I tends to stretch the spring. If, for a given set of conditions, nodes I and J are interchanged, a positive displacement of node J relative to node I tends to compress the spring.
- Rotating damping effect (RotDamp = ON in the **CORIOLIS** command) is taken into account if the diagonal damping characteristics (C11 and C22) are equal and non zero and cross-terms (C12 and C21) are zero.
- Rotating damping effect is ignored if the element has a real constant input as a table array.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

COMBI214 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

Structural Analysis:

- No damping capability; C_{ij} (i=1,2 j=1,2) are not allowed.
- Only stress stiffening and large deflections are allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

CPT215

3-D 8-Node Coupled Pore-Pressure Mechanical Solid

MP ME <> <> <> <> <> <> PP EME MFS
Product Restrictions

CPT215 Element Description

CPT215 is a 3-D eight-node coupled pore-pressure mechanical solid element. The element is defined by eight nodes having four degrees of freedom at each node: translations in the nodal x, y, and z directions, and one pore-pressure degree of freedom at each corner node.

CPT215 has elasticity, stress stiffening, large deflection, and large strain capabilities. Various printout options are available.

For more details about this element, see CPT215 in the *Mechanical APDL Theory Reference*.

Figure 215.1: CPT215 Structural Solid Geometry

A higher-order version of this element is CPT216.

CPT215 Input Data

The geometry and node locations for this element are shown in Figure 215.1: CPT215 Structural Solid Geometry (p. 1131). A prism-shaped element can be formed by defining the same node numbers for nodes K and L, and nodes O and P. A tetrahedral-shaped element and a pyramid-shaped element can also be formed, as shown in the illustration. (CPT217 is a similar element, but is a 10-node tetrahedron.)

In addition to the nodes, the element input data includes the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is described in Coordinate Systems (p. 55).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Element loads are described in Nodal Loading (p. 49). Pressures can be input as surface loads on the element faces as shown by the circled numbers in Figure 215.1: CPT215 Structural Solid Geometry (p. 1131). Positive pressures act into the element. Temperatures can be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input temperature pattern, unspecified temperatures default to TUNIF.

As described in Coordinate Systems (p. 55), you can use the **ESYS** command to orient the material properties and strain/stress output. Use the **RSYS** command to choose output that follows the material coordinate system or the global coordinate system.

The effects of pressure load stiffness are automatically included for this element, and the element generally produces an unsymmetric matrix. To avoid convergence difficulty, issue the **NROPT**,UNSYM command to use the unsymmetric solver.

"CPT215 Input Summary" (p. 1132) contains a summary of element input. For a general description of element input, see Element Input (p. 43).

CPT215 Input Summary

Nodes

I, J, K, L, M, N, O, P

Degrees of Freedom

UX, UY, UZ, PRES

Real Constants

None

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, EY, EZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ or THSX,THSY, THSZ), PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), DENS, GXY, GYZ, GXZ, ALPD, BETD

Surface Loads

Pressures --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperatures --

T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

Special Features

Initial state
Large deflection
Large strain
Stress stiffening

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CPT215 Technology

CPT215 uses the $\bar{B}$ method (also known as selective reduced integration). This approach helps to prevent volumetric mesh locking in nearly incompressible cases. It replaces volumetric strain at the Gauss integration point with the average volumetric strain of the elements. For more information, see the *Mechanical APDL Theory Reference*.

CPT215 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements and pore pressure included in the overall nodal solution
- Additional element output as shown in Table 215.1: CPT215 Output Definitions (p. 1133)

The element stress directions are parallel to the element coordinate system, as shown in Figure 215.2: CPT215 Stress Output (p. 1133). A general description of solution output is given in The Item and Sequence Number Table (p. 52). See the *Basic Analysis Guide* for ways to view results.

Figure 215.2: CPT215 Stress Output

The element stress directions are parallel to the global coordinate system.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 215.1: CPT215 Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
NODES	Nodes - I, J, K, L, M, N, O, P	-	Y
MAT	Material number	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	2
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	-	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	Y	Y
S:1, 2, 3	Principal stresses	-	Y
S:INT	Stress intensity	-	Y
S:EQV	Equivalent stress	-	Y
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	Y	Y
EPEL:1, 2, 3	Principal elastic strains	-	Y
EPEL:EQV	Equivalent elastic strains [3]	-	Y
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	1	1
EPTH:EQV	Equivalent thermal strains [3]	1	1
ESIG:X, Y, Z, XY, XZ, YZ	Effective stresses	-	Y

1. Output only if element has a thermal load

2. Available only at centroid as a ***GET** item

3. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.

Table 215.2: CPT215 Item and Sequence Numbers (p. 1134) lists output available via **ETABLE** using the Sequence Number method. See Element Table for Variables Identified By Sequence Number in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this document for more information. The following notation is used in Table 215.2: CPT215 Item and Sequence Numbers (p. 1134):

Name
output quantity as defined in the Table 215.1: CPT215 Output Definitions (p. 1133)

Item
predetermined Item label for **ETABLE** command

I,J,...,P
sequence number for data at nodes I, J, ..., P

Table 215.2: CPT215 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	I	J	K	L	M	N	O	P
P1	SMISC	2	1	4	3	-	-	-	-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input								
	Item	I	J	K	L	M	N	O	P
P2	SMISC	5	6	-	-	8	7	-	-
P3	SMISC	-	9	10	-	-	12	11	-
P4	SMISC	-	-	13	14	-	-	16	15
P5	SMISC	18	-	-	17	19	-	-	20
P6	SMISC	-	-	-	-	21	22	23	24

CPT215 Assumptions and Restrictions

- The element must not have a zero volume. Also, the element may not be twisted such that the element has two separate volumes (which occurs most frequently when the element is numbered improperly). Elements may be numbered either as shown in Figure 215.1: CPT215 Structural Solid Geometry (p. 1131) or may have the planes IJKL and MNOP interchanged.

- When degenerated into a tetrahedron, wedge, or pyramid element shape (described in Degenerated Shape Elements (p. 39)), the corresponding degenerated shape functions are used. Degeneration to a pyramidal form should be used with caution. The element sizes, when degenerated, should be small to minimize the stress gradients. Pyramid elements are best used as filler elements or in meshing transition zones.

- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). It is ignored in geometrically linear analyses (**NLGEOM**,OFF). Prestress effects can be activated by the **PSTRES** command.

CPT215 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

CPT216

3-D 20-Node Coupled Pore-Pressure Mechanical Solid

MP ME <> <> <> <> <> <> PP EME MFS
Product Restrictions

CPT216 Element Description

CPT216 is a higher-order 3-D 20-node coupled pore-pressure mechanical solid element. The element exhibits quadratic displacement behavior and is well suited to modeling irregular meshes.

The element is defined by twenty nodes having four degrees of freedom at each corner node:

- The translations in the nodal x, y and z directions
- One pore-pressure degree of freedom
- Three degrees of freedom at the mid-side nodes: the translations in the nodal x, y and z directions.

CPT216 has elasticity, stress stiffening, large deflection, and large strain capabilities. The element can have any spatial orientation. Various printout options are available.

See CPT216 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 216.1: CPT216 Geometry

A lower-order version of this element is CPT215.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CPT216 Input Data

The geometry, node locations, and the element coordinate system for this element are shown in Figure 216.1: CPT216 Geometry (p. 1137). A prism-shaped element may be formed by defining the same node numbers for nodes K, L, and S; nodes A and B; and nodes O, P, and W. A tetrahedral-shaped element and a pyramid-shaped element may also be formed as shown in Figure 186.1: SOLID186 Homogeneous Structural Solid Geometry (p. 976). (CPT217 is similar, but is a 10-node tetrahedral element.)

In addition to the nodes, the element input data includes the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems (p. 55).

Element loads are described in Nodal Loading (p. 49). Pressures may be input as surface loads on the element faces as shown by the circled numbers in Figure 216.1: CPT216 Geometry (p. 1137). Positive pressures act into the element.

Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). If all corner node temperatures are specified, each midside node temperature defaults to the average temperature of its adjacent corner nodes. For any other input temperature pattern, unspecified temperatures default to TUNIF.

As described in Coordinate Systems (p. 55), you can use the **ESYS** command to orient the material properties and strain/stress output. Use the **RSYS** command to choose output that follows the material coordinate system or the global coordinate system.

The effects of pressure load stiffness are automatically included for this element, and the element generally produces an unsymmetric matrix. To avoid convergence difficulty, issue the **NROPT**,UNSYM command to use the unsymmetric solver.

The following table summarizes the element input. Element Input (p. 43) provides a general description of element input.

CPT216 Input Summary

Nodes

I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B

Degrees of Freedom

UX, UY, UZ, PRES

Real Constants

None

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, EY, EZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX,THSY, THSZ), PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), DENS, GXY, GYZ, GXZ, ALPD, BETD

Surface Loads

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Body Loads

Temperatures --
T(I), T(J),T(K), T(L), T(M), T(N), T(O), T(P), T(Q), T(R), T(S), T(T), T(U), T(V), T(W), T(X), T(Y), T(Z), T(A), T(B)

Special Features

Initial state
Large deflection
Large strain
Stress stiffening

KEYOPT(2)
Element technology:

0 --
Uniform reduced integration (default)

1 --
Full integration

CPT216 Element Technology

CPT216 uses the uniform reduced integration method or the full integration method, as follows:

- **Uniform reduced integration method**

  Helps to prevent volumetric mesh locking in nearly incompressible cases. However, hourglass mode might propagate in the model if there are not at least two layers of elements in each direction.

- **Full integration**

  The full integration method does not cause hourglass mode, but can cause volumetric locking in nearly incompressible cases. This method is used primarily for purely linear analyses, or when the model has only one layer of elements in each direction.

CPT216 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements and pore pressures included in the overall nodal solution
- Additional element output as shown in Table 216.1: CPT216 Output Definitions (p. 1140)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 216.2: CPT216 Stress Output

The element stress directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 216.1: CPT216 Output Definitions

Name	Definition	O	R
EL	Element number and name	-	Y
NODES	Nodes - I, J, K, L, M, N, O, P	-	Y
MAT	Material number	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	2
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	-	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	Y	Y
S:1, 2, 3	Principal stresses	-	Y
S:INT	Stress intensity	-	Y
S:EQV	Equivalent stress	-	Y
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	Y	Y
EPEL:1, 2, 3	Principal elastic strains	Y	-
EPEL:EQV	Equivalent elastic strains	Y	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	1	1
EPTH:EQV	Equivalent thermal strains	1	1
ESIG:X, Y, Z, XY, XZ, YZ	Effective stresses	-	Y

1. Output only if element has a thermal load

2. Available only at centroid as a ***GET** item.

Table 216.2: CPT216 Item and Sequence Numbers (p. 1141) lists output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this document for more information. The following notation is used in Table 216.2: CPT216 Item and Sequence Numbers (p. 1141):

Name
output quantity as defined in Table 216.1: CPT216 Output Definitions (p. 1140)

Item
predetermined Item label for **ETABLE**

I,J,...,B
sequence number for data at nodes I, J, ..., B

Table information below to be provided.

Table 216.2: CPT216 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input									
	Item	I	J	K	L	M	N	O	P	Q,...,B
P1	SMISC	2	1	4	3	-	-	-	-	-
P2	SMISC	5	6	-	-	8	7		-	-
P3	SMISC	-	9	10	-	-	12	11	-	-
P4	SMISC	-	-	13	14	-	-	16	15	-
P5	SMISC	18	-	-	17	19	-	-	20	-
P6	SMISC	-	-	-	-	21	22	23	24	-

CPT216 Assumptions and Restrictions

- The element must not have a zero volume. Also, the element may not be twisted such that the element has two separate volumes (which occurs most frequently when the element is numbered improperly). Elements may be numbered either as shown in Figure 216.1: CPT216 Geometry (p. 1137) or may have the planes IJKL and MNOP interchanged.

- An edge with a removed midside node implies that the displacement varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information on the use of midside nodes.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Use at least two elements in each direction to avoid the hourglass effect when using reduced integration (KEYOPT(2) = 0).
- When degenerated into a tetrahedron, wedge, or pyramid element shape (described in Degenerated Shape Elements (p. 39)), the corresponding degenerated shape functions are used. Degeneration to a pyramidal form should be used with caution. The element sizes, when degenerated, should be small to minimize the stress gradients. Pyramid elements are best used as filler elements or in meshing transition zones.
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). It is ignored in geometrically linear analyses (**NLGEOM**,OFF). Prestress effects can be activated via the **PSTRES** command.

CPT216 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CPT217

3-D 10-Node Coupled Pore-Pressure Mechanical Solid

MP ME <> <> <> <> <> <> PP EME MFS
Product Restrictions

CPT217 Element Description

CPT217 is a higher-order 3-D 10-node coupled pore-pressure mechanical solid element. The element has a quadratic displacement, and linear pore-pressure and temperature behavior.

The element is defined by ten nodes having four degrees of freedom at each corner node:

- The translations in the nodal x, y and z directions
- One pore-pressure degree of freedom
- Three degrees of freedom at the midside nodes: the translations in the nodal x, y and z directions.

CPT217 has elasticity, stress stiffening, large deflection, and large strain capabilities. See CPT217 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 217.1: CPT217 Geometry

CPT217 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 217.1: CPT217 Geometry (p. 1143).

In addition to the nodes, the element input data includes the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Linear Material Properties in the *Material Reference*.

Element loads are described in Nodal Loading (p. 49). Pressures can be input as surface loads on the element faces as shown by the circled numbers in Figure 217.1: CPT217 Geometry (p. 1143). Positive pressures act into the element. Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). If all corner node temperatures are specified, each midside node temperature defaults to the average

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

temperature of its adjacent corner nodes. For any other input temperature pattern, unspecified temperatures default to TUNIF.

As described in Coordinate Systems (p. 55), you can use the **ESYS** command to orient the material properties and strain/stress output. Issue the **RSYS** command to choose output that follows the material coordinate system or the global coordinate system.

The effects of pressure load stiffness are automatically included for this element, and the element generally produces an unsymmetric matrix. To avoid convergence difficulty, issue the **NROPT**,UNSYM command to use the unsymmetric solver.

The following table summarizes the element input. Element Input (p. 43) gives a general input description.

CPT217 Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R

Degrees of Freedom
UX, UY, UZ, PRES

Real Constants
None

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, EY, EZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ), PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), DENS, GXY, GYZ, GXZ, ALPD, BETD

Surface Loads

Pressures --
face 1 (J-I-K), face 2 (I-J-L), face 3 (J-K-L), face 4 (K-I-L)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P), T(Q), T(R)

Special Features

Initial state
Large deflection
Large strain
Stress stiffening

CPT217 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements and pore pressure included in the overall nodal solution
- Additional element output as shown in Table 217.1: CPT217 Element Output Definitions (p. 1145)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The element stress directions are parallel to the element coordinate system, as shown in Figure 217.2: CPT217 Stress Output (p. 1145). A general description of solution output is given in The Item and Sequence Number Table (p. 52). See the *Basic Analysis Guide* for ways to view results.

Figure 217.2: CPT217 Stress Output

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 217.1: CPT217 Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Nodes - I, J, K, L	-	Y
MAT	Material number	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	2
TEMP	Temperatures T(I), T(J), T(K), T(L)	-	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	Y	Y
S:1, 2, 3	Principal stresses	-	Y
S:INT	Stress intensity	-	Y
S:EQV	Equivalent stress	-	Y
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	Y	Y
EPEL:1, 2, 3	Principal elastic strains	Y	-
EPEL:EQV	Equivalent elastic strains [3]	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	1	1
EPTH: EQV	Equivalent thermal strains [3]	1	1
EPTO:X, Y, Z, XY, YZ, XZ	Total mechanical strains (EPEL + EPPL + EPCR)	Y	-
EPTO:EQV	Total equivalent mechanical strains (EPEL + EPPL + EPCR)	Y	-
ESIG:X, Y, Z, XY, XZ, YZ	Effective stresses	-	Y

1. Output only if element has a thermal load
2. Available only at centroid as a ***GET** item.
3. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.

Table 217.2: CPT217 Item and Sequence Numbers (p. 1146) lists output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this document for more information. The following notation is used in the table:

Name
Output quantity as defined in Table 217.1: CPT217 Element Output Definitions (p. 1145)

Item
Predetermined Item label for **ETABLE** command

I,J,...,R
Sequence number for data at nodes I, J, ..., R

Table 217.2: CPT217 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	I	J	K	L	M,...,R
P1	SMISC	2	1	3	-	-
P2	SMISC	4	5	-	6	-
P3	SMISC	-	7	8	9	-
P4	SMISC	11	-	10	12	-

CPT217 Assumptions and Restrictions

- The element must not have a zero volume.
- Elements may be numbered either as shown in Figure 217.1: CPT217 Geometry (p. 1143) or may have node L below the I, J, K plane.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- An edge with a removed midside node implies that the displacement varies linearly, rather than parabolically, along that edge. For information about using midside nodes, see Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide*.
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). It is ignored in geometrically linear analyses (**NLGEOM**,OFF). Prestress effects can be activated via the **PSTRES** command.

CPT217 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

FLUID220

3-D Acoustic Fluid 20-Node Solid Element

MP ME <> <> <> <> <> <> PP EME MFS
Product Restrictions

FLUID220 Element Description

FLUID220 is a higher order 3-D 20-node solid element that exhibits quadratic pressure behavior. Use it for modeling the fluid medium and the interface in fluid-structure interaction problems.

For detailed information about this element, see Elements for Acoustic Analysis (p. 104).

Figure 220.1: FLUID220 Geometry

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

FLUID221

3-D Acoustic Fluid 10-Node Solid Element

MP ME <> <> <> <> <> <> PP EME MFS
Product Restrictions

FLUID221 Element Description

FLUID221 is a higher order 3-D 10-node solid element that exhibits quadratic pressure behavior. Use it for modeling the fluid medium and the interface in fluid-structure interaction problems.

For detailed information about this element, see Elements for Acoustic Analysis (p. 104).

Figure 221.1: FLUID221 Geometry

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

PLANE223

2-D 8-Node Coupled-Field Solid

MP ME <> <> <> <> <> <> PP EME MFS
Product Restrictions

PLANE223 Element Description

PLANE223 has the following capabilities:

- Structural-Thermal
- Piezoresistive
- Electroelastic
- Piezoelectric
- Thermal-Electric
- Structural-Thermoelectric
- Thermal-Piezoelectric
- Structural-Diffusion
- Thermal-Diffusion
- Structural-Thermal-Diffusion

The element has eight nodes with up to four degrees of freedom per node. Structural capabilities include elasticity, plasticity, viscoelasticity, viscoplasticity, creep, large strain, large deflection, stress stiffening effects, and prestress effects. Thermoelectric capabilities include Seebeck, Peltier, and Thomson effects, as well as Joule heating. In addition to thermal expansion, structural-thermal capabilities include the piezocaloric effect in dynamic analyses. The Coriolis effect is available for analyses with structural degrees of freedom. The thermoplastic effect is available for analyses with structural and thermal degrees of freedom. The diffusion expansion effect is available for analyses with structural and diffusion degrees of freedom. See PLANE223 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 223.1: PLANE223 Geometry

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PLANE223 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 223.1: PLANE223 Geometry (p. 1153). The element input data includes eight nodes and structural, thermal, and electrical material properties. The type of units (MKS or user defined) is specified through the **EMUNIT** command. **EMUNIT** also determines the value of free-space permittivity EPZRO. The **EMUNIT** defaults are MKS units and EPZRO = 8.85e-12 Farads/meter.

KEYOPT(1) determines the element DOF set and the corresponding force labels and reaction solution. KEYOPT(1) is set equal to the sum of the field keys shown in Table 223.1: PLANE223 Field Keys (p. 1154). For example, KEYOPT(1) is set to 11 for a structural-thermal analysis (structural field key + thermal field key = 1 + 10). For a structural-thermal analysis, UX, UY, and TEMP are the DOF labels and force and heat flow are the reaction solution.

Table 223.1: PLANE223 Field Keys

Field	Field Key	DOF Label	Force Label	Reaction Solution
Structural	1	UX, UY	FX, FY	Force
Thermal	10	TEMP	HEAT	Heat Flow
Electric Conduction	100	VOLT	AMPS	Electric Current
Electrostatic	1000	VOLT	CHRG	Electric Charge
Diffusion	100000	CONC	RATE	Diffusion Flow Rate

The coupled-field analysis KEYOPT(1) settings, DOF labels, force labels, reaction solutions, and analysis types are shown in the following table.

Table 223.2: PLANE223 Coupled-Field Analyses

Coupled-Field Analysis	KEYOPT(1)	DOF Label	Force Label	Reaction Solution	Analysis Type
Structural-Thermal [1], [2]	11	UX, UY, TEMP	FX, FY, HEAT	Force, Heat Flow	Static Full Harmonic Full Transient
Piezoresistive	101	UX, UY, VOLT	FX, FY, AMPS	Force, Electric Current	Static Full Transient
Electroelastic	1001 [3]	UX, UY, VOLT	FX, FY, CHRG	Force, Electric Charge (positive)	Static Full Transient

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Coupled-Field Analysis	KEY-OPT(1)	DOF Label	Force Label	Reaction Solution	Analysis Type
Piezoelectric	1001 [3]	UX, UY, VOLT	FX, FY, CHRG	Force, Electric Charge (negative)	Static Modal Full Harmonic Full Transient
Thermal-Electric	110	TEMP, VOLT	HEAT, AMPS	Heat Flow, Electric Current	Static Full Transient
Structural-Thermoelectric [1]	111	UX, UY, TEMP, VOLT	FX, FY, HEAT, AMPS	Force, Heat Flow, Electric Current	Static Full Transient
Thermal-Piezoelectric [1], [2]	1011	UX, UY, TEMP, VOLT	FX, FY, HEAT, CHRG	Force, Heat Flow, Electric Charge (negative)	Static Full Harmonic Full Transient
Structural-Diffusion [1]	100001	UX, UY, UZ, CONC	FX, FY, FZ, RATE	Force, Diffusion Flow Rate	Static Full Transient
Thermal-Diffusion	100010	TEMP, CONC	HEAT, RATE	Heat Flow, Diffusion Flow Rate	Static Full Transient
Structural-Thermal-Diffusion [1]	100011	UX, UY, UZ, TEMP,	FX, FY, FZ, HEAT,	Force, Heat Flow,	Static Full Transient

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Coupled-Field Analysis	KEYOPT(1)	DOF Label	Force Label	Reaction Solution	Analysis Type
		CONC	RATE	Diffusion Flow Rate	

1. For static and full transient analyses, KEYOPT(2) can specify a strong (matrix) or weak (load vector) structural-thermal coupling and/or structural-diffusion coupling.

2. For full harmonic analyses, strong structural-thermal coupling only applies.

3. The electrostatic-structural analysis available with KEYOPT(1) = 1001 defaults to an electroelastic analysis (electrostatic force coupling) unless a piezoelectric matrix is specified on **TB**,PIEZ.

As shown in the following table, material property requirements consist of those required for the individual fields (structural, thermal, electric conduction, electrostatic, or diffusion) and those required for field coupling. Individual material properties are defined via the **MP** and **MPDATA** commands. Nonlinear material models are defined via the **TB** command. (The nonlinear material models do not apply to piezoelectric analyses (**TB**,PIEZ) where KEYOPT(1) = 1001 or 1011).

Table 223.3: PLANE223 Material Properties and Material Models

ALL ANALYSES WITH A STRUCTURAL FIELD	
Structural	EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), GXY, GYZ, GXZ, DENS, ALPD, BETD, DMPR ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ, or THSX, THSY, THSZ), REFT --- Anisotropic elasticity, Bilinear isotropic hardening, Bilinear kinematic hardening, Cast iron, Chaboche nonlinear kinematic hardening, Creep, Extended Drucker-Prager, Elasticity, Gurson pressure-dependent plasticity, Hill anisotropy, Multilinear isotropic hardening, Multilinear kinematic hardening, Voce isotropic hardening law, Plasticity, Prony series constants for viscoelastic materials, Rate-dependent plasticity (viscoplasticity), Material structural damping, Shift function for viscoelastic materials, Shape memory alloy, Uniaxial stress-strain relation
ADDITIONAL MATERIAL PROPERTIES FOR STRUCTURAL-THERMAL ANALYSES	
Thermal	KXX, KYY, DENS, C, ENTH, HF
Coupling	ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ, or THSX, THSY, THSZ), REFT, QRATE
ADDITIONAL MATERIAL PROPERTIES FOR PIEZORESISTIVE ANALYSES	
Electric	RSVX, RSVY, PERX, PERY
Coupling	Piezoresistivity
ADDITIONAL MATERIAL PROPERTIES FOR ELECTROELASTIC ANALYSES	
Electric	PERX, PERY --- Anisotropic electric permittivity

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ADDITIONAL MATERIAL PROPERTIES FOR PIEZOELECTRIC ANALYSES	
Electric	PERX, PERY, LSST (and/or RSVX, RSVY) --- Anisotropic electric permittivity
Coupling	Piezoelectric matrix
ADDITIONAL MATERIAL PROPERTIES FOR STRUCTURAL-DIFFUSION ANALYSES	
Diffusion	DXX, DYY, DZZ, CSAT
Coupling	BETX, BETY, BETZ, CREF
MATERIAL PROPERTIES FOR THERMAL-ELECTRIC ANALYSES	
Thermal	KXX, KYY, DENS, C, ENTH, HF
Electric	RSVX, RSVY, PERX, PERY
Coupling	SBKX, SBKY
ADDITIONAL MATERIAL PROPERTIES FOR STRUCTURAL-THERMOELECTRIC ANALYSES	
Thermal	KXX, KYY, DENS, C, ENTH, HF
Electric	RSVX, RSVY, PERX, PERY
Coupling	ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ, or THSX, THSY, THSZ), SBKX, SBKY, PZRS, QRATE --- Piezoresistivity
ADDITIONAL MATERIAL PROPERTIES FOR THERMAL-PIEZOELECTRIC ANALYSES	
Thermal	KXX, KYY, DENS, C, ENTH, HF
Electric	PERX, PERY, LSST (and/or RSVX, RSVY) --- Anisotropic electric permittivity
Coupling	ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ, or THSX, THSY, THSZ), PIEZ, QRATE
MATERIAL PROPERTIES FOR THERMAL-DIFFUSION ANALYSES	
Thermal	KXX, KYY, DENS, C, ENTH, HF
Diffusion	DXX, DYY, DZZ, CSAT
ADDITIONAL MATERIAL PROPERTIES FOR STRUCTURAL-THERMAL-DIFFUSION ANALYSES	
Thermal	KXX, KYY, DENS, C, ENTH, HF
Diffusion	DXX, DYY, DZZ, CSAT
Coupling	BETX, BETY, BETZ, CREF

Various combinations of nodal loading are available for this element (depending upon the KEYOPT(1) value). Nodal loads are defined with the **D** and the **F** commands. Nodal forces, if any, should be input per unit of depth for a plane analysis and on a full 360° basis for an axisymmetric analysis.

Element loads are described in Nodal Loading (p. 49). Surface loads may be input on the element faces indicated by the circled numbers in Figure 223.1: PLANE223 Geometry (p. 1153) using the **SF** and **SFE**

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

commands. Positive pressures act into the element. Body loads may be input at the element's nodes or as a single element value using the **BF** and **BFE** commands.

PLANE223 surface and body loads are given in the following table.

Table 223.4: PLANE223 Surface and Body Loads

Coupled-Field Analysis	KEYOPT(1)	Load Type	Load	Command Label
Structural-Thermal	11	Surface	Pressure	PRES
			Convection Heat Flux Radiation	CONV HFLUX RDSF
		Body	Force Density	FORC
			Heat Generation -- Nodes I through P	HGEN
Piezoresistive	101	Surface	Pressure	PRES
		Body	Force Density	FORC
			Temperature -- Nodes I through P	TEMP
Electroelastic and Piezoelectric	1001	Surface	Pressure Surface Charge Density	PRES CHRGS[1]
		Body	Force Density	FORC
			Temperature -- Nodes I through P	TEMP
			Volume Charge Density -- Nodes I through P	CHRGD[1]
Thermal-Electric	110	Surface	Convection Heat Flux Radiation	CONV HFLUX RDSF
		Body	Heat Generation -- Nodes I through P	HGEN
Structural-Thermoelectric	111	Surface	Pressure	PRES
			Convection Heat Flux Radiation	CONV HFLUX RDSF
		Body	Force Density	FORC

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Coupled-Field Analysis	KEY-OPT(1)	Load Type	Load	Command Label
			Heat Generation -- Nodes I through P	HGEN
Thermal-Piezoelectric	1011	Surface	Pressure Surface Charge Density	PRES CHRGS[1]
			Convection Heat Flux Radiation	CONV HFLUX RDSF
		Body	Force Density	FORC
			Heat Generation -- Nodes I through P	HGEN
			Volume Charge Density -- Nodes I through P	CHRGD[1]
Structural-Diffusion	100001	Surface	Pressure	PRES
			Diffusion Flux	DFLUX
		Body	Force Density	FORC
			Temperature -- Nodes I through P	TEMP
			Diffusing Substance Generation -- Nodes I through P	DGEN
Thermal-Diffusion	100010	Surface	Convection Heat Flux Radiation	CONV HFLUX RDSF
			Diffusion Flux	DFLUX
		Body	Heat Generation -- Nodes I through P	HGEN
			Diffusing Substance Generation -- Nodes I through P	DGEN
Structural-Thermal-Diffusion	100011	Surface	Pressure	PRES
			Convection Heat Flux Radiation	CONV HFLUX RDSF

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Coupled-Field Analysis	KEYOPT(1)	Load Type	Load	Command Label
			Diffusion Flux	DFLUX
		Body	Force Density	FORC
			Heat Generation -- Nodes I through P	HGEN
			Diffusing Substance Generation -- Nodes I through P	DGEN

1. CHRGS and CHRGD are interpreted as negative surface charge density and negative volume charge density, respectively.

Automatic element technology selections are given in the following table.

Table 223.5: Automatic Element Technology Selection

Coupled-Field Analysis	ETCONTROL Command Suggestions/Resettings
Structural-Thermal (KEYOPT(1) = 11) Structural-Thermoelactric (KEYOPT(1) = 111)	KEYOPT(2) =1 for nonlinear inelastic materials

A summary of the element input is given in "PLANE223 Input Summary" (p. 1160). A general description of element input is given in Element Input (p. 43). For axisymmetric applications see Harmonic Axisymmetric Elements (p. 83).

PLANE223 Input Summary

Nodes

I, J, K, L, M, N, O, P

Degrees of Freedom

Set by KEYOPT(1). See Table 223.2: PLANE223 Coupled-Field Analyses (p. 1154).

Real Constants

None

Material Properties

See Table 223.3: PLANE223 Material Properties and Material Models (p. 1156).

Surface Loads

See Table 223.4: PLANE223 Surface and Body Loads (p. 1158).

Body Loads

See Table 223.4: PLANE223 Surface and Body Loads (p. 1158).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Special Features

Element technology autoselect
Large deflection
Large strain
Linear perturbation (piezoelectric analysis only [KEYOPT(1) = 1001])
Nonlinear stabilization
Stress stiffening

KEYOPT(1)

Element degrees of freedom. See Table 223.2: PLANE223 Coupled-Field Analyses (p. 1154).

KEYOPT(2)

Structural-thermal coupling method in structural-thermal, structural-thermoelectric, thermal-piezoelectric, or structural-thermal-diffusion analyses (KEYOPT(1) = 11, 111, 1011, or 100011) or structural-diffusion coupling method in structural-diffusion or structural-thermal-diffusion analyses (KEYOPT(1)=100001 or 100011).

0 --
Strong (matrix) coupling – produces an unsymmetric matrix. In a linear analysis, a coupled response is achieved after one iteration.

1 --
Weak (load vector) coupling – produces a symmetric matrix and requires at least two iterations to achieve a coupled response.

KEYOPT(3)

Element behavior:

0 --
Plane stress

1 --
Axisymmetric

2 --
Plane strain

KEYOPT(4)

Electrostatic force in electroelastic analysis (KEYOPT(1) = 1001):

0 --
Applied to every element node.

1 --
Applied to the air-structure interface or to element nodes that have constrained structural degrees of freedom.

2 --
Not applied.

For more information, see Electroelastic Analysis in the *Coupled-Field Analysis Guide*.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(9)

Thermoelastic damping (piezocaloric effect) in structural-thermal, structural-thermoelastic, or thermal-piezoelectric analyses (KEYOPT(1) = 11, 111, or 1011). Applicable to harmonic and transient analyses only.

0 --

Active

1 --

Suppressed (required for frictional heating analyses)

KEYOPT(10)

Specific heat matrix in structural-thermal, thermal-electric, structural-thermoelectric, thermal-piezoelectric, thermal-diffusion, structural-thermal-diffusion analyses (KEYOPT(1) = 11, 110, 111, 1011, 100001, or 100011), and damping matrix in structural-diffusion, thermal-diffusion, and structural-thermal-diffusion analyses (KEYOPT(1) = 100001, 100010, and 100011).

0 --

Consistent

1 --

Diagonalized

2 --

Diagonalized. Temperature-dependent specific heat or enthalpy is evaluated at the element centroid.

PLANE223 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in Table 223.6: PLANE223 Element Output Definitions (p. 1162).

The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 223.6: PLANE223 Element Output Definitions

Name	Definition	O	R
ALL ANALYSES			
EL	Element Number	-	Y
NODES	Nodes - I, J, K, L, M, N, O, P	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
MAT	Material number	-	Y
VOLU:	Volume	-	Y
XC, YC	Location where results are reported	-	2
ALL ANALYSES WITH A STRUCTURAL FIELD			
S:X, Y, Z, XY	Stresses (SZ = 0.0 for plane stress elements)	-	1
S:1, 2, 3	Principal stresses	-	1
S:EQV	Equivalent stress	-	1
EPEL:X, Y, Z, XY	Elastic strains	-	1
EPTH:X, Y, Z, XY	Thermal strains	-	1
EPTH:EQV	Equivalent thermal strain [3]	-	1
EPPL:X, Y, Z, XY	Plastic strains	-	1
EPPL:EQV	Equivalent plastic strain [3]	-	1
EPCR:X, Y, Z, XY	Creep strains	-	1
EPCR:EQV	Equivalent creep strain [3]	-	1
EPTO:X, Y, Z, XY	Total mechanical strains (EPEL + EPPL + EPCR)	-	-
EPTO:EQV	Total equivalent mechanical strain (EPEL + EPPL + EPCR)	-	-
ADDITIONAL OUTPUT FOR STRUCTURAL-THERMAL ANALYSES (KEYOPT(1) = 11)			
TG:X, Y, SUM	Thermal gradient components and vector magnitude	-	1
TF:X, Y, SUM	Thermal flux components and vector magnitude	-	1
UT	Total strain energy [7]	-	1
PHEAT	Plastic heat generation rate per unit volume	-	1
ADDITIONAL OUTPUT FOR PIEZORESISTIVE ANALYSES (KEYOPT(1) = 101)			
TEMP	Input temperatures	-	Y
EF:X, Y, SUM	Electric field components (X, Y) and vector magnitude	-	1
JC:X, Y, SUM	Conduction current density components (X, Y) and vector magnitude	-	1
JS:X, Y, SUM	Current density components (in the global Cartesian coordinate system) (X, Y) and vector magnitude [4]	-	1
JHEAT	Joule heat generation per unit volume [5]	-	1
ADDITIONAL OUTPUT FOR ELECTROELASTIC ANALYSES (KEYOPT(1) = 1001)			
TEMP	Input temperatures	-	Y
EF:X, Y, SUM	Electric field components (X, Y) and vector magnitude	-	1
D:X, Y, SUM	Electric flux density components (X, Y) and vector magnitude	-	1
FMAG:X, Y, SUM	Electrostatic force components (X, Y) and vector magnitude	-	1
UE, UD	Stored elastic and dielectric energies	-	1
ADDITIONAL OUTPUT FOR PIEZOELECTRIC ANALYSES (KEYOPT(1) = 1001)			
TEMP	Input temperatures	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
EF:X, Y, SUM	Electric field components (X, Y) and vector magnitude	-	1
D:X, Y, SUM	Electric flux density components (X, Y) and vector magnitude	-	1
JHEAT	Joule heat generation per unit volume [5], [6]	-	1
UE, UD	Stored elastic and dielectric energies	-	1
ADDITIONAL OUTPUT FOR STRUCTURAL-DIFFUSION ANALYSES (KEYOPT(1)=100001)			
TEMP	Input temperatures	-	Y
EPDI:X, Y, Z, XY	Diffusion strains	-	1
CG:X, Y, SUM	Concentration gradient components and vector magnitude	-	1
DF:X, Y, SUM	Diffusion flux components and vector magnitude	-	1
CONC	Element concentration [8]	-	1
ADDITIONAL OUTPUT FOR THERMAL-ELECTRIC ANALYSES (KEYOPT(1) = 110)			
TG:X, Y, SUM	Thermal gradient components and vector magnitude	-	1
TF:X, Y, SUM	Thermal flux components and vector magnitude	-	1
EF:X, Y, SUM	Electric field components and vector magnitude	-	1
JC:X, Y, SUM	Conduction current density components and vector magnitude	-	1
JS:X, Y, SUM	Current density components (in the global Cartesian coordinate system) and vector magnitude [4]	-	1
JHEAT	Joule heat generation per unit volume [5], [6]	-	1
ADDITIONAL OUTPUT FOR STRUCTURAL-THERMOELECTRIC ANALYSES (KEYOPT(1) = 111)			
TG:X, Y, SUM	Thermal gradient components and vector magnitude	-	1
TF:X, Y, SUM	Thermal flux components and vector magnitude	-	1
EF:X, Y, SUM	Electric field components and vector magnitude	-	1
JC:X, Y, SUM	Conduction current density components and vector magnitude	-	1
JS:X, Y, SUM	Current density components (in the global Cartesian coordinate system) and vector magnitude [4]	-	1
JHEAT	Joule heat generation per unit volume [5], [6]	-	1
UT	Total strain energy [7]	-	1
PHEAT	Plastic heat generation rate per unit volume	-	1
ADDITIONAL OUTPUT FOR THERMAL-PIEZOELECTRIC ANALYSES (KEYOPT(1) = 1011)			
TG:X, Y, SUM	Thermal gradient components and vector magnitude	-	1
TF:X, Y, SUM	Thermal flux components and vector magnitude	-	1
EF:X, Y, SUM	Electric field components and vector magnitude	-	1
D:X, Y, SUM	Electric flux density components and vector magnitude	-	1
JHEAT	Joule heat generation per unit volume [5], [6]	-	1
UE, UD	Stored elastic and dielectric energies	-	1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
UT	Total strain energy [7]	-	1
PHEAT	Plastic heat generation rate per unit volume	-	1
THERMAL-DIFFUSION ANALYSES (KEYOPT(1) = 100010)			
TG:X, Y, SUM	Thermal gradient components and vector magnitude	-	1
TF:X, Y, SUM	Thermal flux components and vector magnitude	-	1
CG:X, Y, SUM	Concentration gradient components and vector magnitude	-	1
DF:X, Y, SUM	Diffusion flux components and vector magnitude	-	1
CONC	Element concentration [8]	-	1
ADDITIONAL OUTPUT FOR STRUCTURAL-THERMAL-DIFFUSION ANALYSES (KEYOPT(1) = 100011)			
EPDI:X, Y, Z, XY	Diffusion strains	-	1
TG:X, Y, SUM	Thermal gradient components and vector magnitude	-	1
TF:X, Y, SUM	Thermal flux components and vector magnitude	-	1
CG:X, Y, SUM	Concentration gradient components and vector magnitude	-	1
DF:X, Y, SUM	Diffusion flux components and vector magnitude	-	1
CONC	Element concentration [8]	-	1

1. Solution values are output only if calculated (based on input values).

2. Available only at centroid as a ***GET** item.

3. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.

4. JS represents the sum of element conduction and displacement current densities.

5. Calculated Joule heat generation rate per unit volume (JHEAT) may be made available for a subsequent thermal analysis with companion thermal elements.

6. For a time-harmonic analysis, Joule losses (JHEAT) are time-averaged. These values are stored in both the real and imaginary data sets. For more information, see Quasistatic Electric Analysis in the *Mechanical APDL Theory Reference*.

7. For a time-harmonic analysis, total strain energy (UT) is time-averaged. These values are stored in both the real and imaginary data sets. For more information, see Thermoelasticity in the *Mechanical APDL Theory Reference*.

8. With the normalized concentration approach, CONC is the actual concentration obtained by multiplying the saturated concentration (**MP**,CSAT) and the normalized concentration evaluated at the element centroid. For more information, see Normalized Concentration Approach in the *Mechanical APDL Theory Reference*.

Table 223.7: PLANE223 Item and Sequence Numbers (p. 1166) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) of the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) of this manual for more information. The following notation is used in Table 223.7: PLANE223 Item and Sequence Numbers (p. 1166):

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name
output quantity as defined in the Table 223.6: PLANE223 Element Output Definitions (p. 1162)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 223.7: PLANE223 Item and Sequence Numbers

Output Quantity Name	ETABLE Command Input	
	Item	E
CONC	SMISC	1
UE	NMISC	1
UD	NMISC	2
UT	NMISC	4
PHEAT	NMISC	5

PLANE223 Assumptions and Restrictions

- PLANE223 assumes a unit thickness.
- PLANE223 uses 2 x 2 and 3 point integration rules to calculate the element matrices and load vectors for the quad and triangle geometries, respectively.
- In a piezoelectric analysis, electric charge loading is interpreted as negative electric charge or negative charge density.
- In a coupled-field analysis with structural degrees of freedom, the model should have at least two elements in each direction to avoid the hourglass mode.
- CSAT cannot be temperature-dependent.
- The optimized nonlinear solution defaults (**SOLCONTROL**) are not available in coupled-field analyses.
- The element must lie in a global X-Y plane as shown in Figure 223.1: PLANE223 Geometry (p. 1153) and the Y-axis must be the axis of symmetry for axisymmetric analyses.
- An axisymmetric structure should be modeled in the +X quadrants.
- A face with a removed midside node implies that the degrees-of-freedom vary linearly, rather than parabolically, along that face. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.
- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, the elements must have the same reaction solution for the VOLT DOF. Elements that have an electric charge reaction solution must all have the same electric charge reaction sign. For more information, see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*.
- This element does not support fully incompressible materials.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Graphical Solution Tracking (**/GST**) is not supported with the coupled-diffusion analyses (KEYOPT(1)=100001, 100010, and 100011).

PLANE223 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID226

3-D 20-Node Coupled-Field Solid

MP ME <> <> <> <> <> <> PP EME MFS
Product Restrictions

SOLID226 Element Description

SOLID226 has the following capabilities:

- Structural-Thermal
- Piezoresistive
- Electroelastic
- Piezoelectric
- Thermal-Electric
- Structural-Thermoelectric
- Thermal-Piezoelectric
- Structural-Diffusion
- Thermal-Diffusion
- Structural-Thermal-Diffusion

The element has twenty nodes with up to five degrees of freedom per node. Structural capabilities include elasticity, plasticity, viscoelasticity, viscoplasticity, creep, large strain, large deflection, stress stiffening effects, and prestress effects. Thermoelectric capabilities include Seebeck, Peltier, and Thomson effects, as well as Joule heating. In addition to thermal expansion, structural-thermal capabilities include the piezocaloric effect in dynamic analyses. The Coriolis effect is available for analyses with structural degrees of freedom. The thermoplastic effect is available for analyses with structural and thermal degrees of freedom. The diffusion expansion effect is available for analyses with structural and diffusion degrees of freedom. See SOLID226 in the *Mechanical APDL Theory Reference* for more details about this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 226.1: SOLID226 Geometry

SOLID226 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 226.1: SOLID226 Geometry (p. 1170). The element input data includes twenty nodes and structural, thermal, and electrical material properties. The type of units (MKS or user defined) is specified through the **EMUNIT** command. **EMUNIT** also determines the value of free-space permittivity EPZRO. The **EMUNIT** defaults are MKS units and EPZRO = 8.85e-12 Farads/meter.

KEYOPT(1) determines the element DOF set and the corresponding force labels and reaction solution. KEYOPT(1) is set equal to the sum of the field keys shown in Table 226.1: SOLID226 Field Keys (p. 1170). For example, KEYOPT(1) is set to 11 for a structural-thermal analysis (structural field key + thermal field key = 1 + 10). For a structural-thermal analysis, UX, UY, and TEMP are the DOF labels and force and heat flow are the reaction solution.

Table 226.1: SOLID226 Field Keys

Field	Field Key	DOF Label	Force Label	Reaction Solution
Structural	1	UX, UY, UZ	FX, FY, FZ	Force
Thermal	10	TEMP	HEAT	Heat Flow
Electric Conduction	100	VOLT	AMPS	Electric Current
Electrostatic	1000	VOLT	CHRG	Electric Charge
Diffusion	100000	CONC	RATE	Diffusion Flow Rate

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The coupled-field analysis KEYOPT(1) settings, DOF labels, force labels, reaction solutions, and analysis types are shown in the following table.

Table 226.2: SOLID226 Coupled-Field Analyses

Coupled-Field Analysis	KEYOPT(1)	DOF Label	Force Label	Reaction Solution	Analysis Type
Structural-Thermal [1], [2]	11	UX, UY, UZ, TEMP	FX, FY, FZ, HEAT	Force, Heat Flow	Static Full Harmonic Full Transient
Piezoresistive	101	UX, UY, UZ, VOLT	FX, FY, FZ, AMPS	Force, Electric Current	Static Full Transient
Electroelastic	1001 [3]	UX, UY, UZ, VOLT	FX, FY, FZ, CHRG	Force, Electric Charge (positive)	Static Full Transient
Piezoelectric	1001 [3]	UX, UY, UZ, VOLT	FX, FY, FZ, CHRG	Force, Electric Charge (negative)	Static Modal Full Harmonic Full Transient
Thermal-Electric	110	TEMP, VOLT	HEAT, AMPS	Heat Flow, Electric Current	Static Full Transient
Structural-Thermoelectric [1]	111	UX, UY, UZ, TEMP, VOLT	FX, FY, FZ, HEAT, AMPS	Force, Heat Flow, Electric Current	Static Full Transient
Thermal-Piezoelectric [1], [2]	1011	UX, UY, UZ, TEMP, VOLT	FX, FY, FZ, HEAT, CHRG	Force, Heat Flow, Electric Charge (negative)	Static Full Harmonic Full Transient

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Coupled-Field Analysis	KEY-OPT(1)	DOF Label	Force Label	Reaction Solution	Analysis Type
Structural-Diffusion [1]	100001	UX, UY, UZ, CONC	FX, FY, FZ, RATE	Force, Diffusion Flow Rate	Static Full Transient
Thermal-Diffusion	100010	TEMP, CONC	HEAT, RATE	Heat Flow, Diffusion Flow Rate	Static Full Transient
Structural-Thermal-Diffusion [1]	100011	UX, UY, UZ, TEMP, CONC	FX, FY, FZ, HEAT, RATE	Force, Heat Flow, Diffusion Flow Rate	Static Full Transient

1. For static and full transient analyses, KEYOPT(2) can specify a strong (matrix) or weak (load vector) structural-thermal coupling and/or structural-diffusion coupling.

2. For full harmonic analyses, strong structural-thermal coupling only applies.

3. The electrostatic-structural analysis available with KEYOPT(1) = 1001 defaults to an electroelastic analysis (electrostatic force coupling) unless a piezoelectric matrix is specified on **TB**,PIEZ.

As shown in the following table, material property requirements consist of those required for the individual fields (structural, thermal, electric conduction, electrostatic, or diffusion) and those required for field coupling. Individual material properties are defined via the **MP** and **MPDATA** commands. Nonlinear material models are defined via the **TB** command. (The nonlinear material models do not apply to piezoelectric analyses (**TB**,PIEZ) where KEYOPT(1) = 1001 or 1011).

Table 226.3: SOLID226 Material Properties and Material Models

ALL ANALYSES WITH A STRUCTURAL FIELD	
Structural	EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), GXY, GYZ, GXZ, DENS, ALPD, BETD, DMPR ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ, or THSX, THSY, THSZ), REFT --- Anisotropic elasticity, Bilinear isotropic hardening, Bilinear kinematic hardening, Cast iron, Chaboche nonlinear kinematic hardening, Creep, Extended Drucker-Prager, Elasticity, Gurson pressure-dependent plasticity, Hill anisotropy, Multilinear isotropic hardening, Multilinear kinematic

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	hardening, Voce isotropic hardening law, Plasticity, Prony series constants for viscoelastic materials, Rate-dependent plasticity (viscoplasticity), Material structural damping, Shift function for viscoelastic materials, Shape memory alloy, Uniaxial stress-strain relation
ADDITIONAL MATERIAL PROPERTIES FOR STRUCTURAL-THERMAL ANALYSES	
Thermal	KXX, KYY, KZZ, DENS, C, ENTH, HF
Coupling	ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ, or THSX, THSY, THSZ), REFT, QRATE
ADDITIONAL MATERIAL PROPERTIES FOR PIEZORESISTIVE ANALYSES	
Electric	RSVX, RSVY, RSVZ, PERX, PERY, PERZ
Coupling	Piezoresistivity
ADDITIONAL MATERIAL PROPERTIES FOR ELECTROELASTIC ANALYSES	
Electric	PERX, PERY, PERZ --- Anisotropic electric permittivity
ADDITIONAL MATERIAL PROPERTIES FOR PIEZOELECTRIC ANALYSES	
Electric	PERX, PERY, PERZ, LSST (and/or RSVX, RSVY, RSVZ) --- Anisotropic electric permittivity
Coupling	Piezoelectric matrix
ADDITIONAL MATERIAL PROPERTIES FOR STRUCTURAL-DIFFUSION ANALYSES	
Diffusion	DXX, DYY, DZZ, CSAT
Coupling	BETX, BETY, BETZ, CREF
MATERIAL PROPERTIES FOR THERMAL-ELECTRIC ANALYSES	
Thermal	KXX, KYY, KZZ, DENS, C, ENTH, HF
Electric	RSVX, RSVY, RSVZ, PERX, PERY, PERZ
Coupling	SBKX, SBKY, SBKZ
ADDITIONAL MATERIAL PROPERTIES FOR STRUCTURAL-THERMOELECTRIC ANALYSES	
Thermal	KXX, KYY, KZZ, DENS, C, ENTH, HF
Electric	RSVX, RSVY, RSVZ, PERX, PERY, PERZ
Coupling	ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ, or THSX, THSY, THSZ), SBKX, SBKY, PZRS, QRATE --- Piezoresistivity
ADDITIONAL MATERIAL PROPERTIES FOR THERMAL-PIEZOELECTRIC ANALYSES	
Thermal	KXX, KYY, KZZ, DENS, C, ENTH, HF
Electric	PERX, PERY, PERZ, LSST (and/or RSVX, RSVY, RSVZ) ---

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Anisotropic electric permittivity
Coupling	ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ, or THSX, THSY, THSZ), PIEZ, QRATE
MATERIAL PROPERTIES FOR THERMAL-DIFFUSION ANALYSES	
Thermal	KXX, KYY, KZZ, DENS, C, ENTH, HF
Diffusion	DXX, DYY, DZZ, CSAT
ADDITIONAL MATERIAL PROPERTIES FOR STRUCTURAL-THERMAL-DIFFUSION ANALYSES	
Thermal	KXX, KYY, KZZ, DENS, C, ENTH, HF
Diffusion	DXX, DYY, DZZ, CSAT
Coupling	BETX, BETY, BETZ, CREF

Various combinations of nodal loading are available for this element (depending upon the KEYOPT(1) value). Nodal loads are defined with the **D** and the **F** commands.

Element loads are described in Nodal Loading (p. 49). Loads may be input on the element faces indicated by the circled numbers in Figure 226.1: SOLID226 Geometry (p. 1170) using the **SF** and **SFE** commands. Positive pressures act into the element. Body loads may be input at the element's nodes or as a single element value using the **BF** and **BFE** commands.

SOLID226 surface and body loads are given in the following table. CHRGS and CHRGD are interpreted as negative surface charge density and negative volume charge density, respectively.

Table 226.4: SOLID226 Surface and Body Loads

Coupled-Field Analysis	KEYOPT(1)	Load Type	Load	Command Label
Structural-Thermal	11	Surface	Pressure	PRES
			Convection Heat Flux Radiation	CONV HFLUX RDSF
		Body	Force Density	FORC
			Heat Generation -- Nodes I, J, ..., A, B	HGEN
Piezoresistive	101	Surface	Pressure	PRES
		Body	Force Density	FORC
			Temperature -- Nodes I, J, ..., A, B	TEMP
Electroelastic and Piezoelectric	1001	Surface	Pressure Surface Charge Density	PRES CHRGS[1]
		Body	Force Density	FORC
			Temperature -- Nodes I, J, ..., A, B	TEMP
			Volume Charge Density -- Nodes I, J, ..., A, B	CHRGD[1]

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Coupled-Field Analysis	KEY-OPT(1)	Load Type	Load	Command Label
Thermal-Electric	110	Surface	Convection Heat Flux Radiation	CONV HFLUX RDSF
		Body	Heat Generation -- Nodes I, J, ..., A, B	HGEN
Structural-Thermo-electric	111	Surface	Pressure	PRES
			Convection Heat Flux Radiation	CONV HFLUX RDSF
		Body	Force Density	FORC
			Heat Generation -- Nodes I, J, ..., A, B	HGEN
Thermal-Piezoelectric	1011	Surface	Pressure Surface Charge Density	PRES CHRGS[1]
			Convection Heat Flux Radiation	CONV HFLUX RDSF
		Body	Force Density	FORC
			Heat Generation -- Nodes I, J, ..., A, B	HGEN
			Volume Charge Density -- Nodes I, J, ..., A, B	CHRGD[1]
Structural-Diffusion	100001	Surface	Pressure	PRES
			Diffusion Flux	DFLUX
		Body	Force Density	FORC
			Temperature -- Nodes I through P	TEMP
			Diffusing Substance Generation -- Nodes I through P	DGEN
Thermal-Diffusion	100010	Surface	Convection Heat Flux Radiation	CONV HFLUX RDSF
			Diffusion Flux	DFLUX
		Body	Heat Generation -- Nodes I through P	HGEN
			Diffusing Substance Generation -- Nodes I through P	DGEN
Structural-Thermal-Diffusion	100011	Surface	Pressure	PRES
			Convection	CONV

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Coupled-Field Analysis	KEYOPT(1)	Load Type	Load	Command Label
			Heat Flux Radiation	HFLUX RDSF
			Diffusion Flux	DFLUX
		Body	Force Density	FORC
			Heat Generation -- Nodes I through P	HGEN
			Diffusing Substance Generation -- Nodes I through P	DGEN

1. CHRGS and CHRGD are interpreted as negative surface charge density and negative volume charge density, respectively.

Automatic element technology selections are given in the following table.

Table 226.5: Automatic Element Technology Selection

Coupled-Field Analysis	ETCONTROL Command Suggestions/Resettings
Structural-Thermal (KEYOPT(1) = 11) Structural-Thermoelectric (KEYOPT(1) = 111)	KEYOPT(2) =1 for nonlinear inelastic materials
All analyses with a structural field	KEYOPT(6) =1 for linear elastic materials with Poisson's ratio >0.49 or nonlinear inelastic materials

A summary of the element input is given in "SOLID226 Input Summary" (p. 1176). A general description of element input is given in Element Input (p. 43).

SOLID226 Input Summary

Nodes

I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B

Degrees of Freedom

Set by KEYOPT(1). See Table 226.2: SOLID226 Coupled-Field Analyses (p. 1171).

Real Constants

None

Material Properties

See Table 226.3: SOLID226 Material Properties and Material Models (p. 1172).

Surface Loads

See Table 226.4: SOLID226 Surface and Body Loads (p. 1174).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Body Loads

See Table 226.4: SOLID226 Surface and Body Loads (p. 1174).

Special Features

Element technology autoselect
Large deflection
Large strain
Linear perturbation (piezoelectric analysis only [KEYOPT(1) = 1001])
Nonlinear stabilization
Stress stiffening

KEYOPT(1)

Element degrees of freedom. See Table 226.2: SOLID226 Coupled-Field Analyses (p. 1171).

KEYOPT(2)

Structural-thermal coupling method in structural-thermal, structural-thermoelectric, thermal-piezoelectric, or structural-thermal-diffusion analyses (KEYOPT(1) = 11, 111, 1011, or 100011) or structural-diffusion coupling method in structural-diffusion or structural-thermal-diffusion analyses (KEYOPT(1)=100001 or 100011).

0 --

Strong (matrix) coupling – produces an unsymmetric matrix. In a linear analysis, a coupled response is achieved after one iteration.

1 --

Weak (load vector) coupling – produces a symmetric matrix and requires at least two iterations to achieve a coupled response.

KEYOPT(4)

Electrostatic force in electroelastic analysis (KEYOPT(1) = 1001):

0 --

Applied to every element node.

1 --

Applied to the air-structure interface or to element nodes that have constrained structural degrees of freedom.

2 --

Not applied.

For more information, see Electroelastic Analysis in the *Coupled-Field Analysis Guide*.

KEYOPT(6)

Integration method (applicable to the brick-shaped elements with structural DOFs).

0 --

Full integration - uses 14 integrations points. This method can cause volumetric locking in the models with nearly incompressible materials. It is primary employed for purely linear analyses.

1 --

Uniform reduced integration - uses a 2 x 2 x 2 integration scheme. This method helps prevent volumetric mesh locking in the models with nearly incompressible materials. It is recommended for

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

analyses with structural nonlinearities. To avoid the propagation of hourglass mode associated with the reduced integration, the model must have at least two layers of elements in each direction.

KEYOPT(9)

Thermoelastic damping (piezocaloric effect) in structural-thermal, structural-thermoelastic, or thermal-piezoelectric analyses (KEYOPT(1) = 11, 111, or 1011). Applicable to harmonic and transient analyses only.

0 --

Active

1 --

Suppressed (required for frictional heating analyses)

KEYOPT(10)

Specific heat matrix in structural-thermal, thermal-electric, structural-thermoelectric, thermal-piezoelectric, thermal-diffusion, structural-thermal-diffusion analyses (KEYOPT(1) = 11, 110, 111, 1011, 100001, or 100011), and damping matrix in structural-diffusion, thermal-diffusion, and structural-thermal-diffusion analyses (KEYOPT(1) = 100001, 100010, and 100011).

0 --

Consistent

1 --

Diagonalized

2 --

Diagonalized. Temperature-dependent specific heat or enthalpy is evaluated at the element centroid.

SOLID226 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in Table 226.6: SOLID226 Element Output Definitions (p. 1178).

The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 226.6: SOLID226 Element Output Definitions

Name	Definition	O	R
ALL ANALYSES			

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
EL	Element Number	-	Y
NODES	Nodes - I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B	-	Y
MAT	Material number	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	-	2
ALL ANALYSES WITH A STRUCTURAL FIELD			
S:X, Y, Z, XY, YZ, XZ	Stresses (SZ = 0.0 for plane stress elements)	-	1
S:1, 2, 3	Principal stresses	-	1
S:EQV	Equivalent stress	-	1
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	-	1
EPEL:EQV	Equivalent elastic strain [3]	-	1
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	-	1
EPTH:EQV	Equivalent thermal strain [3]	-	1
EPPL:X, Y, Z, XY, YZ, XZ	Plastic strains	-	1
EPPL:EQV	Equivalent plastic strain [3]	-	1
EPCR:X, Y, Z, XY, YZ, XZ	Creep strains	-	1
EPCR:EQV	Equivalent creep strain [3]	-	1
EPTO:X, Y, Z, XY, YZ, XZ	Total mechanical strains (EPEL + EPPL + EPCR)	-	-
EPTO:EQV	Total equivalent mechanical strain (EPEL + EPPL + EPCR)	-	-
ADDITIONAL OUTPUT FOR STRUCTURAL-THERMAL ANALYSES (KEYOPT(1) = 11)			
TG:X, Y, Z, SUM	Thermal gradient components and vector magnitude	-	1
TF:X, Y, Z, SUM	Thermal flux components and vector magnitude	-	1
UT	Total strain energy [7]	-	1
PHEAT	Plastic heat generation rate per unit volume	-	1
ADDITIONAL OUTPUT FOR PIEZORESISTIVE ANALYSES (KEYOPT(1) = 101)			
TEMP	Input temperatures	-	Y
EF:X, Y, Z, SUM	Electric field components (X, Y, Z) and vector magnitude	-	1
JC:X, Y, Z, SUM	Conduction current density components (X, Y, Z) and vector magnitude	-	1
JS:X, Y, Z, SUM	Current density components (in the global Cartesian coordinate system) and vector magnitude [4]	-	1
JHEAT	Joule heat generation per unit volume [5]	-	1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
ADDITIONAL OUTPUT FOR ELECTROELASTIC ANALYSES (KEYOPT(1) = 1001)			
TEMP	Input temperatures	-	Y
EF:X, Y, Z, SUM	Electric field components (X, Y, Z) and vector magnitude	-	1
D:X, Y, Z, SUM	Electric flux density components (X, Y, Z) and vector magnitude	-	1
FMAG:X, Y, Z, SUM	Electrostatic force components (X, Y, Z) and vector magnitude	-	1
UE, UD	Stored elastic and dielectric energies	-	1
ADDITIONAL OUTPUT FOR PIEZOELECTRIC ANALYSES (KEYOPT(1) = 1001)			
TEMP	Input temperatures	-	Y
EF:X, Y, Z, SUM	Electric field components (X, Y, Z) and vector magnitude	-	1
D:X, Y, Z, SUM	Electric flux density components (X, Y, Z) and vector magnitude	-	1
JHEAT	Joule heat generation per unit volume [5], [6]	-	1
UE, UD	Stored elastic and dielectric energies	-	1
ADDITIONAL OUTPUT FOR STRUCTURAL-DIFFUSION ANALYSES (KEYOPT(1)=100001)			
TEMP	Input temperatures	-	Y
EPDI:X, Y, Z, XY, YZ, XZ	Diffusion strains	-	1
CG:X, Y, Z, SUM	Concentration gradient components and vector magnitude	-	1
DF:X, Y, Z, SUM	Diffusion flux components and vector magnitude	-	1
CONC	Element concentration [8]	-	1
THERMAL-ELECTRIC ANALYSES (KEYOPT(1) = 110)			
TG:X, Y, Z, SUM	Thermal gradient components and vector magnitude	-	1
TF:X, Y, Z, SUM	Thermal flux components and vector magnitude	-	1
EF:X, Y, Z, SUM	Electric field components and vector magnitude	-	1
JC:X, Y, Z, SUM	Conduction current density components and vector magnitude	-	1
JS:X, Y, Z, SUM	Current density components (in the global Cartesian coordinate system) and vector magnitude [4]	-	1
JHEAT	Joule heat generation per unit volume [5], [6]	-	1
ADDITIONAL OUTPUT FOR STRUCTURAL-THERMOELECTRIC ANALYSES (KEYOPT(1) = 111)			
TG:X, Y, Z, SUM	Thermal gradient components and vector magnitude	-	1
TF:X, Y, Z, SUM	Thermal flux components and vector magnitude	-	1
EF:X, Y, Z, SUM	Electric field components and vector magnitude	-	1
JC:X, Y, Z, SUM	Conduction current density components and vector magnitude	-	1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
JS:X, Y, Z, SUM	Current density components components (in the global Cartesian coordinate system) and vector magnitude [4]	-	1
JHEAT	Joule heat generation per unit volume [5], [6]	-	1
UT	Total strain energy [7]	-	1
PHEAT	Plastic heat generation rate per unit volume	-	1
ADDITIONAL OUTPUT FOR THERMAL-PIEZOELECTRIC ANALYSES (KEYOPT(1) = 1011)			
TG:X, Y, Z, SUM	Thermal gradient components and vector magnitude	-	1
TF:X, Y, Z, SUM	Thermal flux components and vector magnitude	-	1
EF:X, Y, Z, SUM	Electric field components and vector magnitude	-	1
D:X, Y, Z, SUM	Electric flux density components and vector magnitude	-	1
JHEAT	Joule heat generation per unit volume [5], [6]	-	1
UE, UD	Stored elastic and dielectric energies	-	1
UT	Total strain energy [7]	-	1
PHEAT	Plastic heat generation rate per unit volume	-	1
THERMAL-DIFFUSION ANALYSES (KEYOPT(1) = 100010)			
TG:X, Y, Z, SUM	Thermal gradient components and vector magnitude	-	1
TF:X, Y, Z, SUM	Thermal flux components and vector magnitude	-	1
CG:X, Y, Z, SUM	Concentration gradient components and vector magnitude	-	1
DF:X, Y, Z, SUM	Diffusion flux components and vector magnitude	-	1
CONC	Element concentration [8]	-	1
ADDITIONAL OUTPUT FOR STRUCTURAL-THERMAL-DIFFUSION ANALYSES (KEYOPT(1) = 100011)			
EPDI:X, Y, Z, XY, YZ, XZ	Diffusion strains	-	1
TG:X, Y, Z, SUM	Thermal gradient components and vector magnitude	-	1
TF:X, Y, Z, SUM	Thermal flux components and vector magnitude	-	1
CG:X, Y, Z, SUM	Concentration gradient components and vector magnitude	-	1
DF:X, Y, Z, SUM	Diffusion flux components and vector magnitude	-	1
CONC	Element concentration [8]	-	1

1. Solution values are output only if calculated (based on input values).

2. Available only at centroid as a ***GET** item.

3. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.

4. JS represents the sum of element conduction and displacement current densities.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Calculated Joule heat generation rate per unit volume (JHEAT) may be made available for a subsequent thermal analysis with companion thermal elements.

6. For a time-harmonic analysis, Joule losses (JHEAT) are time-averaged. These values are stored in both the real and imaginary data sets. For more information, see Quasistatic Electric Analysis in the *Mechanical APDL Theory Reference*.

7. For a time-harmonic analysis, total strain energy (UT) is time-averaged. These values are stored in both the real and imaginary data sets. For more information, see Thermoelasticity in the *Mechanical APDL Theory Reference*.

8. With the normalized concentration approach, CONC is the actual concentration obtained by multiplying the saturated concentration (**MP**,CSAT) and the normalized concentration evaluated at the element centroid. For more information, see Normalized Concentration Approach in the *Mechanical APDL Theory Reference*.

Table 226.6: SOLID226 Element Output Definitions (p. 1178) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) of the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) of this manual for more information. The following notation is used in Table 226.7: SOLID226 Item and Sequence Numbers (p. 1182):

Name
output quantity as defined in the Table 226.6: SOLID226 Element Output Definitions (p. 1178)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 226.7: SOLID226 Item and Sequence Numbers

Output Quantity Name	ETABLE Command Input	
	Item	E
CONC	SMISC	1
UE	NMISC	1
UD	NMISC	2
UT	NMISC	4
PHEAT	NMISC	5

SOLID226 Assumptions and Restrictions

- In a piezoelectric analysis, electric charge loading is interpreted as negative electric charge or negative charge density.
- CSAT cannot be temperature-dependent.
- The optimized nonlinear solution defaults (**SOLCONTROL**) are not available in coupled-field analyses.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- An edge with a removed midside node implies that the degrees-of-freedom varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.
- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, the elements must have the same reaction solution for the VOLT DOF. Elements that have an electric charge reaction solution must all have the same electric charge reaction sign. For more information, see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*.
- The model should have at least two layers of elements in each direction when uniform reduced integration (KEYOPT(6) = 1) is used.
- This element does not support fully incompressible materials.
- Graphical Solution Tracking (**/GST**) is not supported with coupled-diffusion analyses (KEYOPT(1)=100001, 100010, and 100011).

SOLID226 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID227

3-D 10-Node Coupled-Field Solid

MP ME <> <> <> <> <> <> PP EME MFS
Product Restrictions

SOLID227 Element Description

SOLID227 has the following capabilities:

- Structural-Thermal
- Piezoresistive
- Electroelastic
- Piezoelectric
- Thermal-Electric
- Structural-Thermoelectric
- Thermal-Piezoelectric
- Structural-Diffusion
- Thermal-Diffusion
- Structural-Thermal-Diffusion

The element has ten nodes with up to five degrees of freedom per node. Structural capabilities include elasticity, plasticity, viscoelasticity, viscoplasticity, creep, large strain, large deflection, stress stiffening effects, and prestress effects. Thermoelectric capabilities include Seebeck, Peltier, and Thomson effects, as well as Joule heating. In addition to thermal expansion, structural-thermal capabilities include the piezocaloric effect in dynamic analyses. The Coriolis effect is available for analyses with structural degrees of freedom. The thermoplastic effect is available for analyses with structural and thermal degrees of freedom. The diffusion expansion effect is available for analyses with structural and diffusion degrees of freedom. See SOLID227 in the *Mechanical APDL Theory Reference* for more details about this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 227.1: SOLID227 Geometry

SOLID227 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 227.1: SOLID227 Geometry (p. 1186). The element input data includes ten nodes and structural, thermal, and electrical material properties. The type of units (MKS or user defined) is specified through the **EMUNIT** command. **EMUNIT** also determines the value of free-space permittivity EPZRO. The **EMUNIT** defaults are MKS units and EPZRO = 8.85e-12 Farads/meter.

KEYOPT(1) determines the element DOF set and the corresponding force labels and reaction solution. KEYOPT(1) is set equal to the sum of the field keys shown in Table 227.1: SOLID227 Field Keys (p. 1186). For example, KEYOPT(1) is set to 11 for a structural-thermal analysis (structural field key + thermal field key = 1 + 10). For a structural-thermal analysis, UX, UY, and TEMP are the DOF labels and force and heat flow are the reaction solution.

Table 227.1: SOLID227 Field Keys

Field	Field Key	DOF Label	Force Label	Reaction Solution
Structural	1	UX, UY, UZ	FX, FY, FZ	Force
Thermal	10	TEMP	HEAT	Heat Flow
Electric Conduction	100	VOLT	AMPS	Electric Current
Electrostatic	1000	VOLT	CHRG	Electric Charge
Diffusion	100000	CONC	RATE	Diffusion Flow Rate

The coupled-field analysis KEYOPT(1) settings, DOF labels, force labels, reaction solutions, and analysis types are shown in the following table.

Table 227.2: SOLID227 Coupled-Field Analyses

Coupled-Field Analysis	KEYOPT(1)	DOF Label	Force Label	Reaction Solution	Analyses Type
Structural-Thermal [1], [2]	11	UX, UY, UZ,	FX, FY, FZ,	Force,	Static

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Coupled-Field Analysis	KEY-OPT(1)	DOF Label	Force Label	Reaction Solution	Analyses Type
		TEMP	HEAT	Heat Flow	Full Harmonic Full Transient
Piezoresistive	101	UX, UY, UZ, VOLT	FX, FY, FZ, AMPS	Force, Electric Current	Static Full Transient
Electroelastic	1001 [3]	UX, UY, UZ, VOLT	FX, FY, FZ, CHRG	Force, Electric Charge (positive)	Static Full Transient
Piezoelectric	1001 [3]	UX, UY, UZ, VOLT	FX, FY, FZ, CHRG	Force, Electric Charge (negative)	Static Modal Full Harmonic Full Transient
Thermal-Electric	110	TEMP, VOLT	HEAT, AMPS	Heat Flow, Electric Current	Static Full Transient
Structural-Thermoelectric [1]	111	UX, UY, UZ, TEMP, VOLT	FX, FY, FZ, HEAT, AMPS	Force, Heat Flow, Electric Current	Static Full Transient
Thermal-Piezoelectric [1], [2]	1011	UX, UY, UZ, TEMP, VOLT	FX, FY, FZ, HEAT, CHRG	Force, Heat Flow, Electric	Static Full Harmonic

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Coupled-Field Analysis	KEYOPT(1)	DOF Label	Force Label	Reaction Solution	Analyses Type
				Charge (negative)	Full Transient
Structural-Diffusion [1]	100001	UX, UY, UZ, CONC	FX, FY, FZ, RATE	Force, Diffusion Flow Rate	Static Full Transient
Thermal-Diffusion	100010	TEMP, CONC	HEAT, RATE	Heat Flow, Diffusion Flow Rate	Static Full Transient
Structural-Thermal-Diffusion [1]	100011	UX, UY, UZ, TEMP, CONC	FX, FY, FZ, HEAT, RATE	Force, Heat Flow, Diffusion Flow Rate	Static Full Transient

1. For static and full transient analyses, KEYOPT(2) can specify a strong (matrix) or weak (load vector) structural-thermal coupling and/or structural-diffusion coupling.

2. For full harmonic analyses, strong structural-thermal coupling only applies.

3. The electrostatic-structural analysis available with KEYOPT(1) = 1001 defaults to an electroelastic analysis (electrostatic force coupling) unless a piezoelectric matrix is specified on **TB**,PIEZ.

As shown in the following table, material property requirements consist of those required for the individual fields (structural, thermal, electric conduction, electrostatic, or diffusion) and those required for field coupling. Individual material properties are defined via the **MP** and **MPDATA** commands. Nonlinear material models are defined via the **TB** command. (The nonlinear material models do not apply to piezoelectric analyses (**TB**,PIEZ) where KEYOPT(1) = 1001 or 1011).

Table 227.3: SOLID227 Material Properties and Material Models

ALL ANALYSES WITH A STRUCTURAL FIELD	
Structural	EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), GXY, GYZ, GXZ, DENS, ALPD, BETD, DMPR ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ, or THSX, THSY, THSZ), REFT ---

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Anisotropic elasticity, Bilinear isotropic hardening, Bilinear kinematic hardening, Cast iron, Chaboche nonlinear kinematic hardening, Creep, Extended Drucker-Prager, Elasticity, Gurson pressure-dependent plasticity, Hill anisotropy, Multilinear isotropic hardening, Multilinear kinematic hardening, Voce isotropic hardening law, Plasticity, Prony series constants for viscoelastic materials, Rate-dependent plasticity (viscoplasticity), Material structural damping, Shift function for viscoelastic materials, Shape memory alloy, Uniaxial stress-strain relation
ADDITIONAL MATERIAL PROPERTIES FOR STRUCTURAL-THERMAL ANALYSES	
Thermal	KXX, KYY, KZZ, DENS, C, ENTH, HF
Coupling	ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ, or THSX, THSY, THSZ), REFT, QRATE
ADDITIONAL MATERIAL PROPERTIES FOR PIEZORESISTIVE ANALYSES	
Electric	RSVX, RSVY, RSVZ, PERX, PERY, PERZ
Coupling	Piezoresistivity
ADDITIONAL MATERIAL PROPERTIES FOR ELECTROELASTIC ANALYSES	
Electric	PERX, PERY, PERZ --- Anisotropic electric permittivity
ADDITIONAL MATERIAL PROPERTIES FOR PIEZOELECTRIC ANALYSES	
Electric	PERX, PERY, PERZ, LSST (and/or RSVX, RSVY, RSVZ) --- Anisotropic electric permittivity
Coupling	Piezoelectric matrix
ADDITIONAL MATERIAL PROPERTIES FOR STRUCTURAL-DIFFUSION ANALYSES	
Diffusion	DXX, DYY, DZZ, CSAT
Coupling	BETX, BETY, BETZ, CREF
MATERIAL PROPERTIES FOR THERMAL-ELECTRIC ANALYSES	
Thermal	KXX, KYY, KZZ, DENS, C, ENTH, HF
Electric	RSVX, RSVY, RSVZ, PERX, PERY, PERZ
Coupling	SBKX, SBKY, SBKZ
ADDITIONAL MATERIAL PROPERTIES FOR STRUCTURAL-THERMOELECTRIC ANALYSES	
Thermal	KXX, KYY, KZZ, DENS, C, ENTH, HF
Electric	RSVX, RSVY, RSVZ, PERX, PERY, PERZ
Coupling	ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ, or THSX, THSY, THSZ), SBKX, SBKY, PZRS, QRATE --- Piezoresistivity
ADDITIONAL MATERIAL PROPERTIES FOR THERMAL-PIEZOELECTRIC ANALYSES	
Thermal	KXX, KYY, KZZ, DENS, C, ENTH, HF

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Electric	PERX, PERY, PERZ, LSST (and/or RSVX, RSVY, RSVZ) --- Anisotropic electric permittivity
Coupling	ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ, or THSX, THSY, THSZ), PIEZ, QRATE
MATERIAL PROPERTIES FOR THERMAL-DIFFUSION ANALYSES	
Thermal	KXX, KYY, KZZ, DENS, C, ENTH, HF
Diffusion	DXX, DYY, DZZ, CSAT
ADDITIONAL MATERIAL PROPERTIES FOR STRUCTURAL-THERMAL-DIFFUSION ANALYSES	
Thermal	KXX, KYY, KZZ, DENS, C, ENTH, HF
Diffusion	DXX, DYY, DZZ, CSAT
Coupling	BETX, BETY, BETZ, CREF

Various combinations of nodal loading are available for this element (depending upon the KEYOPT(1) value). Nodal loads are defined with the **D** and the **F** commands.

Element loads are described in Nodal Loading (p. 49). Loads may be input on the element faces indicated by the circled numbers in Figure 227.1: SOLID227 Geometry (p. 1186) using the **SF** and **SFE** commands. Positive pressures act into the element. Body loads may be input at the element's nodes or as a single element value using the **BF** and **BFE** commands.

SOLID227 surface and body loads are given in the following table.

Table 227.4: SOLID227 Surface and Body Loads

Coupled-Field Analysis	KEYOPT(1)	Load Type	Load	Command Label
Structural-Thermal	11	Surface	Pressure	PRES
			Convection Heat Flux Radiation	CONV HFLUX RDSF
		Body	Force Density	FORC
			Heat Generation -- Nodes I through R	HGEN
Piezoresistive	101	Surface	Pressure	PRES
		Body	Force Density	FORC
			Temperatures -- Nodes I through R	TEMP
Electroelastic and Piezoelectric	1001	Surface	Pressure Surface Charge Density	PRES CHRGS [1]
		Body	Force Density	FORC
			Temperatures -- Nodes I through R	TEMP

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Coupled-Field Analysis	KEY-OPT(1)	Load Type	Load	Command Label
			Volume Charge Density -- Nodes I through R	CHRGD [1]
Thermal-Electric	110	Surface	Convection Heat Flux Radiation	CONV HFLUX RDSF
		Body	Heat Generation -- Nodes I through R	HGEN
Structural-Thermoelectric	111	Surface	Pressure	PRES
			Convection Heat Flux Radiation	CONV HFLUX RDSF
		Body	Force Density	FORC
			Heat Generation -- Nodes I through R	HGEN
Thermal-Piezoelectric	1011	Surface	Pressure Surface Charge Density	PRES CHRGS [1]
			Convection Heat Flux Radiation	CONV HFLUX RDSF
		Body	Force Density	FORC
			Heat Generation -- Nodes I through R	HGEN
			Volume Charge Density -- Nodes I through R	CHRGD [1]
Structural-Diffusion	100001	Surface	Pressure	PRES
			Diffusion Flux	DFLUX
		Body	Force Density	FORC
			Temperature -- Nodes I through P	TEMP
			Diffusing Substance Generation -- Nodes I through P	DGEN
Thermal-Diffusion	100010	Surface	Convection Heat Flux Radiation	CONV HFLUX RDSF
			Diffusion Flux	DFLUX

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Coupled-Field Analysis	KEYOPT(1)	Load Type	Load	Command Label
		Body	Heat Generation - - Nodes I through P	HGEN
			Diffusing Substance Generation -- Nodes I through P	DGEN
Structural-Thermal-Diffusion	100011	Surface	Pressure	PRES
			Convection Heat Flux Radiation	CONV HFLUX RDSF
			Diffusion Flux	DFLUX
		Body	Force Density	FORC
			Heat Generation - - Nodes I through P	HGEN
			Diffusing Substance Generation -- Nodes I through P	DGEN

1. CHRGS and CHRGD are interpreted as negative surface charge density and negative volume charge density, respectively.

Automatic element technology selections are given in the following table.

Table 227.5: Automatic Element Technology Selection

Coupled-Field Analysis	ETCONTROL Command Suggestions/Resettings
Structural-Thermal (KEYOPT(1) = 11) Structural-Thermoelectric (KEYOPT(1) = 111)	KEYOPT(2) =1 for nonlinear inelastic materials

A summary of the element input is given in "SOLID227 Input Summary" (p. 1192). A general description of element input is given in Element Input (p. 43).

SOLID227 Input Summary

Nodes

I, J, K, L, M, N, O, P, Q, R

Degrees of Freedom

Set by KEYOPT(1). See Table 227.2: SOLID227 Coupled-Field Analyses (p. 1186).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Real Constants

None

Material Properties

See Table 227.3: SOLID227 Material Properties and Material Models (p. 1188).

Surface Loads

See Table 227.4: SOLID227 Surface and Body Loads (p. 1190).

Body Loads

See Table 227.4: SOLID227 Surface and Body Loads (p. 1190).

Special Features

Element technology autoselect
Large deflection
Large strain
Linear perturbation (piezoelectric analysis only [KEYOPT(1) = 1001])
Nonlinear stabilization
Stress stiffening

KEYOPT(1)

Element degrees of freedom. See Table 227.2: SOLID227 Coupled-Field Analyses (p. 1186).

KEYOPT(2)

Structural-thermal coupling method in structural-thermal, structural-thermoelectric, thermal-piezoelectric, or structural-thermal-diffusion analyses (KEYOPT(1) = 11, 111, 1011, or 100011) or structural-diffusion coupling method in structural-diffusion or structural-thermal-diffusion analyses (KEYOPT(1)=100001 or 100011).

0 --

Strong (matrix) coupling – produces an unsymmetric matrix. In a linear analysis, a coupled response is achieved after one iteration.

1 --

Weak (load vector) coupling – produces a symmetric matrix and requires at least two iterations to achieve a coupled response.

KEYOPT(4)

Electrostatic force in electroelastic analysis (KEYOPT(1) = 1001):

0 --

Applied to every element node.

1 --

Applied to the air-structure interface or to element nodes that have constrained structural degrees of freedom.

2 --

Not applied.

For more information, see Electroelastic Analysis in the *Coupled-Field Analysis Guide*.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(9)

Thermoelastic damping (piezocaloric effect) in structural-thermal, structural-thermoelastic, or thermal-piezoelectric analyses (KEYOPT(1) = 11, 111, or 1011). Applicable to harmonic and transient analyses only.

0 --

Active

1 --

Suppressed (required for frictional heating analyses)

KEYOPT(10)

Specific heat matrix in structural-thermal, thermal-electric, structural-thermoelectric, thermal-piezoelectric, thermal-diffusion, structural-thermal-diffusion analyses (KEYOPT(1) = 11, 110, 111, 1011, 100001, or 100011), and damping matrix in structural-diffusion, thermal-diffusion, and structural-thermal-diffusion analyses (KEYOPT(1) = 100001, 100010, and 100011).

0 --

Consistent

1 --

Diagonalized

2 --

Diagonalized. Temperature-dependent specific heat or enthalpy is evaluated at the element centroid.

SOLID227 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in Table 227.6: SOLID227 Element Output Definitions (p. 1194).

The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 227.6: SOLID227 Element Output Definitions

Name	Definition	O	R
ALL ANALYSES			
EL	Element Number	-	Y
NODES	Nodes - I, J, K, L, M, N, O, P, Q, R	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
MAT	Material number	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	-	2
ALL ANALYSES WITH A STRUCTURAL FIELD			
S:X, Y, Z, XY, YZ, XZ	Stresses (SZ = 0.0 for plane stress elements)	-	1
S:1, 2, 3	Principal stresses	-	1
S:EQV	Equivalent stress	-	1
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	-	1
EPEL:EQV	Equivalent elastic strain [3]	-	1
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	-	1
EPTH:EQV	Equivalent thermal strain [3]	-	1
EPPL:X, Y, Z, XY, YZ, XZ	Plastic strains	-	1
EPPL:EQV	Equivalent plastic strain [3]	-	1
EPCR:X, Y, Z, XY, YZ, XZ	Creep strains	-	1
EPCR:EQV	Equivalent creep strain [3]	-	1
EPTO:X, Y, Z, XY, YZ, XZ	Total mechanical strains (EPEL + EPPL + EPCR)	-	-
EPTO:EQV	Total equivalent mechanical strain (EPEL + EPPL + EPCR)	-	-
ADDITIONAL OUTPUT FOR STRUCTURAL-THERMAL ANALYSES (KEYOPT(1) = 11)			
TG:X, Y, Z, SUM	Thermal gradient components and vector magnitude	-	1
TF:X, Y, Z, SUM	Thermal flux components and vector magnitude	-	1
UT	Total strain energy [7]	-	1
PHEAT	Plastic heat generation rate per unit volume	-	1
ADDITIONAL OUTPUT FOR PIEZORESISTIVE ANALYSES (KEYOPT(1) = 101)			
TEMP	Input temperatures	-	Y
EF:X, Y, Z, SUM	Electric field components (X, Y, Z) and vector magnitude	-	1
JC:X, Y, Z, SUM	Conduction current density components (X, Y, Z) and vector magnitude	-	1
JS:X, Y, Z, SUM	Current density components (in the global Cartesian coordinate system) and vector magnitude [4]	-	1
JHEAT	Joule heat generation per unit volume [5]	-	1
ADDITIONAL OUTPUT FOR ELECTROELASTIC ANALYSES (KEYOPT(1) = 1001)			
TEMP	Input temperatures	-	Y
EF:X, Y, Z, SUM	Electric field components (X, Y, Z) and vector magnitude	-	1
D:X, Y, Z, SUM	Electric flux density components (X, Y, Z) and vector magnitude	-	1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
FMAG:X, Y, Z, SUM	Electrostatic force components (X, Y, Z) and vector magnitude	-	1
UE, UD	Stored elastic and dielectric energies	-	1
ADDITIONAL OUTPUT FOR PIEZOELECTRIC ANALYSES (KEYOPT(1) = 1001)			
TEMP	Input temperatures	-	Y
EF:X, Y, Z, SUM	Electric field components (X, Y, Z) and vector magnitude	-	1
D:X, Y, Z, SUM	Electric flux density components (X, Y, Z) and vector magnitude	-	1
JHEAT	Joule heat generation per unit volume [5], [6]	-	1
UE, UD	Stored elastic and dielectric energies	-	1
ADDITIONAL OUTPUT FOR STRUCTURAL-DIFFUSION ANALYSES (KEYOPT(1)=100001)			
TEMP	Input temperatures	-	Y
EPDI:X, Y, Z, XY, YZ, XZ	Diffusion strains	-	1
CG:X, Y, Z, SUM	Concentration gradient components and vector magnitude	-	1
DF:X, Y, Z, SUM	Diffusion flux components and vector magnitude	-	1
CONC	Element concentration [8]	-	1
THERMAL-ELECTRIC ANALYSES (KEYOPT(1) = 110)			
TG:X, Y, Z, SUM	Thermal gradient components and vector magnitude	-	1
TF:X, Y, Z, SUM	Thermal flux components and vector magnitude	-	1
EF:X, Y, Z, SUM	Electric field components and vector magnitude	-	1
JC:X, Y, Z, SUM	Conduction current density components and vector magnitude	-	1
JS:X, Y, Z, SUM	Current density components (in the global Cartesian coordinate system) and vector magnitude [4]	-	1
JHEAT	Joule heat generation per unit volume [5], [6]	-	1
ADDITIONAL OUTPUT FOR STRUCTURAL-THERMOELECTRIC ANALYSES (KEYOPT(1) = 111)			
TG:X, Y, Z, SUM	Thermal gradient components and vector magnitude	-	1
TF:X, Y, Z, SUM	Thermal flux components and vector magnitude	-	1
EF:X, Y, Z, SUM	Electric field components and vector magnitude	-	1
JC:X, Y, Z, SUM	Conduction current density components and vector magnitude	-	1
JS:X, Y, Z, SUM	Current density components (in the global Cartesian coordinate system) and vector magnitude [4]	-	1
JHEAT	Joule heat generation per unit volume [5], [6]	-	1
UT	Total strain energy [7]	-	1
PHEAT	Plastic heat generation rate per unit volume	-	1
ADDITIONAL OUTPUT FOR THERMAL-PIEZOELECTRIC ANALYSES (KEYOPT(1) = 1011)			
TG:X, Y, Z, SUM	Thermal gradient components and vector magnitude	-	1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
TF:X, Y, Z, SUM	Thermal flux components and vector magnitude	-	1
EF:X, Y, Z, SUM	Electric field components and vector magnitude	-	1
D:X, Y, Z, SUM	Electric flux density components and vector magnitude	-	1
JHEAT	Joule heat generation per unit volume [5], [6]	-	1
UE, UD	Stored elastic and dielectric energies	-	1
UT	Total strain energy [7]	-	1
PHEAT	Plastic heat generation rate per unit volume	-	1
THERMAL-DIFFUSION ANALYSES (KEYOPT(1) = 100010)			
TG:X, Y, Z, SUM	Thermal gradient components and vector magnitude	-	1
TF:X, Y, Z, SUM	Thermal flux components and vector magnitude	-	1
CG:X, Y, Z, SUM	Concentration gradient components and vector magnitude	-	1
DF:X, Y, Z, SUM	Diffusion flux components and vector magnitude	-	1
CONC	Element concentration [8]	-	1
ADDITIONAL OUTPUT FOR STRUCTURAL-THERMAL-DIFFUSION ANALYSES (KEYOPT(1) = 100011)			
EPDI:X, Y, Z, XY, YZ, XZ	Diffusion strains	-	1
TG:X, Y, Z, SUM	Thermal gradient components and vector magnitude	-	1
TF:X, Y, Z, SUM	Thermal flux components and vector magnitude	-	1
CG:X, Y, Z, SUM	Concentration gradient components and vector magnitude	-	1
DF:X, Y, Z, SUM	Diffusion flux components and vector magnitude	-	1
CONC	Element concentration [8]	-	1

1. Solution values are output only if calculated (based on input values).

2. Available only at centroid as a ***GET** item.

3. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.

4. JS represents the sum of element conduction and displacement current densities.

5. Calculated Joule heat generation rate per unit volume (JHEAT) may be made available for a subsequent thermal analysis with companion thermal elements.

6. For a time-harmonic analysis, Joule losses (JHEAT) are time-averaged. These values are stored in both the real and imaginary data sets. For more information, see Quasistatic Electric Analysis in the *Mechanical APDL Theory Reference*.

7. For a time-harmonic analysis, total strain energy (UT) is time-averaged. These values are stored in both the real and imaginary data sets. For more information, see Thermoelasticity in the *Mechanical APDL Theory Reference*.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. With the normalized concentration approach, CONC is the actual concentration obtained by multiplying the saturated concentration (**MP**,CSAT) and the normalized concentration evaluated at the element centroid. For more information, see Normalized Concentration Approach in the *Mechanical APDL Theory Reference*.

Table 227.6: SOLID227 Element Output Definitions (p. 1194) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) of the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) of this manual for more information. The following notation is used in Table 227.7: SOLID227 Item and Sequence Numbers (p. 1198):

Name
output quantity as defined in the Table 227.6: SOLID227 Element Output Definitions (p. 1194)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 227.7: SOLID227 Item and Sequence Numbers

Output Quantity Name	ETABLE Command Input	
	Item	E
CONC	SMISC	1
UE	NMISC	1
UD	NMISC	2
UT	NMISC	4
PHEAT	NMISC	5

SOLID227 Assumptions and Restrictions

- In a piezoelectric analysis, electric charge loading is interpreted as negative electric charge or negative charge density.
- CSAT cannot be temperature-dependent.
- The optimized nonlinear solution defaults (**SOLCONTROL**) are not available in coupled-field analyses.
- An edge with a removed midside node implies that the degrees-of-freedom varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.
- This element may not be compatible with other elements with the VOLT degree of freedom. To be compatible, the elements must have the same through reaction solution for the VOLT DOF. Elements that have an electric charge reaction solution must all have the same electric charge reaction sign. For more information, see Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*.
- This element does not support fully incompressible materials.
- Graphical Solution Tracking (**/GST**) is not supported with coupled-diffusion analyses (KEYOPT(1)=100001, 100010, and 100011).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID227 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

PLANE230

2-D 8-Node Electric Solid

MP <> <> <> <> <> EM <> PP EME <>
Product Restrictions

PLANE230 Element Description

PLANE230 is a 2-D, 8-node, current-based electric element. The element has one degree of freedom, voltage, at each node. The 8-node elements have compatible voltage shapes and are well suited to model curved boundaries.

This element is based on the electric scalar potential formulation and it is applicable to the following low frequency electric field analyses: steady-state electric conduction, time-harmonic quasistatic and transient quasistatic. See PLANE230 - 2-D 8-Node Electric Solid in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 230.1: PLANE230 Geometry

PLANE230 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 230.1: PLANE230 Geometry (p. 1201). The element is defined by eight nodes and orthotropic material properties. The type of units (MKS or user defined) is specified through the **EMUNIT** command. **EMUNIT** also determines the value of EPZRO. The **EMUNIT** defaults are MKS units and EPZRO = 8.854 x 10-12 Farad/meter. A triangular-shaped element may be formed by defining the same node number for nodes K, L and O.

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems (p. 55). Properties not input default as described in Linear Material Properties in the *Material Reference*.

Nodal loads are defined with the **D** (*Lab* = VOLT) and **F** (*Lab* = AMPS) commands. The nodal forces, if any, should be input per unit of depth for a plane analysis and on a full 360° basis for an axisymmetric analysis.

The temperature (for material property evaluation only) body loads may be input based on their value at the element's nodes or as a single element value [**BF**, **BFE**]. In general, unspecified nodal values of temperatures default to the uniform value specified with the **BFUNIF** or **TUNIF** commands.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A summary of the element input is given in "PLANE230 Input Summary" (p. 1202). A general description of element input is given in Element Input (p. 43). For axisymmetric applications see Harmonic Axisymmetric Elements (p. 83).

PLANE230 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
VOLT

Real Constants
THK - Thickness (used only if KEYOPT(3) = 3)

Material Properties
MP command: RSVX, RSVY, PERX, PERY, LSST

EMUNIT command: EPZRO

Surface Loads
None

Body Loads

Temperature --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

Special Features
Birth and death

KEYOPT(3)
Element behavior:

0 --
Plane

1 --
Axisymmetric

3 --
Plane with thickness input, specified via real constant THK.

PLANE230 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in Table 230.1: PLANE230 Element Output Definitions (p. 1203).

Several items are illustrated in Figure 230.2: PLANE230 Output (p. 1203). The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results. The element output directions are parallel to the element coordinate system as shown in Figure 230.2: PLANE230 Output (p. 1203).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 230.2: PLANE230 Output

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 230.1: PLANE230 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	2
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	Y	Y
LOC	Output location (X, Y)	1	-
EF:X, Y, SUM	Electric field components and vector magnitude	1	1
JC:X, Y, SUM	Nodal conduction current density components and vector magnitude	1	1
JS:X, Y, SUM	Current density components (in the global Cartesian coordinate system) and vector magnitude [3]	-	1
JT:X, Y, SUM	Element conduction current density components and magnitude [3]	-	1
JHEAT:	Joule heat generation rate per unit volume [4] [5] [6]	-	1
SENE:	Stored electric energy [6]	-	1
D:X, Y, SUM	Electric flux density components and vector magnitude	-	1

1. The solution value is output only if calculated (based upon input data). The element solution is at the centroid.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Available only at centroid as a ***GET** item.

3. JS represents the sum of element conduction and displacement current densities. JT represents the element conduction current density. The element displacement current density (JD) can be derived from JS and JT as JD = JS-JT.

4. For a time-harmonic analysis, calculated Joule heat generation rate per unit volume (JHEAT) includes conduction heating and dielectric heating due to the loss tangent.

5. Calculated Joule heat generation rate per unit volume (JHEAT) may be made available for a subsequent thermal analysis with companion elements [**LDREAD**].

6. For a time-harmonic analysis, Joule losses (JHEAT) and stored energy (SENE) represent time-average values. These values are stored in both the real and imaginary data sets.

Table 230.2: PLANE230 Item and Sequence Numbers (p. 1204) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 230.2: PLANE230 Item and Sequence Numbers (p. 1204):

Name
output quantity as defined in the Table 230.1: PLANE230 Element Output Definitions (p. 1203)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 230.2: PLANE230 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
DX	NMISC	1
DY	NMISC	2
DSUM	NMISC	3
JTX	NMISC	4
JTY	NMISC	5
JTSUM	NMISC	6

PLANE230 Assumptions and Restrictions

- The area of the element must be positive.
- The element must lie in a global X-Y plane as shown in Figure 230.2: PLANE230 Output (p. 1203), and the Y-axis must be the axis of symmetry for axisymmetric analyses.
- An axisymmetric structure should be modeled in the +X quadrants.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- A face with a removed midside node implies that the potential varies linearly, rather than parabolically, along that face. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.
- This element is only compatible with elements having a VOLT degree of freedom and an electric current reaction solution. See Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*) for more information.

PLANE230 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Emag

- The Birth and death special feature is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID231

3-D 20-Node Electric Solid

MP <> <> <> <> <> EM <> PP EME <>
Product Restrictions

SOLID231 Element Description

SOLID231 is a 3-D 20-node, current-based electric element. The element has one degree of freedom, voltage, at each node. It can tolerate irregular shapes without much loss of accuracy. SOLID231 elements have compatible voltage shapes and are well suited to model curved boundaries.

This element is based on the electric scalar potential formulation and it is applicable to the following low frequency electric field analyses: steady-state electric conduction, time-harmonic quasistatic and transient quasistatic. See SOLID231 - 3-D 20-Node Electric Solid in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 231.1: SOLID231 Geometry

SOLID231 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 231.1: SOLID231 Geometry (p. 1207). The element is defined by 20 node points and the material properties. The type of units (MKS or user defined) is specified through the **EMUNIT** command. **EMUNIT** also determines the value of EPZRO. The **EMUNIT** defaults are MKS units and EPZRO = 8.854 x 10-12 Farad/meter. A prism-shaped element may be formed by defining duplicate K, L, and S; A and B; and O, P, and W node

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

numbers. A pyramid-shaped element and a tetrahedral-shaped element may also be formed as shown in Figure 231.1: SOLID231 Geometry (p. 1207).

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems (p. 55). Properties not input default as described in Linear Material Properties in the *Material Reference*.

Nodal loads are defined with the **D** (*Lab* = VOLT) and **F** (*Lab* = AMPS) commands. The temperature (for material property evaluation only) body loads may be input based on their value at the element's nodes or as a single element value [**BF**, **BFE**]. In general, unspecified nodal values of temperatures default to the uniform value specified with the **BFUNIF** or **TUNIF** commands.

A summary of the element input is given in "SOLID231 Input Summary" (p. 1208). A general description of element input is given in Element Input (p. 43).

SOLID231 Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B

Degrees of Freedom
VOLT

Real Constants
None

Material Properties
MP command: RSVX, RSVY, RSVZ, PERX, PERY, PERZ, LSST

EMUNIT command: EPZRO

Surface Loads
None

Body Loads

Temperature --
T(I), T(J), ..., T(Z), T(A), T(B)

Special Features
Birth and death

KEYOPT
None

SOLID231 Output Data

The solution output associated with the element is in two forms:

- Nodal potentials included in the overall nodal solution
- Additional element output as shown in Table 231.1: SOLID231 Element Output Definitions (p. 1209)

The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 231.1: SOLID231 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	2
TEMP	Temperatures T(I), T(J), ..., T(Z), T(A), T(B)	Y	Y
LOC	Output location (X, Y, Z)	1	-
EF:X, Y, Z, SUM	Electric field components and vector magnitude	1	1
JC:X, Y, Z, SUM	Nodal conduction current density components and vector magnitude	1	1
JS:X, Y, Z, SUM	Current density components (in the global Cartesian coordinate system) and vector magnitude [3]	-	1
JT:X, Y, Z, SUM	Element conduction current density components and magnitude [3]	-	1
JHEAT:	Joule heat generation rate per unit volume [4] [5] [6]	-	1
SENE:	Stored electric energy [6]	-	1
D:X, Y, Z, SUM	Electric flux density components and vector magnitude	-	1

1. The solution value is output only if calculated (based upon input data). The element solution is at the centroid.

2. Available only at centroid as a ***GET** item.

3. JS represents the sum of element conduction and displacement current densities. JT represents the element conduction current density. The element displacement current density (JD) can be derived from JS and JT as JD = JS-JT. JS can be used as a source current density for a subsequent magnetostatic analysis with companion elements [**LDREAD**].

4. For a time-harmonic analysis, calculated Joule heat generation rate per unit volume (JHEAT) includes conduction heating and dielectric heating due to the loss tangent.

5. Calculated Joule heat generation rate per unit volume (JHEAT) may be made available for a subsequent thermal analysis with companion elements [**LDREAD**].

6. For a time-harmonic analysis, Joule losses (JHEAT) and stored energy (SENE) represent time-average values. These values are stored in both the real and imaginary data sets.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Table 231.2: SOLID231 Item and Sequence Numbers (p. 1210) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 231.2: SOLID231 Item and Sequence Numbers (p. 1210):

Name
output quantity as defined in Table 231.1: SOLID231 Element Output Definitions (p. 1209)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 231.2: SOLID231 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
DX	NMISC	1
DY	NMISC	2
DZ	NMISC	3
DSUM	NMISC	4
JTX	NMISC	5
JTY	NMISC	6
JTZ	NMISC	7
JTSUM	NMISC	8

SOLID231 Assumptions and Restrictions

- The element must not have a zero volume or a zero length side. This occurs most frequently when the element is not numbered properly. Elements may be numbered either as shown in Figure 231.1: SOLID231 Geometry (p. 1207) or in an opposite fashion.
- An edge with a removed midside node implies that the potential varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information on the use of midside nodes.
- Degeneration to the form of pyramid should be used with caution. The element sizes, when degenerated, should be small in order to minimize the field gradients. Pyramid elements are best used as filler elements in meshing transition zones.
- This element is only compatible with elements having a VOLT degree of freedom and an electric current reaction solution. See Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*) for more information.
- The solenoidal current density is required for a solution, or for any postprocessing operations.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID231 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Emag

- The Birth and death special feature is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID232

3-D 10-Node Tetrahedral Electric Solid

MP <> <> <> <> <> EM <> PP EME <>
Product Restrictions

SOLID232 Element Description

SOLID232 is a 3-D, 10-node, current-based electric element. It is well suited to model irregular meshes (such as produced from various CAD/CAM systems). The element has one degree of freedom, voltage, at each node.

This element is based on the electric scalar potential formulation and it is applicable to the following low frequency electric field analyses: steady-state electric conduction, time-harmonic quasistatic and transient quasistatic. See SOLID232 - 3-D 10-Node Tetrahedral Electric Solid in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 232.1: SOLID232 Geometry

SOLID232 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 232.1: SOLID232 Geometry (p. 1213). The element is defined by 10 node points and the material properties. The type of units (MKS or user defined) is specified through the **EMUNIT** command. **EMUNIT** also determines the value of EPZRO. The **EMUNIT** defaults are MKS units and EPZRO = 8.854 x 10-12 Farad/meter.

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems (p. 55). Properties not input default as described in Linear Material Properties in the *Material Reference*.

Nodal loads are defined with the **D** (*Lab* = VOLT) and **F** (*Lab* = AMPS) commands. The temperature (for material property evaluation only) body loads may be input based on their value at the element's nodes or as a single element value [**BF**, **BFE**]. In general, unspecified nodal values of temperatures default to the uniform value specified with the **BFUNIF** or **TUNIF** commands.

A summary of the element input is given in "SOLID232 Input Summary" (p. 1214). A general description of element input is given in Element Input (p. 43).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID232 Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R

Degrees of Freedom
VOLT

Real Constants
None

Material Properties
MP command: RSVX, RSVY, RSVZ, PERX, PERY, PERZ, LSST

EMUNIT command: EPZRO

Surface Loads
None

Body Loads

Temperature --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P), T(Q), T(R)

Special Features
Birth and death

KEYOPTS
None

SOLID232 Output Data

The solution output associated with the element is in two forms:

- Nodal potentials included in the overall nodal solution
- Additional element output as shown in Table 232.1: SOLID232 Element Output Definitions (p. 1215)

The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50) in the *Element Reference*. See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 232.1: SOLID232 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	2
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P), T(Q), T(R)	Y	Y
LOC	Output location (X, Y, Z)	1	-
EF:X, Y, Z, SUM	Electric field components and vector magnitude	1	1
JC:X, Y, Z, SUM	Nodal conduction current density components and vector magnitude	1	1
JS:X, Y, Z, SUM	Current density components (in the global Cartesian coordinate system) and vector magnitude [3]	-	1
JT:X, Y, Z, SUM	Element conduction current density components and magnitude [3]	-	1
JHEAT:	Joule heat generation rate per unit volume [4] [5] [6]	-	1
SENE:	Stored electric energy [6]	-	1
D:X, Y, Z, SUM	Electric flux density components and vector magnitude	-	1

1. The solution value is output only if calculated (based upon input data). The element solution is at the centroid.

2. Available only at centroid as a ***GET** item.

3. JS represents the sum of element conduction and displacement current densities. JT represents the element conduction current density. The element displacement current density (JD) can be derived from JS and JT as JD = JS-JT. JS can be used as a source current density for a subsequent magnetostatic analysis with companion elements [**LDREAD**].

4. For a time-harmonic analysis, calculated Joule heat generation rate per unit volume (JHEAT) includes conduction heating and dielectric heating due to the loss tangent.

5. Calculated Joule heat generation rate per unit volume (JHEAT) may be made available for a subsequent thermal analysis with companion elements [**LDREAD**].

6. For a time-harmonic analysis, Joule losses (JHEAT) and stored energy (SENE) represent time-average values. These values are stored in both the real and imaginary data sets.

Table 232.2: SOLID232 Item and Sequence Numbers (p. 1216) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 232.2: SOLID232 Item and Sequence Numbers (p. 1216):

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name
output quantity as defined in the Table 232.1: SOLID232 Element Output Definitions (p. 1215)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 232.2: SOLID232 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
DX	NMISC	1
DY	NMISC	2
DZ	NMISC	3
DSUM	NMISC	4
JTX	NMISC	5
JTY	NMISC	6
JTZ	NMISC	7
JTSUM	NMISC	8

SOLID232 Assumptions and Restrictions

- The element must not have a zero volume or a zero length side. This occurs most frequently when the element is not numbered properly. Elements may be numbered either as shown in Figure 232.1: SOLID232 Geometry (p. 1213) or in an opposite fashion.
- An edge with a removed midside node implies that the potential varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information on the use of midside nodes.
- This element is only compatible with elements having a VOLT degree of freedom and an electric current reaction solution. See Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*) for more information.
- The solenoidal current density is required for a solution, or for any postprocessing operations.

SOLID232 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Emag

- The birth and death special feature is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PLANE233

2-D 8-Node Electromagnetic Solid

MP <> <> <> <> <> EM <> PP EME <>
Product Restrictions

PLANE233 Element Description

PLANE233 is a 2-D element capable of modeling planar or axisymmetric electromagnetic fields. The element is defined by 8 or 6 nodes, and has up to 2 degrees of freedom per node: Z component of the magnetic vector potential (AZ) and electric scalar potential (VOLT). The element also has an option to perform an electromagnetic analysis with time-integrated electric potential (VOLT) as a degree of freedom.

In an electromagnetic analysis, the electric degree of freedom is the electric potential (VOLT) defined at each node. All VOLT degrees of freedom must be coupled in a 2-D electromagnetic region (**CP**) such that the voltage drop across the thickness has a single value. The element also has an option for performing an electromagnetic analysis with time-integrated electric potential.

In a stranded coil analysis, the electric degrees of freedom are the voltage drop across the coil (VOLT) and the electromotive force (EMF). In a coil region, all the VOLT and EMF degrees of freedom must be coupled (**CP**). The element has the option to perform a stranded coil analysis with time-integrated voltage drop or time-integrated electromotive force.

PLANE233 is applicable to 2-D static, time-harmonic and time-transient electromagnetic analyses. The magnetic analysis option is typically used to model air, iron, nonferrous materials and permanent magnets. The analysis is driven by the current density applied as an element body load. The electromagnetic analysis option is suitable for modeling solid (massive) conductors. This analysis may be voltage- or current-driven, or circuit-fed. The electromagnetic analysis has the option to suppress the eddy current effect in time-varying analyses to model stranded conductors. The velocity effects can be taken into account in a static, time-harmonic or time-transient electromagnetic analysis.

The stranded coil analysis option is suitable for modeling a stranded winding with a prescribed current flow direction vector. The stranded coil can be voltage- or current-driven, as well as circuit-fed.

The following command macros can be used with PLANE233 for solution postprocessing: **CURR2D**, **EMAGERR**, **EMFT**, **FLUXV**, **MMF**, **PLF2D**, **POWERH**. See Electric and Magnetic Macros in the *Low-Frequency Electromagnetic Analysis Guide* for more details.

See PLANE233 theory in the *Mechanical APDL Theory Reference* for more details about this element. The element has nonlinear magnetic capability for modeling B-H curves or permanent magnet demagnetization curves for static and time-transient analyses.

Figure 233.1: PLANE233 Geometry

PLANE233 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 233.1: PLANE233 Geometry (p. 1217). A triangular-shaped element may be formed by defining the same node number for nodes K, L and O.

The type of units (MKS or user defined) is specified via the **EMUNIT** command. **EMUNIT** also determines the value of MUZRO and EPZRO. The **EMUNIT** defaults are MKS units and MUZRO = $4\pi 10^{-7}$ Henry/meter and EPZRO = 8.854 x 10^{-12} Farad/meter. In addition to MUZRO and EPZRO, orthotropic relative permeability is specified through the MURX and MURY material property labels. The Z-depth resistivity and permittivity are specified using the RSVZ and PERZ material property labels respectively. MGXX and MGYY represent vector components of the coercive force for permanent magnet materials. The magnitude of the coercive force is the square root of the sum of the squares of the components. The direction of polarization is determined by the components MGXX and MGYY. Permanent magnet polarization directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems (p. 55). Nonlinear magnetic B-H properties are entered with the **TB** command as described in Material Models in the *Material Reference*. Nonlinear orthotropic magnetic properties may be specified with a combination of a B-H curve and linear relative permeability. The B-H curve will be used in each element coordinate direction where a zero value of relative permeability is specified. Only one B-H curve may be specified per material.

For the plane option (KEYOPT(3) = 0), you can also define the element thickness (Z depth) using the real constant THK. For the axisymmetric option (KEYOPT(3) = 1), you can specify the fraction of the 360° basis using the same real constant.

Nodal loads are defined via the **D** and **F** commands. The nodal forces, if any, should be input per unit of depth for a plane analysis and on a full 360° basis for an axisymmetric analysis unless the THK real constant is specified. For edge-based analysis, the **D** command with *Lab* = AZ applies the edge-flux constraint to the node. Flux-parallel boundary conditions are prescribed by setting AZ to zero. No AZ constraint is required to set flux-normal boundary conditions.

For massive conductors (KEYOPT(1) = 1), *Lab* = VOLT is valid with the **D** command and *VALUE* defines the electric potential. Note that electric potential is time-integrated if KEYOPT(2) = 2. With the **F** command, *Lab* = AMPS and *VALUE* corresponds to the total current.

For stranded coils (KEYOPT(1) = 2), *Lab* = VOLT is valid with the **D** command and *VALUE* defines the voltage drop across the coil. The **D** command with *Lab* = EMF can be used to apply constraints on the electromotive force. Note that voltage drop and the electromotive force are time-integrated if KEYOPT(2) = 2. The total current through the coil can be applied via the **F** command using *Lab* = AMPS.

The temperature (for material property evaluation only) body loads may be input based on their value at the element's nodes or as a single element value [**BF**, **BFE**]. In general, unspecified nodal values of temperatures default to the uniform value specified with the **BFUNIF** or **TUNIF** commands. For modeling stranded conductors with KEYOPT(1) = 0, source current density may be applied to an area [**BFA**] or input as an element value [**BFE**].

A summary of the element input is given in "PLANE233 Input Summary". A general description of element input is given in Element Input.

PLANE233 Input Summary

Nodes

I, J, K, L, M, N, O, P

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Degrees of Freedom

Set by KEYOPT(1).

Real Constants

Thickness (THK) is the only real constant for KEYOPT(1) = 0 or 1.

The following are the real constants for KEYOPT(1) = 2:

THK, SC, NC, RAD, TZ, R
SYM

See Table 233.1: PLANE233 Real Constants (p. 1221) for more information.

Material Properties

TB command: Magnetic

MP command: MURX, MURY, MGXX, MGYY, RSVZ, PERZ (See ""PLANE233 Assumptions and Restrictions" (p. 1224)")

EMUNIT command: EPZRO, MUZERO

Surface Loads

None

Body Loads

Temperature --

T(I), T(J), ..., T(O), T(P)

Source Current Density (valid for KEYOPT(1) = 0 only) --

spare, spare, JSZ(I), PHASE(I),

spare, spare, JSZ(J), PHASE(J),

...

spare, spare, JSZ(O), PHASE(O),

spare, spare, JSZ(P), PHASE(P)

Velocity (valid for KEYOPT(1) = 1 only) --

VELOX(I), VELOY(I), spare, spare, spare, OMEGAZ(I)

VELOX(J), VELOY(J), spare, spare, spare, OMEGAZ(J)

...

VELOX(P), VELOY(P), spare, spare, spare, OMEGAZ(P)

Special Features

Linear perturbation

KEYOPT(1)

Element capability and degrees of freedom:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0 --
Magnetic:

AZ

1 --
Electromagnetic:

AZ, VOLT

2 --
Stranded coil:

AZ, VOLT, EMF

KEYOPT(2)
Coupling method between magnetic and electric degrees of freedom (KEYOPT(1) = 1 or 2); also defines the meaning of the VOLT and EMF degrees of freedom for KEYOPT(1) = 1 or 2:

0 --
Strong (matrix) coupling. Produces an unsymmetric matrix. In a linear analysis, a coupled response is achieved after one iteration. Applicable to all analysis types.

1 --
Weak (load vector). Produces a symmetric matrix and requires at least two iterations to achieve a coupled response. Applicable to static and transient analyses only. (See "PLANE233 Assumptions and Restrictions" (p. 1224))

2 --
Strong (matrix) coupling with time-integrated electric potential (VOLT) for KEYOPT(1) = 1 (electromagnetic analysis). Produces a symmetric matrix.

Strong (matrix) coupling with time-integrated voltage drop (VOLT) and time-integrated electromotive force (EMF) for KEYOPT(1) = 2 (stranded coil analysis). Produces a symmetric matrix if the coil symmetry factor is 1; produces a nonsymmetric matrix if the coil symmetry factor is greater than 1.

In a linear analysis, a coupled response is achieved after one iteration. Applicable to harmonic and transient analyses only.

KEYOPT(3)
Element behavior:

0 --
Plane

1 --
Axisymmetric

KEYOPT(5)
Eddy currents or velocity effects in electromagnetic (KEYOPT(1) =1) analyses:

0 --
Active

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
Eddy currents are suppressed in harmonic or transient analyses

2 --
Velocity effects are suppressed

3 --
Eddy currents and velocity effects are suppressed

KEYOPT(7)
Electromagnetic force output:

0 --
At each element node (corner and midside)

1 --
At element corner nodes only (midside node forces are condensed to the corner nodes)

KEYOPT(8)
Electromagnetic force calculation:

0 --
Maxwell

1 --
Lorentz

Table 233.1: PLANE233 Real Constants

No.	Name	Description	Default	Definition
1	THK	Thickness (plane) or fraction of the 360° basis (axisymmetric)	1	Applicable to all KEYOPT(1) = 0, 1, or 2.
2	SC	Coil cross-sectional area	none	True physical cross-section of the coil regardless of symmetry modeling considerations. It includes the cross-sectional area of the wire and the non-conducting material filling the space between the winding.
3	NC	Number of coil turns	1	Total number of winding turns in a coil regardless of any symmetry modeling considerations.
4	RAD	Mean radius of the coil (axisymmetric)	none	Mean radius of the axisymmetric coil model. If the mean radius is not known, input VC/((2π)(SC)(THK)) , where VC is the full symmetry true physical volume of the coil. VC includes the volume occupied by the wire and the non-conducting material filling the space between the winding.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

No.	Name	Description	Default	Definition
5	TZ	Current polarity	1 (plane) -1 (axisym-metric)	The current flow direction (1 or -1) with respect to Z-axis
6	R	Coil resistance	none	Total coil DC resistance regard-less of any symmetry modeling considerations.
7	SYM	Coil symmetry factor	1	Ratio of the full symmetry coil cross-sectional area (SC) to the modeled coil area. The input should be greater or equal to 1.

PLANE233 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in Table 233.2: PLANE233 Element Output Definitions (p. 1222)

The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [**ETABLE**, **ESOL**]. The O column indicates the availability of the items in the file Jobname.OUT. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is always available, a number refers to a table footnote that describes when the item is conditionally available, and a - indicates that the item is not available.

Table 233.2: PLANE233 Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
NODES	Nodes - I, J,…, O,P	-	Y
MAT	Material number	-	Y
THICK	Thickness	-	Y
VOLU:	Volume	-	Y
XC, YC	Location where results are reported	-	2
TEMP	Input temperatures T(I), T(J), ..., T(O), T(P)	-	Y
LOC	Output location (X, Y)	-	-
B: X, Y, SUM	Magnetic flux density components and vector magnitude	-	1
H: X, Y, SUM	Magnetic field intensity components and vector magnitude	-	1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
EF: Z, SUM	Electric field intensity Z component and vector magnitude [7]	-	1
JC: Z, SUM	Conduction current density Z component and vector magnitude [7]	-	1
FMAG: X, Y, SUM	Electromagnetic force components and magnitude [3]	-	1
JT: Z, SUM	Conduction current density Z component (in the global Cartesian coordinate system) and vector magnitude [6]	-	1
JS: Z, SUM	Current density Z component (in the global Cartesian coordinate system) and vector magnitude [4] [6]	-	1
JHEAT:	Joule heat generation rate per unit volume [3] [5] [6]	-	1
SENE or MENE:	Magnetic energy [3]	-	1
COEN	Magnetic co-energy [3]	-	1
AENE	Apparent magnetic energy [3]	-	1
IENE	Incremental magnetic energy [3]	-	1

1. The solution value is output only if calculated (based upon input data). The element solution is at the centroid.

2. Available only at centroid as a ***GET** item.

3. For a time-harmonic analysis, electromagnetic forces (FMAG), Joule losses (JHEAT) and stored energy (SENE, MENE) represent time-average values. These values are stored in both the real and imaginary data sets. In a linear perturbation analysis, only incremental and apparent energy values are time-averaged.

4. JS represents the sum of element conduction and displacement current densities.

5. Calculated Joule heat generation rate per unit volume (JHEAT) may be made available for a subsequent thermal analysis with companion elements [**LDREAD**].

6. For the stranded coil analysis option (KEYOPT(1) = 2), JT and JS are the effective current densities as they are calculated based on the coil cross-sectional area (SC) that includes the wire and the non-conducting material filling the space between the winding. JHEAT represents the effective Joule heat generation rate per unit volume as it is calculated based on the modeled coil volume that includes the wire and the non-conducting material filling the space between the winding.

7. Not available with the stranded coil option (KEYOPT(1) = 2).

Table 233.3: PLANE233 Item and Sequence Numbers (p. 1224) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and Table 233.3: PLANE233 Item and Sequence Numbers (p. 1224) in this manual for more information. The following notation is used in Table 233.3: PLANE233 Item and Sequence Numbers (p. 1224):

Name: output quantity as defined in Table 233.2: PLANE233 Element Output Definitions (p. 1222)

Item: predetermined Item label for **ETABLE** command

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

E: sequence number for single-valued or constant element data

Table 233.3: PLANE233 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
JTZ	NMISC	1

PLANE233 Assumptions and Restrictions

- The area of the element must be positive.
- The element must lie in a global X-Y plane as shown in Figure 233.1: PLANE233 Geometry (p. 1217) and the Y-axis must be the axis of symmetry for axisymmetric analyses. An axisymmetric structure should be modeled in the +X quadrants.
- A face with a removed midside node implies that the potentials vary linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information on the use of midside nodes.
- Permanent magnets are not permitted in a harmonic analysis.
- It is not recommended to use the weak coupling option (KEYOPT(2) = 1) in a transient electromagnetic analysis with eddy currents or a transient stranded coil analysis because multiple iterations may be required to achieve convergence.
- In a transient analysis, the *THETA* integration parameter defaults to the values shown in the following table. You can use the **TINTP** command to modify the default setting.

  Table 233.4: THETA Default Values

Analysis Type	KEYOPT Values	THETA Default Value
Strongly coupled transient electromagnetic analysis with electric potential or stranded coil analysis with voltage drop (VOLT)	KEYOPT(1) = 1 or 2 and KEYOPT(2) = 0	1.0
Strongly coupled transient electromagnetic analysis with time-integrated electric potential or stranded coil analysis with time-integrated voltage drop (VOLT)	KEYOPT(1) = 1 or 2 and KEYOPT(2) = 2	0.5

- The electrical permittivity material input (**MP**,PERZ) is applicable to electromagnetic harmonic analyses (KEYOPT(1) = 1) only.
- In an electromagnetic (KEYOPT(1) = 1) domain, all VOLT degrees of freedom must be coupled (**CP**).
- In a stranded coil (KEYOPT(1) = 2) domain, all VOLT and EMF degrees of freedom must be coupled (**CP**).
- Unlike the 2-D magnetic elements PLANE53 and PLANE13 that model the eddy current effects with the AZ option (KEYOPT(1) = 0) when **BFE**,,JS is not specified, PLANE233 always acts as a stranded conductor

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in a harmonic or transient analysis with KEYOPT(1) = 0. In this respect, the PLANE233 behavior is consistent with the 3-D electromagnetic elements (e.g. SOLID236) behavior.

- This element may not be compatible with other elements having a VOLT degree of freedom. See Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*) for more information. The electromagnetic analysis with time-integrated electric potential (KEYOPT(2) = 2) cannot be used with current-based circuit (e.g. CIRCU124) or low-frequency electric (e.g. PLANE230) elements.

PLANE233 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Emag: The birth and death special feature is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID236

3-D 20-Node Electromagnetic Solid

MP <> <> <> <> <> EM <> PP EME <>
Product Restrictions

SOLID236 Element Description

SOLID236 is a 3-D 20-node element capable of modeling electromagnetic fields. The element has magnetic and electric degrees of freedom. Magnetic degrees of freedom are based on the edge-flux formulation.

The edge-flux (AZ) degrees of freedoms are the line integrals of the magnetic vector potential along the element edges. They are defined at the midside nodes only, and there are no magnetic degrees of freedom associated with the corner nodes. The edge-flux formulation uses tree gauging (**GAUGE**) to produce a unique solution.

In an electromagnetic analysis, the electric degree of freedom is the electric potential (VOLT) defined at each node. The element also has an option to perform an electromagnetic analysis with time-integrated electric potential.

In a stranded coil analysis, the electric degrees of freedom are the voltage drop across the coil (VOLT) and the electromotive force (EMF). In a coil region, all the VOLT and EMF degrees of freedom must be coupled (**CP**). The element has the option to perform a stranded coil analysis with time-integrated voltage drop or time-integrated electromotive force.

The element is applicable to 3-D static, time-harmonic and time-transient electromagnetic analyses. The magnetic analysis option typically is used to model air, iron, nonferrous materials and permanent magnets. The analysis is driven by the current density applied as an element body load.

The electromagnetic analysis option is suitable for modeling solid (massive) conductors. The solid (massive) conductor may be voltage-driven or current-driven, as well as circuit-fed. The electromagnetic analysis has the option to suppress the eddy current effect in time-varying analyses to model stranded conductors. The Hall effect can be taken into account in a static or transient electromagnetic analysis. The velocity effects can be taken into account in a static, time-harmonic or time-transient electromagnetic analysis.

The stranded coil analysis option is suitable for modeling a stranded winding with a prescribed current flow direction vector. The stranded coil may be voltage-driven or current-driven, as well as circuit-fed.

The following command macros can be used with SOLID236 for solution postprocessing: **EMAGERR**, **EMFT**, **MMF**, **POWERH**. See Electric and Magnetic Macros in the *Low-Frequency Electromagnetic Analysis Guide* for more details.

See SOLID236 theory in the *Mechanical APDL Theory Reference* for more details about this element. The element has nonlinear magnetic capability for modeling B-H curves or permanent magnet demagnetization curves for static and time-transient analyses.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 236.1: SOLID236 Geometry

SOLID236 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 236.1: SOLID236 Geometry (p. 1228). The element is defined by 20 node points and the material properties. A prism-shaped element may be formed by defining duplicate K, L, and S; A and B; and O, P, and W node numbers. A pyramid-shaped element and a tetrahedral-shaped element may also be formed as shown in Figure 236.1: SOLID236 Geometry (p. 1228).

The type of units (MKS or user defined) is specified via the **EMUNIT** command. **EMUNIT** also determines the value of MUZRO and EPZRO. The **EMUNIT** defaults are MKS units and MUZRO = $4\pi 10^{-7}$ Henry/meter and EPZRO = 8.854 x 10^{-12} Farad/meter. In addition to MUZRO and EPZRO, orthotropic relative permeability and permittivity is available and is specified through the MURX, MURY, and MURZ and PERX, PERY, PERZ material options, respectively. Orthotropic resistivity is specified through RSVX, RSVY, and RSVZ material property labels. MGXX, MGYY, and MGZZ represent vector components of the coercive force for permanent magnet materials. The magnitude of the coercive force is the square root of the sum of the squares of the components. The direction of polarization is determined by the components MGXX, MGYY, and MGZZ. Permanent magnet polarization directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems (p. 55). Nonlinear magnetic B-H properties are entered via the **TB** command, as described in Material Models in the *Material Reference*. Nonlinear orthotropic magnetic properties may be specified with a combination of a B-H curve and linear relative permeability. The B-H curve will be used in each element coordinate direction where a zero value of relative permeability is specified. Only one B-H curve may be specified per material. In an electromagnetic analysis, the Hall coefficient is specified through the RH material option.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Nodal loads are defined via the **D** and **F** commands. For edge-based analyses, the edge-flux constraint is applied to the node via the **D** command with *Lab* = AZ. Flux-parallel boundary conditions are prescribed by setting AZ to zero. No AZ constraint is required to set flux-normal boundary conditions. The **DFLX** command can be used to impose a uniform magnetic flux on the selected nodes.

For massive conductors (KEYOPT(1) = 1), *Lab* = VOLT is valid with the **D** command and *VALUE* defines the electric potential. Note that electric potential is time-integrated if KEYOPT(2) = 2. With the **F** command, *Lab* = AMPS and *VALUE* corresponds to the total current.

For stranded coils (KEYOPT(1) = 2), *Lab* = VOLT is valid with the **D** command and *VALUE* defines the voltage drop across the coil. The **D** command with *Lab* = EMF can be used to apply constraints on the electromotive force. Note that voltage drop and the electromotive force are time-integrated if KEYOPT(2) = 2. The total current through the coil can be applied via the **F** command using *Lab* = AMPS.

The temperature (for material property evaluation only) body loads may be input based on their value at the element's nodes or as a single element value [**BF**, **BFE**]. In general, unspecified nodal values of temperatures default to the uniform value specified via the **BFUNIF** or **TUNIF** commands.

For modeling stranded conductors with KEYOPT(1) = 0, source current density may be applied to an area or volume [**BFA** or **BFV**] or input as an element value [**BFE**]. The vector components of the current density are with respect to the element coordinate system. See "SOLID236 Assumptions and Restrictions" (p. 1234) for a description of the solenoidal condition.

A summary of the element input is given in "SOLID236 Input Summary" (p. 1229). A general description of element input is given in Element Input (p. 43).

SOLID236 Input Summary

Nodes

I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B

Degrees of Freedom

See KEYOPT(1)

Real Constants

There are no real constants for KEYOPT(1) = 0 or 1.

The following are the real constants for KEYOPT(1) = 2:

SC, NC, VC, TX, TY, TZ
R, SYM

See Table 236.1: SOLID 236 Real Constants (p. 1232) for more information.

Material Properties

TB command: Magnetic

MP command: MURX, MURY, MURZ, MGXX, MGYY, MGZZ, RSVX, RSVY, RSVZ, RH, PERX, PERY, PERZ (see "SOLID236 Assumptions and Restrictions" (p. 1234))

EMUNIT command: EPZRO, MUZERO

Surface Loads

None

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Body Loads

Temperature --
T(I), T(J), ..., T(Z), T(A), T(B)

Source Current Density (valid for KEYOPT(1) = 0 only)
JSX(I), JSY(I), JSZ(I), PHASE(I),

JSX(J), JSY(J), JSZ(J), PHASE(J),

...

JSX(Z), JSY(Z), JSZ(Z), PHASE(Z),

JSX(A), JSY(A), JSZ(A), PHASE(A),

JSX(B), JSY(B), JSZ(B), PHASE(B)

Velocity (valid for KEYOPT(1) = 1 only) --

VELOX(I), VELOY(I), VELOZ(I), OMEGAX(I), OMEGAY(I), OMEGAZ(I)

VELOX(J), VELOY(J), VELOZ(J), OMEGAX(J), OMEGAY(J), OMEGAZ(J)

...

VELOX(Z), VELOY(Z), VELOZ(Z), OMEGAX(Z), OMEGAY(Z), OMEGAZ(Z)

VELOX(A), VELOY(A), VELOZ(A), OMEGAX(A), OMEGAY(A), OMEGAZ(A)

VELOX(B), VELOY(B), VELOZ(B), OMEGAX(B), OMEGAY(B), OMEGAZ(B)

Special Features
Linear perturbation

Magnetic materials

KEYOPT(1)
Element capability and degrees of freedom:

0 --
Magnetic:

AZ

1 --
Electromagnetic:

AZ, VOLT

2 --
Stranded coil:

AZ, VOLT, EMF

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(2)

Coupling method between magnetic and electric degrees of freedom (KEYOPT(1) = 1 or 2); also defines the meaning of the VOLT and EMF degrees of freedom for KEYOPT(1) = 1 or 2:

0 --

Strong (matrix) coupling. Produces an unsymmetric matrix. In a linear analysis, a coupled response is achieved after one iteration. Applicable to all analysis types.

1 --

Weak (load vector). Produces a symmetric matrix and requires at least two iterations to achieve a coupled response. Applicable to static and transient analyses only. (see "SOLID236 Assumptions and Restrictions" (p. 1234))

2 --

Strong (matrix) coupling with time-integrated electric potential (VOLT) for KEYOPT(1) = 1 (electromagnetic analysis). Produces a symmetric matrix.

Strong (matrix) coupling with time-integrated voltage drop (VOLT) and time-integrated electromotive force (EMF) for KEYOPT(1) = 2 (stranded coil analysis). Produces a symmetric matrix if the coil symmetry factor is 1; produces a nonsymmetric matrix if the coil symmetry factor is greater than 1.

In a linear analysis, a coupled response is achieved after one iteration. Applicable to harmonic and transient analyses only.

KEYOPT(5)

Eddy currents or velocity effects in electromagnetic (KEYOPT(1) =1) analyses:

0 --

Active

1 --

Eddy currents are suppressed in harmonic or transient analyses

2 --

Velocity effects are suppressed

3 --

Eddy currents and velocity effects are suppressed

KEYOPT(7)

Electromagnetic force output:

0 --

At each element node (corner and midside)

1 --

At element corner nodes only (midside node forces are condensed to the corner nodes)

KEYOPT(8)

Electromagnetic force calculation:

0 --

Maxwell

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
Lorentz

Table 236.1: SOLID 236 Real Constants

No.	Name	Description	Default	Definition
1	SC	Coil cross-sectional area	none	True physical cross-section of the coil regardless of symmetry modeling considerations. It includes the cross-sectional area of the wire and the non-conducting material filling the space between the winding.
2	NC	Number of coil turns	1	Total number of winding turns in a coil regardless of any symmetry modeling considerations.
3	VC	Coil volume	none	True physical volume of the coil regardless of symmetry modeling considerations. It includes the volume occupied by the wire and the non-conducting material filling the space between the winding.
4	TX	Coil winding X-directional cosine	0	The coil direction vector T = {TX, TY, TZ}^{T} is a unit vector tangent to the coil winding. It designates the current flow direction.
5	TY	Coil winding Y-directional cosine	1	
6	TZ	Coil winding Z-directional cosine	0	
7	R	Coil resistance	none	Total coil DC resistance regardless of any symmetry modeling considerations.
8	SYM	Coil symmetry factor	1	Ratio of the true physical volume of the coil (real constant VC) to the modeled coil volume. The input should be greater or equal to 1.

SOLID236 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in Table 236.2: SOLID236 Element Output Definitions (p. 1233)

The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 236.2: SOLID236 Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
NODES	Nodes - I, J,…, Z, A, B	-	Y
MAT	Material number	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	-	2
TEMP	Input temperatures T(I), T(J), ..., T(Z), T(A), T(B)	-	Y
LOC	Output location (X, Y, Z)	-	-
B: X, Y, Z, SUM	Magnetic flux density components and vector magnitude	-	1
H: X, Y, Z, SUM	Magnetic field intensity components and vector magnitude	-	1
EF: X, Y, Z, SUM	Electric field intensity components and magnitude [7]	-	1
JC: X, Y, Z, SUM	Nodal conduction current density components and magnitude [7]	-	1
FMAG: X, Y, Z, SUM	Electromagnetic force components and magnitude [3]	-	1
JT: X, Y, Z, SUM	Element conduction current density components (in the global Cartesian coordinate system) and vector magnitude [6	-	1
JS: X, Y, Z, SUM	Element current density components (in the global Cartesian coordinate system) and vector magnitude [4] [6	-	1
JHEAT:	Joule heat generation rate per unit volume [3] [5] [6]	-	1
SENE or MENE:	Magnetic energy [3]	-	1
COEN	Magnetic co-energy [3]	-	1
AENE	Apparent magnetic energy [3]	-	1
IENE	Incremental magnetic energy [3]	-	1

1. The solution value is output only if calculated (based upon input data). The element solution is at the centroid.

2. Available only at centroid as a ***GET** item.

3. For a time-harmonic analysis, electromagnetic forces (FMAG), Joule losses (JHEAT) and stored energy (SENE, MENE) represent time-average values. These values are stored in both the real and imaginary data sets. In a linear perturbation analysis, only incremental and apparent energy values are time-averaged.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. JS represents the sum of element conduction and displacement current densities.

5. Calculated Joule heat generation rate per unit volume (JHEAT) may be made available for a subsequent thermal analysis with companion elements [**LDREAD**].

6. For the stranded coil analysis option (KEYOPT(1) = 2), JT and JS are the effective current densities as they are calculated based on the coil cross-sectional area (SC) that includes the wire and the non-conducting material filling the space between the winding. JHEAT represents the effective Joule heat generation rate per unit volume as it is calculated based on the modeled coil volume that includes the wire and the non-conducting material filling the space between the winding.

7. Not available with the stranded coil option (KEYOPT(1) = 2).

Table 236.3: SOLID236 Item and Sequence Numbers (p. 1234) lists output available via the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 236.3: SOLID236 Item and Sequence Numbers (p. 1234):

Name
output quantity as defined in Table 236.2: SOLID236 Element Output Definitions (p. 1233)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 236.3: SOLID236 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
JTX	NMISC	1
JTY	NMISC	2
JTZ	NMISC	3
JTSUM	NMISC	4

SOLID236 Assumptions and Restrictions

- The element must not have a zero volume or a zero length side. This occurs most frequently when the element is not numbered properly. Elements may be numbered either as shown in Figure 236.1: SOLID236 Geometry (p. 1228) or in an opposite fashion.

- Midside nodes may not be removed.

- Degeneration to the form of pyramid should be used with caution. The element sizes, when degenerated, should be small in order to minimize the field gradients. Pyramid elements are best used as filler elements in meshing transition zones.

- The magnetic analysis option (KEYOPT(1) = 0) requires the source current density specified via the **BFE**,,JS command to be solenoidal.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Permanent magnets are not permitted in a harmonic analysis.
- It is not recommended to use the weak coupling option (KEYOPT(2) = 1) in a transient electromagnetic analysis with eddy currents or a transient stranded coil analysis because multiple iterations may be required to achieve convergence.
- In a transient analysis, the *THETA* integration parameter defaults to the values shown in the following table. Issue the **TINTP** command to modify the default setting.

Table 236.4: THETA Default Values

Analysis Type	KEYOPT Values	THETA Default Value
Strongly coupled transient electromagnetic analysis with electric potential or stranded coil analysis with voltage drop (VOLT)	KEYOPT(1) = 1 or 2 and KEYOPT(2) = 0	1.0
Strongly coupled transient electromagnetic analysis with time-integrated electric potential or stranded coil analysis with time-integrated voltage drop (VOLT)	KEYOPT(1) = 1 or 2 and KEYOPT(2) = 2	0.5

- The electrical permittivity material input (**MP**,PERX, also PERY, PERZ) is applicable to electromagnetic harmonic analyses (KEYOPT(1) = 1) only.
- In a stranded coil (KEYOPT(1) = 2) domain, the winding direction vector $T = \{TX, TY, TZ\}^T$ must be specified in the element coordinate system and all VOLT and EMF degrees of freedom must be coupled (**CP**).
- This element may not be compatible with other elements having a VOLT degree of freedom. See Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*) for more information. The electromagnetic analysis with time-integrated electric potential (KEYOPT(2) = 2) cannot be used with current-based circuit (e.g. CIRCU124) or low-frequency electric (e.g. SOLID231) elements.

SOLID236 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Emag

- The birth and death special feature is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID237

3-D 10-Node Electromagnetic Solid

MP <> <> <> <> <> EM <> PP EME <>
Product Restrictions

SOLID237 Element Description

SOLID237 is a 3-D 10-node, element capable of modeling electromagnetic fields. The element has magnetic and electric degrees of freedom. Magnetic degrees of freedom are based on the edge-flux formulation.

The edge-flux (AZ) degrees of freedoms are the line integrals of the magnetic vector potential along the element edges. They are defined at the midside nodes only and there are no magnetic degrees of freedom associated with the corner nodes. The edge-flux formulation uses tree gauging (see the **GAUGE** command) to produce a unique solution.

In an electromagnetic analysis, the electric degree of freedom is the electric potential (VOLT) defined at each node. The element also has an option to perform an electromagnetic analysis with time-integrated electric potential.

In a stranded coil analysis, the electric degrees of freedom are the voltage drop across the coil (VOLT) and the electromotive force (EMF). In a coil region, all the VOLT and EMF degrees of freedom must be coupled using the **CP** command. The element has the option to perform a stranded coil analysis with time-integrated voltage drop or time-integrated electromotive force.

The element is applicable to 3-D static, time-harmonic and time-transient electromagnetic analyses. The magnetic analysis option typically is used to model air, iron, nonferrous materials and permanent magnets. The analysis is driven by the current density applied as an element body load.

The electromagnetic analysis option is suitable for modeling solid (massive) conductors. The solid (massive) conductor may be voltage- or current-driven, as well as circuit-fed. The electromagnetic analysis has the option to suppress the eddy current effect in time-varying analyses to model stranded conductors. The Hall effect can be taken into account in a static or transient electromagnetic analysis. The velocity effects can be taken into account in a static, time-harmonic or time-transient electromagnetic analysis.

The stranded coil analysis option is suitable for modeling a stranded winding with a prescribed current flow direction vector. The stranded coil may be voltage- or current-driven, as well as circuit-fed.

The following command macros can be used with SOLID237 for solution postprocessing: **EMAGERR**, **EMFT**, **MMF**, **POWERH**. See Electric and Magnetic Macros in the *Low-Frequency Electromagnetic Analysis Guide* for more details.

The element has nonlinear magnetic capability for modeling B-H curves or permanent magnet demagnetization curves for static and time-transient analyses. See SOLID237 in the *Mechanical APDL Theory Reference* for more details about this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 237.1: SOLID237 Geometry

SOLID237 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 237.1: SOLID237 Geometry (p. 1238). The element is defined by 10 node points and the material properties.

The type of units (MKS or user defined) is specified through the **EMUNIT** command. **EMUNIT** also determines the value of MUZRO and EPZRO. The **EMUNIT** defaults are MKS units and MUZRO = $4\pi 10^{-7}$ Henry/meter and EPZRO = 8.854 x 10^{-12} Farad/meter. In addition to MUZRO and EPZRO, orthotropic relative permeability and permittivity is available and is specified through the MURX, MURY, and MURZ and PERX, PERY, PERZ material options, respectively. Orthotropic resistivity is specified through RSVX, RSVY, and RSVZ material property labels. MGXX, MGYY, and MGZZ represent vector components of the coercive force for permanent magnet materials. The magnitude of the coercive force is the square root of the sum of the squares of the components. The direction of polarization is determined by the components MGXX, MGYY, and MGZZ. Permanent magnet polarization directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems (p. 55). Nonlinear magnetic B-H properties are entered with the **TB** command as described in Material Models in the *Material Reference*. Nonlinear orthotropic magnetic properties may be specified with a combination of a B-H curve and linear relative permeability. The B-H curve will be used in each element coordinate direction where a zero value of relative permeability is specified. Only one B-H curve may be specified per material. In an electromagnetic analysis, the Hall coefficient is specified through the RH material option.

Nodal loads are defined with the **D** and **F** commands. For edge-based analyses, the edge-flux constraint is applied to the node using the **D** command with *Lab* = AZ. Flux-parallel boundary conditions are prescribed by setting AZ to zero. No AZ constraint is required to set flux-normal boundary conditions. The **DFLX** command can be used to impose a uniform magnetic flux on the selected nodes.

For massive conductors (KEYOPT(1) = 1), *Lab* = VOLT is valid with the **D** command and *VALUE* defines the electric potential. Note that electric potential is time-integrated if KEYOPT(2) = 2. With the **F** command, *Lab* = AMPS and *VALUE* corresponds to the total current.

For stranded coils (KEYOPT(1) = 2), *Lab* = VOLT is valid with the **D** command and *VALUE* defines the voltage drop across the coil. The **D** command with *Lab* = EMF can be used to apply constraints on the electromotive force. Note that voltage drop and the electromotive force are time-integrated if KEYOPT(2) = 2. The total current through the coil can be applied with the **F** command using *Lab* = AMPS.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The temperature (for material property evaluation only) body loads may be input based on their value at the element's nodes or as a single element value [**BF**, **BFE**]. In general, unspecified nodal values of temperatures default to the uniform value specified with the **BFUNIF** or **TUNIF** commands.

For modeling stranded conductors KEYOPT(1) = 0, source current density may be applied to an area or volume [**BFE** or **BFV**] or input as an element value [**BFE**]. The vector components of the current density are with respect to the element coordinate system. See "SOLID237 Assumptions and Restrictions" (p. 1244) for a description of the solenoidal condition.

A summary of the element input is given in "SOLID237 Input Summary" (p. 1239). A general description of element input is given in Element Input (p. 43).

SOLID237 Input Summary

Nodes

I, J, K, L, M, N, O, P, Q, R

Degrees of Freedom

See KEYOPT(1)

Real Constants

There are no real constants for KEYOPT(1) = 0 or 1.

The following are the real constants for KEYOPT(1) = 2:

SC, NC, VC, TX, TY, TZ
R, SYM

See Table 237.1: SOLID 237 Real Constants (p. 1241) for more information.

Material Properties

TB command: Magnetic

MP command: MURX, MURY, MURZ, MGXX, MGYY, MGZZ

RSVX, RSVY, RSVZ, RH, PERX, PERY, PERZ (see "SOLID237 Assumptions and Restrictions" (p. 1244))

EMUNIT command: EPZRO, MUZERO

Surface Loads

None

Body Loads

Temperature --

T(I), T(J), ..., T(P), T(Q), T(R)

Source Current Density (valid for KEYOPT(1) = 0 only)

JSX(I), JSY(I), JSZ(I), PHASE(I),

JSX(J), JSY(J), JSZ(J), PHASE(J),

...

JSX(P), JSY(P), JSZ(P), PHASE(P),

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

JSX(Q), JSY(Q), JSZ(Q), PHASE(Q),

JSX(R), JSY(R), JSZ(R), PHASE(R)

Velocity (valid for KEYOPT(1) = 1 only) --

VELOX(I), VELOY(I), VELOZ(I), OMEGAX(I), OMEGAY(I), OMEGAZ(I)

VELOX(J), VELOY(J), VELOZ(J), OMEGAX(J), OMEGAY(J), OMEGAZ(J)

...

VELOX(P), VELOY(P), VELOZ(P), OMEGAX(P), OMEGAY(P), OMEGAZ(P)

VELOX(Q), VELOY(Q), VELOZ(Q), OMEGAX(Q), OMEGAY(Q), OMEGAZ(Q)

VELOX(R), VELOY(R), VELOZ(R), OMEGAX(R), OMEGAY(R), OMEGAZ(R)

Special Features

Linear perturbation

KEYOPT(1)

Element capability and degrees of freedom:

0 --

Magnetic:

AZ

1 --

Electromagnetic:

AZ, VOLT

2 --

Stranded coil:

AZ, VOLT, EMF

KEYOPT(2)

Coupling method between magnetic and electric degrees of freedom (KEYOPT(1) = 1 or 2); also defines the meaning of the VOLT and EMF degrees of freedom for KEYOPT(1) = 1 or 2:

0 --

Strong (matrix) coupling. Produces an unsymmetric matrix. In a linear analysis, a coupled response is achieved after one iteration. Applicable to all analysis types

1 --

Weak (load vector). Produces a symmetric matrix and requires at least two iterations to achieve a coupled response. Applicable to static and transient analyses only. (see "SOLID237 Assumptions and Restrictions" (p. 1244))

2 --

Strong (matrix) coupling with time-integrated electric potential (VOLT) for KEYOPT(1) = 1 (electromagnetic analysis). Produces a symmetric matrix.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Strong (matrix) coupling with time-integrated voltage drop (VOLT) and time-integrated electromotive force (EMF) for KEYOPT(1) = 2 (stranded coil analysis). Produces a symmetric matrix if the coil symmetry factor is 1; produces a nonsymmetric matrix if the coil symmetry factor is greater than 1.

In a linear analysis, a coupled response is achieved after one iteration. Applicable to harmonic and transient analyses only.

KEYOPT(5)

Eddy currents or velocity effects in electromagnetic (KEYOPT(1) =1) analyses:

0 --
Active

1 --
Eddy currents are suppressed in harmonic or transient analyses

2 --
Velocity effects are suppressed

3 --
Eddy currents and velocity effects are suppressed

KEYOPT(7)

Electromagnetic force output:

0 --
At each element node (corner and midside)

1 --
At element corner nodes only (midside nodes forces are condensed to the corner nodes)

KEYOPT(8)

Electromagnetic force calculation:

0 --
Maxwell

1 --
Lorentz

Table 237.1: SOLID 237 Real Constants

No.	Name	Description	Default	Definition
1	SC	Coil cross-sectional area	none	True physical cross-section of the coil regardless of symmetry modeling considerations. It includes the cross-sectional area of the wire and the non-conducting material filling the space between the winding.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

No.	Name	Description	Default	Definition
2	NC	Number of coil turns	1	Total number of winding turns in a coil regardless of any symmetry modeling considerations.
3	VC	Coil volume	none	True physical volume of the coil regardless of symmetry modeling considerations. It includes the volume occupied by the wire and the non-conducting material filling the space between the winding.
4	TX	Coil winding X-directional cosine	0	The coil direction vector T = {TX, TY, TZ}^{T} is a unit vector tangent to the coil winding. It designates the current flow direction.
5	TY	Coil winding Y-directional cosine	1	
6	TZ	Coil winding Z-directional cosine	0	
7	R	Coil resistance	none	Total coil DC resistance regardless of any symmetry modeling considerations.
8	SYM	Coil symmetry factor	1	Ratio of the true physical volume of the coil (real constant VC) to the modeled coil volume. The input should be greater or equal to 1.

SOLID237 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in Table 237.2: SOLID237 Element Output Definitions (p. 1243)

The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 237.2: SOLID237 Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
NODES	Nodes - I, J,…, P, Q, R	-	Y
MAT	Material number	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	-	2
TEMP	Input temperatures T(I), T(J), ..., T(P), T(Q), T(R)	-	Y
LOC	Output location (X, Y, Z)	-	-
B: X, Y, Z, SUM	Magnetic flux density components and vector magnitude	-	1
H: X, Y, Z, SUM	Magnetic field intensity components and vector magnitude	-	1
EF: X, Y, Z, SUM	Electric field intensity components and magnitude [7]	-	1
JC: X, Y, Z, SUM	Nodal conduction current density components and magnitude [7]	-	1
FMAG: X, Y, Z, SUM	Electromagnetic force components and magnitude [3]	-	1
JT: X, Y, Z, SUM	Element conduction current density components (in the global Cartesian coordinate system) and vector magnitude [6]	-	1
JS: X, Y, Z, SUM	Element current density components (in the global Cartesian coordinate system) and vector magnitude [4] [6]	-	1
JHEAT:	Joule heat generation rate per unit volume [3] [5] [6]	-	1
SENE or MENE:	Magnetic energy [3]	-	1
COEN	Magnetic co-energy [3]	-	1
AENE	Apparent magnetic energy [3]	-	1
IENE	Incremental magnetic energy [3]	-	1

1. The solution value is output only if calculated (based upon input data). The element solution is at the centroid.

2. Available only at centroid as a ***GET** item.

3. For a time-harmonic analysis, electromagnetic forces (FMAG), Joule losses (JHEAT) and stored energy (SENE, MENE) represent time-average values. These values are stored in both the real and imaginary data sets. In a linear perturbation analysis, only incremental and apparent energy values are time-averaged.

4. JS represents the sum of element conduction and displacement current densities.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Calculated Joule heat generation rate per unit volume (JHEAT) may be made available for a subsequent thermal analysis with companion elements [**LDREAD**].

6. For the stranded coil analysis option (KEYOPT(1) = 2), JT and JS are the effective current densities as they are calculated based on the coil cross-sectional (SC) that includes the wire and the non-conducting material filling the space between the winding. JHEAT represents the effective Joule heat generation rate per unit volume as it is calculated based on the modeled coil volume that includes the wire and the non-conducting material filling the space between the winding.

7. Not available with the stranded coil option (KEYOPT(1) = 2).

Table 237.3: SOLID237 Item and Sequence Numbers (p. 1244) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 236.3: SOLID236 Item and Sequence Numbers (p. 1234):

Name
output quantity as defined in Table 237.2: SOLID237 Element Output Definitions (p. 1243)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 237.3: SOLID237 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
JTX	NMISC	1
JTY	NMISC	2
JTZ	NMISC	3
JTSUM	NMISC	4

SOLID237 Assumptions and Restrictions

- The element must not have a zero volume or a zero length side. This occurs most frequently when the element is not numbered properly. Elements may be numbered either as shown in Figure 237.1: SOLID237 Geometry (p. 1238) or in an opposite fashion.

- Midside nodes may not be removed.

- The magnetic analysis option (KEYOPT(1) = 0) requires the source current density specified with the **BFE**,,JS command to be solenoidal.

- Permanent magnets are not permitted in a harmonic analysis.

- It is not recommended to use the weak coupling option (KEYOPT(2) = 1) in a transient electromagnetic analysis with eddy currents or a transient stranded coil analysis because multiple iterations may be required to achieve convergence.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- In a transient analysis, the *THETA* integration parameter defaults to the values shown in the following table. You can use the **TINTP** command to modify the default setting.

Table 237.4: THETA Default Values

Analysis Type	KEYOPT Values	THETA Default Value
Strongly coupled transient electromagnetic analysis with electric potential or stranded coil analysis with voltage drop (VOLT)	KEYOPT(1) = 1 or 2 and KEYOPT(2) = 0	1.0
Strongly coupled transient electromagnetic analysis with time-integrated electric potential or stranded coil analysis with time-integrated voltage drop (VOLT)	KEYOPT(1) = 1 or 2 and KEYOPT(2) = 2	0.5

- The electrical permittivity material input (**MP**,PERX, also PERY, PERZ) is applicable to electromagnetic harmonic analyses (KEYOPT(1) = 1) only.

- In a stranded coil (KEYOPT(1) = 2) domain, the winding direction vector T = $\{TX, TY, TZ\}^T$ must be specified in the element coordinate system and all VOLT and EMF degrees of freedom must be coupled (**CP** command).

- This element may not be compatible with other elements having a VOLT degree of freedom. See Element Compatibility in the *Low-Frequency Electromagnetic Analysis Guide*) for more information. The electromagnetic analysis with time-integrated electric potential (KEYOPT(2) = 2) cannot be used with current-based circuit (e.g. CIRCU124) or low-frequency electric (e.g. SOLID232) elements.

SOLID237 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Emag

- The birth and death special feature is not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

PLANE238

2-D 8-Node Diffusion Solid

MP ME <> <> <> <> <> <> PP EME <>

PLANE238 Element Description

PLANE238 is a 2-D, 8-node diffusion element. The element has one degree of freedom, concentration (CONC), at each node. The 8-node elements have compatible concentration shapes and are well suited to model curved boundaries.

This element is applicable to steady-state and transient diffusion analyses. See PLANE238 - 2-D 8-Node Diffusion Solid in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 238.1: PLANE238 Geometry

PLANE238 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 238.1: PLANE238 Geometry (p. 1247). The element is defined by eight nodes and orthotropic material properties. A triangular-shaped element may be formed by defining the same node number for nodes K, L and O.

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems (p. 55). Properties not input by default are as described in Linear Material Properties in the *Mechanical APDL Material Reference*. When saturated concentration (**MP**,CSAT) is specified, the meaning of the CONC degree of freedom is normalized concentration. The actual concentration is evaluated at the element centroid and output as SMISC,1. When unspecified, the saturated concentration (**MP**,CSAT) defaults to 1.0 and the nodal degree of freedom CONC is actual concentration. See Diffusion in the *Mechanical APDL Theory Reference* for more information on the diffusion analysis.

Nodal loads are defined with the **D** (*Lab* = CONC) and **F** (*Lab* = RATE) commands. The nodal forces, if any, should be input per unit of depth for a plane analysis and on a full 360° basis for an axisymmetric analysis.

The temperature (for material property evaluation only) body loads may be input based on their value at the element's nodes or as a single element value [**BF**, **BFE**]. In general, unspecified nodal values of temperatures default to the uniform value specified with the **BFUNIF** or **TUNIF** commands.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A summary of the element input is given in "PLANE238 Input Summary" (p. 1248). A general description of element input is given in Element Input (p. 43). For axisymmetric applications see Harmonic Axisymmetric Elements (p. 83).

PLANE238 Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
CONC

Real Constants
THK - Thickness (used only if KEYOPT(3) = 3)

Material Properties
MP command: DXX, DYY, CSAT

Surface Loads

Diffusion Flux (DFLUX) --
Face 1 (J-I), face 2 (K-J), face 3 (L-K), face 4 (I-L)

Body Loads

Temperature --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

Diffusing Substance Generations (DGEN) --
DG(I),DG(J), DG(K), DG(L), DG(M), DG(N), DG(O), DG(P)

Special Features
Birth and Death

KEYOPT(1)
Diffusion damping matrix:

0 --
Consistent

1 --
Diagonalized

KEYOPT(3)
Element behavior:

0 --
Plane

1 --
Axisymmetric

3 --
Plane with thickness input, specified via real constant THK.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PLANE238 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in Table 238.1: PLANE238 Element Output Definitions (p. 1249).

The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

Figure 238.2: PLANE238 Output

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file. In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 238.1: PLANE238 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC	Location where results are reported	Y	2
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)	Y	Y
LOC	Output location (X, Y)	1	-
CG:X, Y, SUM	Concentration gradient components and vector magnitude	-	1
DF:X, Y SUM	Diffusion flux components and vector magnitude	-	1
CONC	Element concentration [3]	-	1

1. The solution value is output only if calculated (based on input data). The element solution is at the centroid.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Available only at centroid as a ***GET** item.

3. Actual concentration obtained by multiplying the saturated concentration (**MP**,CSAT) and the normalized concentration evaluated at the element centroid. For more information, see Normalized Concentration Approach in the *Mechanical APDL Theory Reference*.

Table 238.2: PLANE238 Item and Sequence Numbers (p. 1250) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Mechanical APDL Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 238.2: PLANE238 Item and Sequence Numbers (p. 1250):

Name
Output quantity as defined in Table 238.1: PLANE238 Element Output Definitions (p. 1249).

Item
Predetermined Item label for **ETABLE** command.

E
Sequence number for single-valued or constant element data.

Table 238.2: PLANE238 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
CONC	SMISC	1

PLANE238 Assumptions and Restrictions

- The area of the element must be positive.
- The element must lie in a global X-Y plane as shown in Table 238.2: PLANE238 Item and Sequence Numbers (p. 1250), and the Y-axis must be the axis of symmetry for axisymmetric analyses.
- An axisymmetric structure should be modeled in the +X quadrants.
- A face with a removed midside node implies that the concentration varies linearly, rather than parabolically, along that face. See Quadratic Elements (Midside Nodes) in the *Mechanical APDL Modeling and Meshing Guide* for more information about the use of midside nodes.
- CSAT cannot be temperature-dependent.

PLANE238 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID239

3-D 20-Node Diffusion Solid

MP ME <> <> <> <> <> <> PP EME <>

SOLID239 Element Description

SOLID239 is a 3-D 20-node diffusion element. The element has a single degree of freedom, concentration (CONC), at each node. The 20-node elements have compatible concentration shapes and are well suited to model curved boundaries.

The 20-node diffusion element is applicable to a steady-state or transient diffusion analysis. See SOLID239 in the SOLID239 - 3-D 20-Node Diffusion Solid in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 239.1: SOLID239 Geometry

SOLID239 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 239.1: SOLID239 Geometry (p. 1251). The element is defined by 20 nodes and the material properties. A prism-shaped element may be formed by defining duplicate K, L, and S; A and B; and O, P, and W node numbers. A tetrahedral-shaped element and a pyramid-shaped element may also be formed as shown in Figure 239.1: SOLID239 Geometry (p. 1251).

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is described in *Coordinate Systems*. Properties not input by default are as described

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in Linear Material Properties in the *Mechanical APDL Material Reference*. When saturated concentration (**MP**,CSAT) is specified, the meaning of the CONC degree of freedom is normalized concentration. The actual concentration is evaluated at the element centroid and output as SMISC,1. When unspecified, the saturated concentration (**MP**,CSAT) defaults to 1.0 and the nodal degree of freedom CONC is actual concentration. See Diffusion in the *Mechanical APDL Theory Reference* for more information on the diffusion analysis.

Nodal loads are defined with the **D** (*Lab* = CONC) and **F** (*Lab* = RATE) commands. The temperature (for material property evaluation only) body loads may be input based on their value at the element's nodes or as a single element value [**BF**, **BFE**]. In general, unspecified nodal values of temperatures default to the uniform value specified with the **BFUNIF** or **TUNIF** commands.

A summary of the element input is given in "SOLID239 Input Summary" (p. 1252). A general description of element input is given in Element Input (p. 43).

SOLID239 Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B

Degrees of Freedom
CONC

Real Constants
None

Material Properties
MP command: DXX, DYY, DZZ, CSAT

Surface Loads

Diffusion Flux (DFLUX) --
Face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N), face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperature --
T(I), T(J), ..., T(Z), T(A), T(B)

Diffusing Substance Generations (DGEN) --
DG(I), DG(J), ..., DG(A), DG(B)

Special Features
Birth and Death

KEYOPT(1)
Diffusion damping matrix:

0 --
Consistent

1 --
Diagonalized

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID239 Output Data

The solution output associated with the element is in two forms:

- Nodal concentrations included in the overall nodal solution
- Additional element output as shown in Table 239.1: SOLID239 Element Output Definitions (p. 1253)

The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file. In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 239.1: SOLID239 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	1	2
TEMP	Input temperatures T(I), T(J), ..., T(Z), T(A), T(B)	Y	Y
CG:X, Y, Z, SUM	Diffusion gradient components and vector magnitude	-	1
DF:X, Y, Z, SUM	Diffusion flux components and vector magnitude	-	1
CONC	Element concentration [3]	-	1

1. The solution value is output only if calculated (based on input data). The element solution is at the centroid.
2. Available only at centroid as a ***GET** item.
3. Actual concentration obtained by multiplying the saturated concentration (**MP**,CSAT) and the normalized concentration evaluated at the element centroid. For more information, see Normalized Concentration Approach in the *Mechanical APDL Theory Reference*.

Table 239.2: SOLID239 Item and Sequence Numbers (p. 1254) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Mechanical APDL Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) of this manual for more information. The following notation is used in Table 239.2: SOLID239 Item and Sequence Numbers (p. 1254):

Name

Output quantity as defined in Table 239.1: SOLID239 Element Output Definitions (p. 1253).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Item

Predetermined Item label for **ETABLE** command.

E

Sequence number for single-valued or constant element data.

Table 239.2: SOLID239 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
CONC	SMISC	1

SOLID239 Assumptions and Restrictions

- The element must not have a zero volume or a zero length side. This occurs most frequently when the element is not numbered properly. Elements may be numbered either as shown in Figure 239.1: SOLID239 Geometry (p. 1251) or in an opposite fashion.
- An edge with a removed midside node implies that the concentration varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Mechanical APDL Modeling and Meshing Guide* for more information on the use of midside nodes.
- Degeneration to the form of the pyramid should be used with caution. When degenerated, the element sizes should be small in order to minimize the field gradients. Pyramid elements are best used as filler elements in meshing transition zones.
- CSAT cannot be temperature-dependent.

SOLID239 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID240

3-D 10-Node Tetrahedral Diffusion Solid

MP ME <> <> <> <> <> <> PP EME <>

SOLID240 Element Description

SOLID240 is a 3-D, 10-node diffusion element. It is well suited to model irregular meshes (such as produced from various CAD/CAM systems). The element has one degree of freedom, concentration (CONC), at each node.

This element is applicable to the steady-state and transient diffusion analyses. See SOLID240 - 3-D 10-Node Tetrahedral Diffusion Solid in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 240.1: SOLID240 Geometry

SOLID240 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 238.1: PLANE238 Geometry (p. 1247). The element is defined by 10 nodes and the material properties.

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is described in Coordinate Systems (p. 55). Properties not input by default are as described in Linear Material Properties in the *Mechanical APDL Material Reference*. When saturated concentration (**MP**,CSAT) is specified, the meaning of the CONC degree of freedom is normalized concentration. The actual concentration is evaluated at the element centroid and output as SMISC,1. When unspecified, the saturated concentration (**MP**,CSAT) defaults to 1.0 and the nodal degree of freedom CONC is actual concentration. See Diffusion in the *Mechanical APDL Theory Reference* for more information on the diffusion analysis.

Nodal loads are defined with the **D** (*Lab* = CONC) and **F** (*Lab* = RATE) commands. The temperature (for material property evaluation only) body loads may be input based on their value at the element's nodes or as a single element value [**BF**, **BFE**]. In general, unspecified nodal values of temperatures default to the uniform value specified with the **BFUNIF** or **TUNIF** commands.

A summary of the element input is given in "SOLID240 Input Summary" (p. 1256). A general description of element input is given in Element Input (p. 43).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID240 Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R

Degrees of Freedom
CONC

Real Constants
None

Material Properties
MP command: DXX, DYY, DZZ, CSAT

Surface Loads

Diffusion Flux (DFLUX) --
Face 1 (J-I-K), face 2 (I-J-L), face 3 (J-K-L), face 4 (K-I-L)

Body Loads

Temperature --
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P), T(Q), T(R)

Diffusing Substance Generations (DGEN) --
DG(I),DG(J), ..., DG(Q), DG(R)

Special Features
Birth and Death

KEYOPT(1)
Diffusion damping matrix:

0 --
Consistent

1 --
Diagonalized

SOLID240 Output Data

The solution output associated with the element is in two forms:

- Nodal concentrations included in the overall nodal solution
- Additional element output as shown in Table 240.1: SOLID240 Element Output Definitions (p. 1257)

The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 240.1: SOLID240 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P, Q, R	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	2
TEMP	Temperatures T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P), T(Q), T(R)	Y	Y
LOC	Output location (X, Y, Z)	1	-
CG:X, Y, Z, SUM	Concentration gradient components and vector magnitude	-	1
DF:X, Y, Z, SUM	Diffusion flux components and vector magnitude	-	1
CONC	Element concentration [3]	-	1

1. The solution value is output only if calculated (based on input data). The element solution is at the centroid.

2. Available only at centroid as a ***GET** item.

3. Actual concentration obtained by multiplying the saturated concentration (**MP**,CSAT) and the normalized concentration evaluated at the element centroid. For more information, see the Normalized Concentration Approach in the *Mechanical APDL Theory Reference*.

Table 240.2: SOLID240 Item and Sequence Numbers (p. 1257) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Mechanical APDL Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 240.2: SOLID240 Item and Sequence Numbers (p. 1257):

Name
Output quantity as defined in Table 240.1: SOLID240 Element Output Definitions (p. 1257).

Item
Predetermined Item label for **ETABLE** command.

E
Sequence number for single-valued or constant element data.

Table 240.2: SOLID240 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
CONC	SMISC	1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID240 Assumptions and Restrictions

- The element must not have a zero volume or a zero length side. This occurs most frequently when the element is not numbered properly. Elements may be numbered either as shown in Figure 240.1: SOLID240 Geometry (p. 1255) or in an opposite fashion.
- An edge with a removed midside node implies that the concentration varies linearly, rather than parabolically, along that edge. See Quadratic Elements (Midside Nodes) in the *Mechanical APDL Modeling and Meshing Guide* for more information on the use of midside nodes.
- CSAT cannot be temperature-dependent.

SOLID240 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HSFLD241

2-D Hydrostatic Fluid

MP ME <> <> <> <> <> <> PP EME MFS
Product Restrictions

HSFLD241 Element Description

HSFLD241 is used to model fluids that are fully enclosed by solids (containing vessels). The hydrostatic fluid element is well suited for calculating fluid volume and pressure for coupled problems involving fluid-solid interaction. The pressure in the fluid volume is assumed to be uniform (no pressure gradients), so sloshing effects cannot be included. Temperature effects and compressibility may be included, but fluid viscosity cannot be included.

Hydrostatic fluid elements are overlaid on the faces of 2-D solid elements enclosing the fluid volume. See HSFLD241 in the *Mechanical APDL Theory Reference* for more details about this element. See HSFLD242 for a 3-D version of this element.

Figure 241.1: HSFLD241 Geometry

HSFLD241 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 241.1: HSFLD241 Geometry (p. 1259). The hydrostatic fluid element is defined by three or four nodes. Nodes I, J and K on the surface (face 1) are shared with the solid element and have two degrees of freedom at each node: translation in x and y directions. Node K is not used if the underlying solid element does not have a midside node. For the case of a degenerate solid element sharing the collapsed face with the hydrostatic fluid element, the surface nodes of the hydrostatic fluid element will be coincident. Node L is a pressure node with a hydrostatic pressure degree of freedom. Hydrostatic fluid elements can be generated automatically using the **ESURF** command.

The pressure node (L) can be located anywhere in the fluid volume, except when the fluid volume has symmetry boundaries; in this case the pressure node must be located on the symmetry line or on the intersection point of multiple symmetry lines. The pressure node is automatically moved to the centroid of the fluid volume if there are no displacement degree-of-freedom constraints specified. To keep the pressure node on a symmetry line, you must specify symmetry boundary conditions at this node. (The displacement degrees of freedom at the pressure node do not have any displacement solution associated with them. They are only available for applying displacement degree of freedom constraints.) The pressure node is shared by all the hydrostatic fluid elements used to define the fluid volume. It is also used to apply temperature loads, fluid mass flow rate, or hydrostatic pressure degree-of-freedom constraints for the fluid.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can define a hydrostatic fluid element without an underlying solid element in situations where the underlying solid has a discontinuity. In this case, the surface nodes (I and J) of the hydrostatic fluid element must be shared with adjacent solid elements, or the displacement degrees of freedom at the surface nodes should be constrained. For example, the gap between the cylinder and piston in a cylinder-piston assembly with fluid may be bridged with a hydrostatic fluid element by sharing one of its surface nodes (I) with a solid element on the cylinder and its other surface node (J) with a solid element on the piston (see Example Model Using Hydrostatic Fluid Elements in the *Structural Analysis Guide*).

You can define material properties for hydrostatic fluid elements using **MP** or **TB** commands. All hydrostatic fluid elements sharing a pressure node must use the same material property definition and must have the same real constant values (THK and PREF).

You can input element thickness and reference pressure (must be specified for compressible gas defined via **TB** command with *Lab* = FLUID and *TBOPT* = GAS) as real constants THK and PREF. The THK value is used to calculate element volume.

You can define the initial state of the hydrostatic fluid by defining initial pressure (input via the **IC** command with *Lab* = HDSP) at the pressure node. Specify the reference temperature by using the **TREF** command or the **MP**,REFT command. For compressible gas (defined via the **TB** command with *Lab* = FLUID and *TBOPT* = GAS), the initial pressure and the reference pressure (input as real constant PREF) are added internally to get the total initial pressure for the Ideal Gas Law. Internal force corresponding to initial pressure is calculated internally and applied over the first load step.

You can prescribe uniform pressure for the fluid as a hydrostatic pressure degree-of-freedom constraint at the pressure node (input on **D** command with *Lab* = HDSP). The change in hydrostatic pressure value is assumed to occur as a result of the addition or removal of fluid mass to or from the containing vessel. Applying a hydrostatic pressure degree-of-freedom constraint is equivalent to applying a surface load on the underlying solid element surface. Element loads are described in Nodal Loading (p. 49). You can apply fluid mass flow rate as a load on the pressure node (input via **F** command with *Lab* = DVOL); a positive value indicates fluid mass flowing into the containing vessel. You can also input fluid temperature as an element body load at the pressure node (input via **BF** command with *Lab* = TEMP). The nodal temperature defaults to **TUNIF**.

You can model fluid flow between two fluid volumes in two containing vessels by using FLUID116 coupled thermal-fluid pipe elements to connect the pressure nodes of the fluid volumes. You must activate the PRES degree of freedom (KEYOPT(1) = 1) on pressure nodes of the hydrostatic fluid elements. In addition, you must set KEYOPT(1) = 3 on the FLUID116 element.

KEYOPT(1) defines degrees of freedom for the hydrostatic fluid element:

Use KEYOPT(1) = 0 (default) to activate UX and UY degrees of freedom on the surface nodes (I, J, K) and HDSP degree of freedom on the pressure node (L).
Use KEYOPT(1) = 1 to activate UX, and UY degrees of freedom on surface nodes (I, J, K) and HDSP and PRES degrees of freedom on the pressure node (L). You must activate the PRES degree of freedom if the pressure node of the hydrostatic fluid element is shared with a coupled thermal-fluid pipe (FLUID116) element to model fluid flow.

KEYOPT(3) defines the hydrostatic fluid element behavior:

Use KEYOPT(3) = 0 (default) to model planar behavior. The choice of plane stress or plane strain is made automatically based on the attached solid element.
Use KEYOPT(3) = 1 to model 2-D axisymmetric behavior.

KEYOPT(5) specifies how mass is computed for the hydrostatic fluid element:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use KEYOPT(5) = 0 (default) to ignore the mass contribution from the fluid element. However, you can attach MASS21 elements to the nodes of the underlying 2-D solid elements to account for the fluid mass.
Use KEYOPT(5) = 1 to distribute the fluid element mass to the surface nodes (I, J, K) based on the volume of the fluid element. No mass is added to the surface nodes if the volume of the fluid element becomes negative.
Use KEYOPT(5) = 2 to distribute the fluid element mass to the surface nodes (I, J, K) based on the ratio of element surface area (area of face 1) to the total fluid surface area.

KEYOPT(6) defines the hydrostatic fluid element compressibility:

Use KEYOPT(6) = 0 (default) to model the hydrostatic fluid element as compressible. You need to define a fluid material property (use the **TB** command with *Lab* = FLUID) to relate changes in fluid pressure to fluid volume.
Use KEYOPT(6) = 1 to model the hydrostatic fluid element as incompressible. The fluid volume is kept constant, even as the solid enclosing the fluid undergoes large deformations. The fluid volume, however, can change when fluid mass is added to or taken out of the containing vessel; this is achieved by applying a fluid mass flow rate or by prescribing a non-zero hydrostatic pressure degree-of-freedom constraint at the pressure node. The fluid volume can also change when a temperature load is applied at the pressure node for a fluid with a non-zero coefficient of thermal expansion.

You can define contact and target surfaces on the attached 2-D solid elements to model self-contact between walls of the containing vessel after the fluid has been removed. Note that contact should not cause a single fluid region to be separated into two since the pressure-volume calculations are performed assuming a single cavity.

For more information on using hydrostatic fluid elements to model fluids enclosed by solids, see Modeling Hydrostatic Fluids in the *Structural Analysis Guide*.

"HSFLD241 Input Summary" (p. 1261) contains a summary of the element input. See Element Input (p. 43) in this document for a general description of element input.

HSFLD241 Input Summary

Nodes

I, J, K, L

Degrees of Freedom

UX, UY for surface nodes (I, J, K)

HDSP and PRES for pressure node (L)

Real Constants

THK - Thickness

PREF - Reference pressure for compressible gas defined via **TB** command with *Lab* = FLUID and *TBOPT* = GAS

Material Properties

TB command: Fluid

MP command: ALPX, DENS, ALPD

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface Loads

None

Body Loads

Temperatures --

T(L)

Special Features

Large deflection
Linear perturbation
Nonlinearity

KEYOPT(1)

Degrees of freedom:

0 --

UX and UY degrees of freedom at surface nodes, HDSP degree of freedom at pressure node (default)

1 --

UX and UY degrees of freedom at surface nodes, HDSP and PRES degrees of freedom at pressure node

KEYOPT(3)

Element behavior:

0 --

Planar (default)

1 --

Axisymmetric

KEYOPT(5)

Fluid mass:

0 --

No fluid mass (default)

1 --

Fluid mass calculated based on the volume of the fluid element

2 --

Fluid mass calculated based on the surface area of the fluid element

KEYOPT(6)

Fluid compressibility:

0 --

Compressible (default)

1 --

Incompressible

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HSFLD241 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in Table 241.1: HSFLD241 Element Output Definitions (p. 1263)

A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 241.1: HSFLD241 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L	Y	Y
MAT	Material number	Y	Y
AREA	Element surface area (face 1)	Y	Y
VOLU	Element volume	Y	Y
XC, YC	Location on the surface (face 1) where results are reported	Y	1
DENSITY	Fluid density	Y	Y
TEMP	Temperature at nodes: T(I), T(J), T(K), T(L)	Y	Y
TVOL	Total volume of the fluid in the containing vessel	2	2
TMAS	Total mass of the fluid in the containing vessel	3	3
MFLO	Fluid mass flow rate	4	4
TVOLO	Total original volume of the fluid in the containing vessel	2	2

1. Available only at centroid as a ***GET** item.
2. Elements that share a pressure node have the same TVOL and TVOLO output value.
3. Elements that share a pressure node have the same TMAS output value.
4. Elements that share a pressure node have the same MFLO output value.

Table 241.2: HSFLD241 Item and Sequence Numbers (p. 1264) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic*

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Analysis Guide and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 241.2: HSFLD241 Item and Sequence Numbers (p. 1264):

Name
output quantity as defined in Table 241.1: HSFLD241 Element Output Definitions (p. 1263)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

Table 241.2: HSFLD241 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
DENSITY	NMISC	1
AREA	NMISC	2
TVOL	NMISC	3
TMAS	NMISC	4
MFLO	NMISC	5
TVOLO	NMISC	6

HSFLD241 Assumptions and Restrictions

- The fluid volume has no free surface; it is completely enclosed by the solid (containing vessel).
- The fluid volume has uniform pressure, temperature, and density without any gradients.
- All elements used to define a fluid volume share a pressure node with a hydrostatic pressure degree of freedom.
- The pressure node can be located anywhere within the fluid volume; it is automatically moved to the centroid of the fluid volume if there are no displacement degree-of-freedom constraints specified. However, if the fluid volume is bounded by one or more symmetry lines, the pressure node must be on the symmetry line or intersecting corner of multiple symmetry lines, and it must have symmetry boundary conditions.
- The fluid may be modeled as incompressible or compressible without any viscosity.
- The PRES degree of freedom must be active (KEYOPT(1) = 1) on the pressure node of the hydrostatic fluid element to model fluid flow with FLUID116 coupled thermal-fluid pipe elements. In this case, the PRES (pressure) and HDSP (hydrostatic pressure) degrees of freedom are made to be the same at the pressure node.
- Inertial effects such as sloshing cannot be included, but fluid mass can be added to the surface nodes (I, J, K) shared with the underlying 2-D solid by using KEYOPT(5).
- This element can be used in linear and nonlinear static and transient analyses and modal analyses.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HSFLD241 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

HSFLD242

3-D Hydrostatic Fluid

MP ME <> <> <> <> <> <> PP EME MFS
Product Restrictions

HSFLD242 Element Description

HSFLD242 is used to model fluids that are fully enclosed by solids (containing vessels). The hydrostatic fluid element is well suited for calculating fluid volume and pressure for coupled problems involving fluid-solid interaction. The pressure in the fluid volume is assumed to be uniform (no pressure gradients), so sloshing effects cannot be included. Temperature effects and compressibility may be included, but fluid viscosity cannot be included.

Hydrostatic fluid elements are overlaid on the faces of 3-D solid or shell elements enclosing the fluid volume. See HSFLD242 in the *Mechanical APDL Theory Reference* for more details about this element. See HSFLD241 for a 2-D version of this element.

Figure 242.1: HSFLD242 Geometry

HSFLD242 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 242.1: HSFLD242 Geometry (p. 1267). The hydrostatic fluid element is defined by five to nine nodes. Nodes I, J, K, L, M, N, O, and P on the surface (face 1) are shared with the solid or shell element and have three degrees of freedom at each node: translation in x, y, and z directions. Nodes M, N, O, and P are not used if the underlying solid or shell element does not have midside nodes. For the case of a degenerate solid or shell element sharing the collapsed face with the hydrostatic fluid element, some of the surface nodes of the hydrostatic fluid element will be coincident. Node Q is a pressure node with a hydrostatic pressure degree of freedom. Hydrostatic fluid elements can be generated automatically using the **ESURF** command.

The pressure node (Q) can be located anywhere in the fluid volume, except when the fluid volume has symmetry boundaries; in this case the pressure node must be located on the symmetry plane or on the intersection point or edge of multiple symmetry planes. The pressure node is automatically moved to the centroid of the fluid volume if there are no displacement degree-of-freedom constraints specified. To keep the pressure node on a symmetry line, you must specify symmetry boundary conditions at this node. (The displacement degrees of freedom at the pressure node do not have any displacement solution associated with them. They are only available for applying displacement degree of freedom constraints.) The pressure node is shared by all the hydrostatic fluid elements used to define the fluid

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

volume. It is also used to apply temperature loads, fluid mass flow rate, or hydrostatic pressure degree-of-freedom constraints for the fluid.

You can define a hydrostatic fluid element without an underlying solid or shell element in situations where the underlying solid has a discontinuity. In this case, the surface nodes (I through P) of the hydrostatic fluid element must be shared with adjacent solid or shell elements, or the displacement degrees of freedom at the surface nodes should be constrained. For example, the gap between the cylinder and piston in a cylinder-piston assembly with fluid may be bridged with a hydrostatic fluid element by sharing some of its surface nodes with a solid or shell element on the cylinder and sharing its other surface nodes with a solid or shell element on the piston (see Example Model Using Hydrostatic Fluid Elements in the *Structural Analysis Guide*).

You can define material properties for hydrostatic fluid elements using **MP** or **TB** commands. All hydrostatic fluid elements sharing a pressure node must use the same material property definition and must have the same real constant values (PREF).

You can input a reference pressure (must be specified for compressible gas defined via **TB** command with *Lab* = FLUID and *TBOPT* = GAS) as real constant PREF.

You can define the initial state of the hydrostatic fluid by defining initial pressure (input via the **IC** command with *Lab* = HDSP) at the pressure node. Specify the reference temperature by using the **TREF** command or the **MP**,REFT command. For compressible gas (defined via the **TB** command with *Lab* = FLUID and *TBOPT* = GAS), the initial pressure and the reference pressure (input as real constant PREF) are added internally to get the total initial pressure for the Ideal Gas Law. Internal force corresponding to initial pressure is calculated internally and applied over the first load step.

You can prescribe uniform pressure for the fluid as a hydrostatic pressure degree-of-freedom constraint at the pressure node (input via the **D** command with *Lab* = HDSP). The change in hydrostatic pressure value is assumed to occur as a result of the addition or removal of fluid mass to or from the containing vessel. Applying a hydrostatic pressure degree-of-freedom constraint is equivalent to applying a surface load on the underlying solid or shell element surface. Element loads are described in Nodal Loading (p. 49). You can apply a fluid mass flow rate as a load on the pressure node (input via the **F** command with *Lab* = DVOL); a positive value indicates fluid mass flowing into the containing vessel. You can also input fluid temperature as an element body load at the pressure node (input via the **BF** command with *Lab* = TEMP). The nodal temperature defaults to **TUNIF**.

You can model fluid flow between two fluid volumes in two containing vessels by using FLUID116 coupled thermal-fluid pipe elements to connect the pressure nodes of the fluid volumes. You must activate the PRES degree of freedom (KEYOPT(1) = 1) on pressure nodes of the hydrostatic fluid elements. In addition, you must set KEYOPT(1) = 3 on the FLUID116 element.

KEYOPT(1) defines degrees of freedom for the hydrostatic fluid element:

Use KEYOPT(1) = 0 (default) to activate UX, UY, and UZ degrees of freedom on the surface nodes (I through P) and HDSP degree of freedom on the pressure node (Q).
Use KEYOPT(1) = 1 to activate UX, UY, and UZ degrees of freedom on surface nodes (I through P) and HDSP and PRES degrees of freedom on the pressure node (Q). You must activate the PRES degree of freedom if the pressure node of the hydrostatic fluid element is shared with a coupled thermal-fluid pipe (FLUID116) element to model fluid flow. In this case, you must also set KEYOPT(1) = 3 on the FLUID116 element.

KEYOPT(5) specifies how mass is computed for the hydrostatic fluid element:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use KEYOPT(5) = 0 (default) to ignore the mass contribution from the fluid element. However, you can attach MASS21 elements to the nodes of the underlying 3-D solid or shell elements to account for the fluid mass.
Use KEYOPT(5) = 1 to distribute the fluid element mass to the surface nodes (I through P) based on the volume of the fluid element. No mass is added to the surface nodes if the volume of the fluid element becomes negative.
Use KEYOPT(5) = 2 to distribute the fluid element mass to the surface nodes (I through P) based on the ratio of element surface area (area of face 1) to the total fluid surface area.

KEYOPT(6) defines the hydrostatic fluid element compressibility:

Use KEYOPT(6) = 0 (default) to model the hydrostatic fluid element as compressible. You need to define a fluid material property (use the **TB** command with *Lab* = FLUID) to relate changes in fluid pressure to fluid volume.
Use KEYOPT(6) = 1 to model the hydrostatic fluid element as incompressible. The fluid volume is kept constant, even as the solid enclosing the fluid undergoes large deformations. The fluid volume, however, can change when fluid mass is added to or taken out of the containing vessel; this is achieved by applying a fluid mass flow rate or by prescribing a non-zero hydrostatic pressure degree-of-freedom constraint at the pressure node. The fluid volume can also change when a temperature load is applied at the pressure node for a fluid with a non-zero coefficient of thermal expansion.

You can define contact and target surfaces on the underlying 3-D solid or shell elements to model self-contact between walls of the vessel after the fluid has been removed. Note that contact should not cause a single fluid region to be separated into two since the pressure-volume calculations are performed assuming a single cavity.

For more information on using hydrostatic fluid elements to model fluids enclosed by solids, see Modeling Hydrostatic Fluids in the *Structural Analysis Guide*.

"HSFLD242 Input Summary" (p. 1269) contains a summary of the element input. See Element Input (p. 43) in this document for a general description of element input.

HSFLD242 Input Summary

Nodes

I, J, K, L, M, N, O, P, Q

Degrees of Freedom

UX, UY, UZ for surface nodes (I, J, K, L, M, N, O, P)

HSDP and PRES for pressure node (Q)

Real Constants

PREF - Reference pressure for compressible gas defined via **TB** command with *Lab* = FLUID and *TBOPT* = GAS

Material Properties

TB command: Fluid

MP command: ALPX, DENS, ALPD

Surface Loads

None

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Body Loads

Temperatures --
T(Q)

Special Features

Large deflection
Linear perturbation
Nonlinearity

KEYOPT(1)
Degrees of freedom:

0 --
UX, UY, and UZ degrees of freedom at surface nodes, HDSP degree of freedom at pressure node (default)

1 --
UX, UY, and UZ degrees of freedom at surface nodes, HDSP and PRES degrees of freedom at pressure node

KEYOPT(5)
Fluid mass:

0 --
No fluid mass (default)

1 --
Fluid mass calculated based on the volume of the fluid element

2 --
Fluid mass calculated based on the surface area of the fluid element

KEYOPT(6)
Fluid compressibility:

0 --
Compressible (default)

1 --
Incompressible

HSFLD242 Output Data

The solution output associated with the element is in two forms:

- Nodal degrees of freedom included in the overall nodal solution
- Additional element output as shown in Table 242.1: HSFLD242 Element Output Definitions (p. 1271)

A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 242.1: HSFLD242 Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
AREA	Element surface area (face 1)	Y	Y
VOLU	Element volume	Y	Y
XC, YC, ZC	Location on the surface (face 1) where results are reported	Y	1
DENSITY	Fluid density	Y	Y
TEMP	Temperature at nodes: T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P), T(Q)	Y	Y
TVOL	Total volume of the fluid in the containing vessel	2	2
TMAS	Total mass of the fluid in the containing vessel	3	3
MFLO	Fluid mass flow rate	4	4
TVOLO	Total original volume of the fluid in the containing vessel	2	2

1. Available only at centroid as a ***GET** item.

2. Elements that share a pressure node have the same TVOL and TVOLO output values.

3. Elements that share a pressure node have the same TMAS output value.

4. Elements that share a pressure node have the same MFLO output value.

Table 242.2: HSFLD242 Item and Sequence Numbers (p. 1272) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 242.2: HSFLD242 Item and Sequence Numbers (p. 1272):

Name
output quantity as defined in Table 242.1: HSFLD242 Element Output Definitions (p. 1271)

Item
predetermined Item label for **ETABLE** command

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

E

sequence number for single-valued or constant element data

Table 242.2: HSFLD242 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
DENSITY	NMISC	1
AREA	NMISC	2
TVOL	NMISC	3
TMAS	NMISC	4
MFLO	NMISC	5
TVOLO	NMISC	6

HSFLD242 Assumptions and Restrictions

- The fluid volume has no free surface; it is completely enclosed by the solid (containing vessel).
- The fluid volume has uniform pressure, temperature, and density without any gradients.
- All elements used to define a fluid volume share a pressure node with a hydrostatic pressure degree of freedom.
- The pressure node can be located anywhere within the fluid volume; it is automatically moved to the centroid of the fluid volume if there are no displacement degree-of-freedom constraints specified. However, if the fluid volume is bounded by one or more symmetry lines, the pressure node must be on the symmetry plane, intersecting point, or edge of multiple symmetry planes, and it must have symmetry boundary condition.
- The fluid may be modeled as incompressible or compressible without any viscosity.
- The PRES degree of freedom must be active (KEYOPT(1) = 1) on the pressure node of the hydrostatic fluid element to model fluid flow with FLUID116 coupled thermal-fluid pipe elements. In this case, the PRES (pressure) and HDSP (hydrostatic pressure) degrees of freedom are made to be the same at the pressure node.
- Inertial effects such as sloshing cannot be included, but fluid mass can be added to the surface nodes (I through P) shared with the underlying 3-D solid or shell elements by using KEYOPT(5).
- This element can be used in linear and nonlinear static and transient analyses and modal analyses.

HSFLD242 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SURF251

2-D Radiosity Surface

MP ME <> PR PRN <> <> <> PP EME MFS
Product Restrictions

SURF251 Element Description

SURF251 is used for radiation surface loads and can be used only with the radiosity solver method. It can be overlaid onto a face of any 2-D thermal solid element that supports temperature DOF. This element is applicable to 2-D thermal analyses (planar or axisymmetric). Various other loads and surface effects may exist simultaneously (e.g., SURF151 and SURF153 and SURF251 may be applied on the same solid element faces to support convection heat flux and radiation heat flux loads).

This element can be created only by the **RSURF** command. The underlying solid surface must also have the RDSF flag.

Figure 251.1: SURF251 Geometry

SURF251 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 251.1: SURF251 Geometry (p. 1273). The element is defined by two nodes, regardless of the underlying solid element.

You would typically generate SURF251 elements via the **RSURF** command, creating elements which are coincident with the solid element surface. However, if you are using decimation (**RDEC**), then the surface elements created will not coincide with the underlying solid element topology. See Figure 251.2: SURF251 Elements Without Coincident Nodes (p. 1274). Symmetrical SURF251 elements (produced when using the symmetry options [**RSYMM**]) can have no underlying solid elements. The **RSURF** command always produces extra nodes to define the SURF251 topology as shown in Figure 251.2: SURF251 Elements Without Coincident Nodes (p. 1274), regardless if **RDEC** is used.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 251.2: SURF251 Elements Without Coincident Nodes

You cannot apply any loads on this element. During solution, the element extracts the temperature of the solid element and computes the radiation heat flux, which is transferred back as a surface load to the solid element.

The next table summarizes the element input. Element Input (p. 43) gives a general description of element input.

SURF251 Input Summary

Element Name
SURF251

Nodes
I, J

Degrees of Freedom
None

Real Constants
None

Material Properties
None

Surface Loads
None

Body Loads
None

Special Features
None

KEYOPTS
None

SURF251 Output Data

Table 251.1: SURF251 Item and Sequence Numbers (p. 1275) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic*

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Analysis Guide and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 151.3: SURF151 Item and Sequence Numbers (p. 516):

Name
output quantity as defined in Table 251.1: SURF251 Item and Sequence Numbers (p. 1275)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J
sequence number for data at nodes I and J

Table 251.1: SURF251 Item and Sequence Numbers

Name	Item	E
CENTROID X	NMISC	1
CENTROID Y	NMISC	2
CENTROID Z	NMISC	3
AREA	NMISC	4
TEMP	NMISC	5
EMISSIVITY	NMISC	6
Net radiation heat flux	NMISC	7
Emitted radiation heat flux	NMISC	8
Reflected radiation heat flux	NMISC	9
Incident radiant heat flux	NMISC	10
Enclosure No.	NMISC	18

The net radiation heat flux is the sum of the directly emitted radiation flux [$\varepsilon\sigma T^4$] plus the reflected radiation flux [$(1-\varepsilon)q_i$] minus the incoming radiation [q_i], as shown in Figure 251.3: Net Radiation Heat Flux (p. 1276).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 251.3: Net Radiation Heat Flux

Net outgoing radiant heat flux = $\underbrace{\varepsilon\sigma T^4}_{\text{emitted}}+\underbrace{(1-\varepsilon)q_i}_{\text{reflected}}-\underbrace{q_i}_{\text{incident}}$

SURF251 Assumptions and Restrictions

- The element must not have a zero length.

SURF251 Product Restrictions

None

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SURF252

3-D Radiosity Surface

MP ME <> PR PRN <> <> <> PP EME MFS
Product Restrictions

SURF252 Element Description

SURF252 is used for radiation surface loads and can be used only with the radiosity solver method. It can be overlaid onto a face of any 3-D thermal solid or shell element that supports temperature DOF. This element is applicable to 3-D thermal analyses. Various other loads and surface effects may exist simultaneously (e.g., SURF152 and SURF154 and SURF252 may be applied on the same solid element faces to support convection heat flux and radiation heat flux loads).

This element can be created only by the **RSURF** command. The surface must also have the RDSF flag.

Figure 252.1: SURF252 Geometry

SURF252 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 252.1: SURF252 Geometry (p. 1277). The element is defined by three or four nodes, regardless of the underlying solid element.

You would typically generate SURF252 elements via the **RSURF** command, creating elements which are coincident with the solid element surface. However, if you are using decimation (**RDEC**), then the surface elements created will not coincide with the underlying solid element topology. See Figure 252.2: SURF252 Elements Without Coincident Nodes (p. 1278). Symmetrical SURF252 elements (produced when using the symmetry options [**RSYMM**]) will have no underlying solid elements. The **RSURF** command always produces extra nodes to define the SURF252 topology as shown in Figure 252.2: SURF252 Elements Without Coincident Nodes (p. 1278), regardless if **RDEC** is used.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 252.2: SURF252 Elements Without Coincident Nodes

You cannot apply any loads on this element. During solution, the element extracts the temperature of the solid element and computes the radiation heat flux, which is transferred back as a surface load to the solid element.

The next table summarizes the element input. Element Input (p. 43) gives a general description of element input.

SURF252 Input Summary

Element Name
SURF252

Nodes
I, J, K, L

Degrees of Freedom
None

Real Constants
None

Material Properties
None

Surface Loads
None

Body Loads
None

Special Features
None

KEYOPTS
None

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SURF252 Output Data

Table 252.1: SURF252 Item and Sequence Numbers (p. 1279) lists output available through the **ETABLE** command using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 151.3: SURF151 Item and Sequence Numbers (p. 516):

Name
output quantity as defined in Table 252.1: SURF252 Item and Sequence Numbers (p. 1279)

Item
predetermined Item label for **ETABLE** command

E
sequence number for single-valued or constant element data

I,J
sequence number for data at nodes I and J

Table 252.1: SURF252 Item and Sequence Numbers

Name	Item	E
CENTROID X	NMISC	1
CENTROID Y	NMISC	2
CENTROID Z	NMISC	3
AREA	NMISC	4
TEMP	NMISC	5
EMISSIVITY	NMISC	6
Net radiation heat flux	NMISC	7
Emitted radiation heat flux	NMISC	8
Reflected radiation heat flux	NMISC	9
Incident radiant heat flux	NMISC	10
Enclosure No.	NMISC	18

The net radiation heat flux is the sum of the directly emitted radiation flux [$\varepsilon\sigma T^4$] plus the reflected radiation flux [$(1-\varepsilon)q_i$] minus the incoming radiation [q_i], as shown in Figure 252.3: Net Radiation Heat Flux (p. 1280).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 252.3: Net Radiation Heat Flux

SURF252 Assumptions and Restrictions

- The element must not have a zero length.

SURF252 Product Restrictions

None

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

REINF263

2-D Smeared Reinforcing

MP ME ST PR PRN <> <> <> PP EME MFS
Product Restrictions

REINF263 Element Description

Use REINF263 with standard 2-D solid and shell elements (referred to here as the *base elements*) to provide extra reinforcing to those elements.

The element uses a smeared approach and is suitable for modeling evenly spaced reinforcing fibers that appear in layered form. Each reinforcing layer contains a cluster of fibers with unique orientation, material, and cross-section area, and is simplified as a homogeneous membrane having unidirectional stiffness. You can specify multiple layers of reinforcing in one REINF263 element. The nodal locations, degrees of freedom, and connectivity of the REINF263 element are identical to those of the base element.

REINF263 has plasticity, stress stiffening, creep, large deflection, and large strain capabilities. See REINF263 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 263.1: REINF263 Geometry

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 263.2: REINF263 Coordinate System

X = Layer x-axis if layer orientation angle θ is specified (**SECDATA**,,,,*THETA*)
X_0 = Layer x-axis if layer orientation angle θ is *not* specified

REINF263 Input Data

The geometry and nodal locations for this element are shown in Figure 263.1: REINF263 Geometry (p. 1281). The REINF263 element and its base element share the same nodes and element connectivity.

You can easily create REINF263 elements from the selected base elements via the **EREINF** command. Section commands (**SECTYPE** and **SECDATA**) define the material ID, cross-section area, spacing, location, and orientation of reinforcing fibers.

The equivalent thickness *h* of the smeared reinforcing layer is given by

$$h = A / S$$

where *A* is the cross-section area of a single fiber, and *S* is the distance between two adjacent fibers.

The coordinate systems for one reinforcing layer are shown in Figure 263.2: REINF263 Coordinate System (p. 1282). Each reinforcing layer is indicated by its intersection points (II, JJ for linear base elements, and II, JJ, KK, for quadratic base elements) with the base elements. Fibers in this layer are always parallel to the first coordinate axis x. The x axis is default to the first parametric direction S1 at the center of the layer. The default axis is defined as

$$S_1 = \frac{\partial\{x\}}{\partial s} \Big/ \left(\left| \frac{\partial\{x\}}{\partial s} \right| \right)$$

where

$\{x\} = h_1\{x\}^{II} + h_2\{x\}^{JJ} + h_3\{x\}^{KK}$
$\{x\}^{II}, \{x\}^{JJ}, \{x\}^{KK}$ = global coordinates of intersection points
h_1, h_2, h_3 = line shape functions

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can reorient the layer coordinate system by angle θ (in degrees) for each layer. The value of θ is also provided for each layer via the **SECDATA** command. For more information about visualizing fiber orientations, see the **/PSYMB** command documentation.

You can use REINF263 to reinforce 2-D solid elements with plane stress, plane strain, axisymmetric, and generalized plane strain behaviors, and axisymmetric shells/membranes with or without uniform torsion capability. The element accounts for various base element behaviors automatically.

REINF263 allows tension-only or compression-only reinforcing fibers. You can specify the desired fiber behavior (**SECCONTROL**).

The REINF263 element does not accept element loading. Apply element loading only to the base element. The temperature of the REINF263 element is identical to the temperature of the base element.

You can import an initial stress state for this element (**INISTATE**). For more information, see Initial State in the *Basic Analysis Guide*.

A summary of the element input follows.

REINF263 Input Summary

Nodes

Same as those of the base element, as shown:

Base Element	REINF263 Nodes
2-D 4-Node Solid	I,J,K,L
2-D 8-Node Solid	I,J,K,L,M,N,O,P
2-D 2-Node Axisymmetric Shell	I,J
2-D 3-Node Axisymmetric Shell	I,J,K

Degrees of Freedom

Same as those of the base element, as shown:

Base Element	REINF263 DOFs
2-D 4-Node or 8-Node Solid	UX, UY
2-D 2-Node Axisymmetric Shell	UX, UY, ROTZ
2-D 2-Node Axisymmetric Membrane	UX, UY
2-D 2-Node Axisymmetric Shell Allowing Uniform Torsion	UX, UY, UZ, ROTX
2-D 2-Node Axisymmetric Membrane Allowing Uniform Torsion	UX, UY, UZ

Real Constants

None

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, (PRXY or NUXY), ALPX (or CTEX *or* THSX), DENS, GXY, ALPD, BETD

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface Loads

None

Body Loads

Temperatures --
Same as those of the base element

Special Features

Birth and death
Initial state
Large deflection
Large strain
Stress stiffening

KEYOPTS
None

REINF263 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 263.1: REINF263 Element Output Definitions (p. 1285).

The following figure illustrates the axial stress component:

Figure 263.3: REINF263 Stress Output

Unlike layered solid or shell elements (such as SHELL181), REINF263 always outputs the element solution for *all* reinforcing layers. You can select solution items for a *specific* reinforcing layer (**LAYER**) for listing and visualization by using full graphics (**/GRAPHICS**,FULL). Visualization via PowerGraphics (**/GRAPH-**

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ICS,POWER) is not affected by the **LAYER** command; all reinforcing layers are displayed simultaneously. See the *Basic Analysis Guide* for ways to review results.

To inspect REINF263 element results, select only REINF263 element results *or* adjust translucency level of the base elements before executing any plotting command. REINF263 display options are also available directly via the GUI (**Main Menu> Preprocessor> Sections> Reinforcing> Display Options**).

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 263.1: REINF263 Element Output Definitions

Name	Definition	O	R
EL	Element number and name	-	Y
NODES	Nodes (as shown in "REINF263 Input Summary" (p. 1283))	-	Y
MAT	Material number	-	Y
AREA	Averaged cross-section area of reinforcing fibers	-	Y
SPACING	Averaged distance between two adjacent fibers	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Center location	-	3
TEMP	T1, T2 for reinforcing layer 1; T3, T4 for reinforcing layer 2; ending with temperatures for the last reinforcing layer NL (2*NL maximum)	-	Y
S:X	Axial stresses	2	Y
EPEL:X	Axial elastic strains	2	Y
EPTH:X	Axial thermal strains	2	Y
EPPL:X	Axial plastic strains	2	1
EPCR:X	Axial creep strains	2	1
EPTO:X	Total axial mechanical strains (EPEL + EPPL + EPCR)	Y	-
EPTT:X	Total axial strains (EPEL + EPPL + EPCR + EPTH)	Y	-
NL:EPEQ	Accumulated equivalent plastic strain	-	1
NL:CREQ	Accumulated equivalent creep strain	-	1
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	-	1
NL:PLWK	Plastic work/volume	-	1
N11	Averaged axial force	-	Y
LOCI:X, Y, Z	Integration point locations	-	4

1. Nonlinear solution output if the element has a nonlinear material.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Stresses, total strains, plastic strains, elastic strains, creep strains, and thermal strains in the element coordinate system are available for output.

3. Available only at centroid as a ***GET** item.

4. Available only if **OUTRES**,LOCI is used.

Table 263.2: REINF263 Item and Sequence Numbers (p. 1286) lists output available via **ETABLE** using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this document for more information. The following notation is used in Table 263.2: REINF263 Item and Sequence Numbers (p. 1286):

Name
output quantity as defined in Table 263.1: REINF263 Element Output Definitions (p. 1285)

Item
predetermined Item label for **ETABLE**

E
sequence number for single-valued or constant element data

Table 263.2: REINF263 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
N11	SMISC	(i - 1) * 3 + 1
AREA	SMISC	(i - 1) * 3 + 2
SPACING	SMISC	(i - 1) * 3 + 3

The i value (where $i = 1, 2, 3, ..., NL$) represents the reinforcing layer number of the element. NL is the maximum reinforcing layer number.

REINF263 Assumptions and Restrictions

- Zero-volume elements are invalid.
- This element can be used only with base element types SHELL208, SHELL209, PLANE182, and PLANE183.
- A valid base element must be present for each REINF263 element.
- The reinforcing element is firmly attached to its base element. No relative movement between the reinforcing element and the base is allowed.
- Through-thickness reinforcing is not permitted in shells and layered solid elements.
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). You can also activate *prestress* effects (**PSTRES**).
- To simulate tension-/compression-only reinforcing fibers, a nonlinear iterative solution approach is necessary.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

REINF263 Product Restrictions

None.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

REINF264

3-D Discrete Reinforcing

MP ME ST PR PRN <> <> <> PP EME MFS
Product Restrictions

REINF264 Element Description

Use REINF264 with standard 3-D link, beam, shell and solid elements (referred to here as the *base elements*) to provide extra reinforcing to those elements.

The element is suitable for simulating reinforcing fibers with arbitrary orientations. Each fiber is modeled separately as a spar that has only uniaxial stiffness. You can specify multiple reinforcing fibers in one REINF264 element. The nodal locations, degrees of freedom, and connectivity of the REINF264 element are identical to those of the base element.

For *smeared* reinforcing modeling options, use the REINF263 and REINF265 elements.

REINF264 has plasticity, stress stiffening, creep, large deflection, and large strain capabilities. See REINF264 in the *Mechanical APDL Theory Reference* for more details about this element.

Table 264.1: REINF264 Geometry

3-D 8-Node Solid or Solid Shell

3-D 20-Node Solid

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3-D 4-Node Tetrahedral Solid

3-D 10-Node Tetrahedral Solid

3-D 4-Node Shell

3-D 8-Node Shell

3-D 2-Node Beam

3-D 3-Node Beam

3-D 2-Node Spar

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 264.1: REINF264 Coordinate System

REINF264 Input Data

The geometry and nodal locations for this element are shown in Table 264.1: REINF264 Geometry (p. 1289). The REINF264 element and its base element share the same nodes and element connectivity. Each reinforcing fiber is indicated by its intersection points (II, JJ for linear base elements, and II, JJ, KK for quadratic base elements) with the base elements.

You can easily create REINF264 elements from the selected base elements (**EREINF**). Section commands (**SECTYPE** and **SECDATA**) define the material ID, cross-section area, and location of reinforcing fibers.

REINF264 allows tension-only or compression-only reinforcing fibers. You can specify the desired fiber behavior (**SECCONTROL**).

The coordinate system for one reinforcing fiber is shown in Figure 264.1: REINF264 Coordinate System (p. 1291). The coordinate system is solely determined by intersection points II, JJ, and KK; therefore, the element coordinate system (**/PSYMB**,ESYS) is not relevant for this element.

The REINF264 element does not accept element loading. Apply element loading only to the base element. The temperature of the REINF264 element is identical to the temperature of the base element.

You can import an initial stress state for this element (**INISTATE**). For more information, see Initial State in the *Basic Analysis Guide*.

A summary of the element input follows.

REINF264 Input Summary

Nodes

Same as those of the base element, as shown:

Base Element	REINF264 Nodes
3-D 8-Node Solid or Solid Shell	I,J,K,L,M,N,O,P
3-D 20-Node Solid	I,J,K,L,M,N,O,P,Q,R,S,T,U,V,W,X,Y,Z,A,B
3-D 4-Node Tetrahedral Solid	I,J,K,L
3-D 10-Node Tetrahedral Solid	I,J,K,L,M,N,O,P,Q,R
3-D 4-Node Shell	I,J,K,L
3-D 8-Node Shell	I,J,K,L,M,N,O,P
3-D 2-Node Beam	I,J,K (K is an optional orientation node)
3-D 3-Node Beam	I,J,K,L (L is an optional orientation node)
3-D 2-Node Spar	I,J

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Degrees of Freedom

Same as those of the base element, as shown:

Base Element	REINF264 DOFs
3-D 8-Node Solid or Solid Shell	UX, UY, UZ
3-D 20-Node Solid	UX, UY, UZ
3-D 14-Node Tetrahedral Solid	UX, UY, UZ
3-D 10-Node Tetrahedral Solid	UX, UY, UZ
3-D 4-Node Shell	UX, UY, UZ, ROTX, ROTY, ROTZ
3-D 8-Node Shell	UX, UY, UZ, ROTX, ROTY, ROTZ
3-D 2-Node Beam	UX, UY, UZ, ROTX, ROTY, ROTZ
3-D 3-Node Beam	UX, UY, UZ, ROTX, ROTY, ROTZ
3-D 2-Node Spar	UX, UY, UZ

Real Constants

None

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, (PRXY or NUXY), ALPX (or CTEX *or* THSX), DENS, GXY, ALPD, BETD

Surface Loads

None

Body Loads

Temperatures --

Same as those of the base element

Special Features

Birth and death
Initial state
Large deflection
Large strain
Stress stiffening

KEYOPTS

None

REINF264 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 264.2: REINF264 Element Output Definitions (p. 1293).

The axial stress component is illustrated in Figure 264.2: REINF264 Stress Output (p. 1293).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 264.2: REINF264 Stress Output

Unlike layered solid or shell elements (such as SHELL181), REINF264 always outputs the element solution for *all* reinforcing layers. You can select solution items for a *specific* reinforcing layer (**LAYER**) for listing and visualization by using full graphics (**/GRAPHICS**,FULL). Visualization via PowerGraphics (**/GRAPHICS**,POWER) is not affected by the **LAYER** command; all reinforcing layers are displayed simultaneously. See the *Basic Analysis Guide* for ways to review results.

To inspect REINF264 element results, select only REINF264 element results *or* adjust the translucency level of the base elements before executing any plotting command. REINF264 display options are also available directly via the GUI (**Main Menu> Preprocessor> Sections> Reinforcing> Display Options**).

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 264.2: REINF264 Element Output Definitions

Name	Definition	O	R
EL	Element number and name	-	Y
NODES	Nodes (as shown in "REINF264 Input Summary" (p. 1291))	-	Y
MAT	Material number	-	Y
AREA	Averaged cross-section area of reinforcing fibers	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Center location	-	3
TEMP	T1, T2 for reinforcing fiber 1; T3, T4 for reinforcing fiber 2; ending with temperatures for the last reinforcing fiber NL (2 * NL maximum)	-	Y
S:X	Axial stresses	2	Y
EPEL:X	Axial elastic strains	2	Y
EPTH:X	Axial thermal strains	2	Y
EPPL:X	Axial plastic strains	2	1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
EPCR:X	Axial creep strains	2	1
EPTO:X	Total axial mechanical strains (EPEL + EPPL + EPCR)	Y	-
NL:EPEQ	Accumulated equivalent plastic strain	-	1
NL:CREQ	Accumulated equivalent creep strain	-	1
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	-	1
NL:PLWK	Plastic work/volume	-	1
N11	Averaged axial force	-	Y
LOCI:X, Y, Z	Integration point locations	-	4

1. Nonlinear solution output if the element has a nonlinear material.
2. Stresses, total strains, plastic strains, elastic strains, creep strains, and thermal strains in the element coordinate system are available for output.
3. Available only at centroid as a ***GET** item.
4. Available only if **OUTRES**,LOCI is used.

Table 264.3: REINF264 Item and Sequence Numbers (p. 1294) lists output available via the **ETABLE** command using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this document for more information. The following notation is used in Table 264.3: REINF264 Item and Sequence Numbers (p. 1294):

Name
output quantity as defined in Table 264.2: REINF264 Element Output Definitions (p. 1293)

Item
predetermined Item label for **ETABLE**

E
sequence number for single-valued or constant element data

Table 264.3: REINF264 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
N11	SMISC	(i - 1) * 2 + 1
AREA	SMISC	(i - 1) * 2 + 2

The i value (where i = 1, 2, 3, ..., NL) represents the reinforcing fiber number of the element. NL is the maximum reinforcing fiber number.

REINF264 Assumptions and Restrictions

- Zero-volume elements are invalid.
- This element can be used only with base element types LINK180, SHELL181, SHELL281, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, and SOLID285.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- A valid base element must be present for each REINF264 element.
- The reinforcing element is firmly attached to its base element. No relative movement between the reinforcing element and the base is allowed.
- Through-thickness reinforcing is not permitted in shells and layered solid elements.
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). You can activate *prestress* effects via the **PSTRES** command.
- The warping degree of freedom in beam base elements are not accounted for.
- REINF264 does not support BEAM188 with the quadratic or cubic interpolation option (KEYOPT(3)).
- To simulate the tension-/compression-only reinforcing fibers, a nonlinear iterative solution approach is necessary.

REINF264 Product Restrictions

None.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

REINF265

3-D Smeared Reinforcing

MP ME ST PR PRN <> <> <> PP EME MFS
Product Restrictions

REINF265 Element Description

Use REINF265 with standard 3-D solid and shell elements (referred to here as the *base elements*) to provide extra reinforcing to those elements.

The element uses a smeared approach and is suitable for modeling evenly spaced reinforcing fibers that appear in layered form. Each reinforcing layer contains a cluster of fibers with unique orientation, material, and cross-section area, and is simplified as a homogeneous membrane having unidirectional stiffness. You can specify multiple layers of reinforcing in one REINF265 element. The nodal locations, degrees of freedom, and connectivity of the REINF265 element are identical to those of the base element.

For *discrete* reinforcing modeling options, see the documentation for the REINF264 element.

REINF265 has plasticity, stress stiffening, creep, large deflection, and large strain capabilities. See REINF265 in the *Mechanical APDL Theory Reference* for more details about this element.

Figure 265.1: REINF265 Geometry

3-D 8-Node Solid or Solid Shell

3-D 20-Node Solid

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3-D 10-Node Tetrahedral Solid

3-D 4-Node Shell

3-D 8-Node Shell

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 265.2: REINF265 Coordinate System

X = Layer x-axis if the local coordinate system reference number (**SECDATA**,,,*KCN*) for the layer is specified
X_0 = Layer x-axis if the local coordinate system reference number is *not* specified

REINF265 Input Data

The geometry and nodal locations for this element are shown in Figure 265.1: REINF265 Geometry (p. 1297). The REINF265 element and its base element share the same nodes and element connectivity.

You can easily create REINF265 elements from the selected base elements via the **EREINF** command. Section commands (**SECTYPE** and **SECDATA**) define the material ID, cross-section area, spacing, location, and orientation of reinforcing fibers.

The equivalent thickness *h* of the smeared reinforcing layer is given by

$$h = A / S$$

where *A* is the cross-section area of a single fiber, and *S* is the distance between two adjacent fibers.

The coordinate systems for one reinforcing layer are shown in Figure 265.2: REINF265 Coordinate System (p. 1299). Each reinforcing layer is indicated by its intersection points (II, JJ, KK, LL for linear base elements, and II, JJ, KK, LL, MM, NN, OO, PP for quadratic base elements) with the base elements. Fibers in this layer are always parallel to the first coordinate axis x. The x axis is default to the first parametric direction S1 at the center of the layer. The default axis is defined as

$$S_1 = \frac{\partial\{x\}}{\partial s} \Big/ \left(\left| \frac{\partial\{x\}}{\partial s} \right| \right)$$

where

$$\frac{\partial\{x\}}{\partial s} = \left(\frac{1}{4}\right)\left[-\{x\}^{II} + \{x\}^{JJ} + \{x\}^{KK} - \{x\}^{LL}\right]$$

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$\{x\}^{II}$, $\{x\}^{JJ}$, $\{x\}^{KK}$, $\{x\}^{LL}$ = global nodal coordinates

You can reorient the default layer coordinate system by projecting a local coordinate system (**LOCAL**) to the layer plane. One local coordinate system is allowed for each layer. The local coordinate system reference number is given via the **SECDATA** command.

You can further rotate the layer coordinate system by angle θ (in degrees) for each layer. The value of θ is also provided for each layer via **SECDATA**. For more information about visualizing fiber orientations, see the **/PSYMB** command documentation.

REINF265 allows tension-only or compression-only reinforcing fibers. You can specify the desired fiber behavior (**SECCONTROL**).

The REINF265 element does not accept element loading. Apply element loading only to the base element. The temperature of the REINF265 element is identical to the temperature of the base element.

You can import an initial stress state for this element (**INISTATE**). For more information, see Initial State in the *Basic Analysis Guide*.

A summary of the element input follows.

REINF265 Input Summary

Nodes

Same as those of the base element, as shown:

Base Element	REINF265 Nodes
3-D 8-Node Solid or Solid Shell	I,J,K,L,M,N,O,P
3-D 20-Node Solid	I,J,K,L,M,N,O,P,Q,R,S,T,U,V,W,X,Y,Z,A,B
3-D 10-Node Tetrahedral Solid	I,J,K,L,M,N,O,P,Q,R
3-D 4-Node Shell	I,J,K,L
3-D 8-Node Shell	I,J,K,L,M,N,O,P

Degrees of Freedom

Same as those of the base element, as shown:

Base Element	REINF265 DOFs
3-D 8-Node Solid or Solid Shell	UX, UY, UZ
3-D 20-Node Solid	UX, UY, UZ
3-D 10-Node Tetrahedral Solid	UX, UY, UZ
3-D 4-Node Shell	UX, UY, UZ, ROTX, ROTY, ROTZ
3-D 8-Node Shell	UX, UY, UZ, ROTX, ROTY, ROTZ

Real Constants

None

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MP command: EX, (PRXY or NUXY), ALPX (or CTEX *or* THSX), DENS, GXY, ALPD, BETD

Surface Loads

None

Body Loads

Temperatures --
Same as those of the base element

Special Features

Birth and death
Initial state
Large deflection
Large strain
Stress stiffening

KEYOPTS
None

REINF265 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 265.1: REINF265 Element Output Definitions (p. 1302).

The following figure illustrates the axial stress component:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 265.3: REINF265 Stress Output

X = Layer x-axis if the local coordinate system reference number (**SECDATA**,,,,*KCN*) for the layer is specified
X_0 = Layer x-axis if the local coordinate system reference number is *not* specified

Unlike layered solid or shell elements (such as SHELL181), REINF265 always outputs the element solution for *all* reinforcing layers. You can select solution items for a *specific* reinforcing layer (**LAYER**) for listing and visualization by using full graphics (**/GRAPHICS**,FULL). Visualization via PowerGraphics (**/GRAPHICS**,POWER) is not affected by the **LAYER** command; all reinforcing layers are displayed simultaneously. See the *Basic Analysis Guide* for ways to review results.

To inspect REINF265 element results, select only REINF265 element results *or* adjust translucency level of the base elements before executing any plotting command. REINF265 display options are also available directly via the GUI (**Main Menu> Preprocessor> Sections> Reinforcing> Display Options**).

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 265.1: REINF265 Element Output Definitions

Name	Definition	O	R
EL	Element number and name	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
NODES	Nodes (as shown in "REINF265 Input Summary" (p. 1300))	-	Y
MAT	Material number	-	Y
AREA	Averaged cross-section area of reinforcing fibers	-	Y
SPACING	Averaged distance between two adjacent fibers	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Center location	-	3
TEMP	T1, T2, T3, T4 for reinforcing layer 1; T5, T6, T7, T8 for reinforcing layer 2; ending with temperatures for the last reinforcing layer NL (4*NL maximum)	-	Y
S:X	Axial stresses	2	Y
EPEL:X	Axial elastic strains	2	Y
EPTH:X	Axial thermal strains	2	Y
EPPL:X	Axial plastic strains	2	1
EPCR:X	Axial creep strains	2	1
EPTO:X	Total axial mechanical strains (EPEL + EPPL + EPCR)	Y	-
NL:EPEQ	Accumulated equivalent plastic strain	-	1
NL:CREQ	Accumulated equivalent creep strain	-	1
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	-	1
NL:PLWK	Plastic work/volume	-	1
N11	Averaged axial force	-	Y
LOCI:X, Y, Z	Integration point locations	-	4

1. Nonlinear solution output if the element has a nonlinear material.
2. Stresses, total strains, plastic strains, elastic strains, creep strains, and thermal strains in the element coordinate system are available for output.
3. Available only at centroid as a ***GET** item.
4. Available only if **OUTRES**,LOCI is used.

Table 265.2: REINF265 Item and Sequence Numbers (p. 1304) lists output available through **ETABLE** using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this document for more information. The following notation is used in Table 265.2: REINF265 Item and Sequence Numbers (p. 1304):

Name

output quantity as defined in Table 265.1: REINF265 Element Output Definitions (p. 1302)

Item

predetermined Item label for **ETABLE**

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

E

sequence number for single-valued or constant element data

Table 265.2: REINF265 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input	
	Item	E
N11	SMISC	(i - 1) * 3 + 1
AREA	SMISC	(i - 1) * 3 + 2
SPACING	SMISC	(i - 1) * 3 + 3

The i value (where $i = 1, 2, 3, ..., NL$) represents the reinforcing layer number of the element. NL is the maximum reinforcing layer number ($1 \leq NL \leq 250$).

REINF265 Assumptions and Restrictions

- Zero-volume elements are invalid.
- This element can be used only with base element types SHELL181, SHELL281, SOLID185, SOLID186, SOLID187, and SOLSH190.
- A valid base element must be present for each REINF265 element.
- The reinforcing element is firmly attached to its base element. No relative movement between the reinforcing element and the base is allowed.
- Through-thickness reinforcing is not permitted in shells and layered solid elements.
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). You can activate *prestress* effects via the **PSTRES** command.
- To simulate tension-/compression-only reinforcing fibers, a nonlinear iterative solution approach is necessary.

REINF265 Product Restrictions

None.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID272

General Axisymmetric Solid with 4 Base Nodes

MP ME ST <> <> <> <> <> PP EME MFS
Product Restrictions

SOLID272 Element Description

Use SOLID272 to model axisymmetric solid structures. It is defined by four nodes on the master plane, and nodes created automatically in the circumferential direction based on the four master plane nodes. The total number of nodes depends on the number of nodal planes (KEYOPT(2)). Each node has three degrees of freedom: translations in the nodal x, y and z directions. The element allows a triangle as the degenerated shape on the base plane to simulate irregular areas. The element has plasticity, hyperelasticity, stress stiffening, large deflection, and large strain capabilities. It also has mixed-formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and nearly and fully incompressible hyperelastic materials.

For more details about this element, see SOLID272 in the *Mechanical APDL Theory Reference*, and General Axisymmetric Elements (p. 76) in this document.

Figure 272.1: SOLID272 Geometry (KEYOPT(2) = 3)

SOLID272 Input Data

The geometry and node locations for this element (when KEYOPT(2) = 3) are shown in Figure 272.1: SOLID272 Geometry (KEYOPT(2) = 3) (p. 1305). The element input data includes nodes and the orthotropic material properties. The total number of nodes is the four base nodes times the number of nodal planes. (For information about how Fourier nodes are generated, see the **NAXIS** command documentation.) The default element coordinate system is the cylindrical coordinate system with the z axis as the

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

axisymmetric axis (defined via the **SECDATA** command) and the circumferential direction as θ. (See General Axisymmetric Elements (p. 76) for details.) Use the **ESYS** command to define an element coordinate system, which forms the basis for orthotropic material directions.

Element loads are described in Nodal Loading (p. 49). Pressures must be input as element surface loads on the element edges of the nodal planes as shown by the circled numbers in Figure 272.1: SOLID272 Geometry (KEYOPT(2) = 3) (p. 1305). Positive pressures act into the element and the maximum face edge is 4n, where n is the number of nodal planes. If pressure is applied on the element edge with face numbers less than or equal to 4 and no load on other edges, the pressure loads are the same on the 360 degrees of circumferential surfaces. (If pressure is applied on a single element edge with a face number *greater* than 4, the pressure is ignored.) If pressure is applied on the element edges with faces p and 4q+p (where q = 1 . . . n-1), the pressure changes linearly with respect to θ within the part of the surface bounded by the edges p and 4q+p; on the rest of the surface, the pressure is zero.

Temperatures may be input as element body loads at the nodes. For the four nodes on the master plane, the node I1 temperature $T(I_1)$ defaults to TUNIF. If all other temperatures are unspecified, they default to $T(I_1)$. For any other input pattern, unspecified temperatures default to TUNIF. For the nodes generated in the circumferential direction based on the master node, they default to the value of their base nodes ($T(I_1)$, $T(J_1)$, $T(K_1)$ or $T(L_1)$, depending on their location) if all other temperatures are unspecified. For any other input pattern, unspecified temperatures default to TUNIF.

KEYOPT(6) = 1 sets the element for using mixed formulation. For details on the use of mixed formulation, see Applications of Mixed u-P Formulations in this document.

As described in Coordinate Systems, you can use the **ESYS** command to orient the material properties and strain/stress output. Use **RSYS** to choose output that follows the material coordinate system or the global coordinate system.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, issue a **NROPT**,UNSYM command. For a geometric nonlinear analysis when convergence is an issue, use an unsymmetric matrix.

"SOLID272 Input Summary" (p. 1306) contains a summary of the element input. See Element Input (p. 43) in this document for a general description of element input.

SOLID272 Input Summary

Nodes

I_1, J_1, K_1, L_1, I_2, J_2, K_2, L_2, . . . , I_n, J_n, K_n, L_n (where n = KEYOPT(2), the number of nodal planes)

Degrees of Freedom

UX, UY, UZ

Real Constants

None

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.

MP command: EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ),

ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ),

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DENS, GXY, GYZ, GXZ, ALPD, BETD

Surface Loads

Pressures --
edge 1 (J_1-I_1), edge 2 (K_1-J_1), edge 3 (L_1-K_1), edge 4 (I_1-L_1), edge 5 (J_2-I_2), edge 6 (K_2-J_2), . . . , edge 4n-3(J_n-I_n), edge 4n-2 (K_n-J_n), edge 4n-1 (L_n-K_n), edge 4n (I_n-L_n)

Body Loads

Temperatures --
T(I_1), T(J_1), T(K_1), T(L_1), T(I_2), T(J_2), T(K_2), T(L_2), . . . , T(I_n), T(J_n), T(K_n), T(L_n)

Special Features

Birth and death
Element technology autoselect
Large deflection
Large strain
Nonlinear stabilization
Stress stiffening

KEYOPT(2)
Number of Fourier nodes in the circumferential direction (that is, the number of nodal planes):

1 --
Axisymmetric deformation (may have torsion)

3 - 12 --
General 3-D deformation

This KEYOPT has no default. You must specify a valid value. (0 is not valid.)

For large-rotation not about the axisymmetric axis, KEYOPT(2) $\geq$ 5 is recommended.

For information about specifying the number of Fourier nodes, see General Axisymmetric Elements in this document.

For information about how Fourier nodes are generated, see the **NAXIS** command documentation.

KEYOPT(6)
Element formulation:

0 --
Use pure displacement formulation (default)

1 --
Use mixed u-P formulation

SOLID272 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Additional element output as shown in Table 272.1: SOLID272 Element Output Definitions (p. 1308)

As shown in Figure 272.2: SOLID272 Stress Output (p. 1308), the element stress directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

To view 3-D mode shapes for a modal or eigenvalue buckling analysis, expand the modes with element results calculation active (via the **MXPAND** command's `Elcalc` = YES option).

Figure 272.2: SOLID272 Stress Output

Element stress directions SX, SY, and SZ shown in the global coordinate system.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 272.1: SOLID272 Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Nodes - I_1, J_1, K_1, L_1, I_2, J_2, K_2, L_2, . . .	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
MAT	Material number	-	Y
VOLU	Volume	-	Y
XC, YC, ZC	Location where results are reported	-	3
PRES	Pressures P1 at nodes J,I; P2 at K,J; P3 at L,K; P4 at I,L See Table 272.2: SOLID272 Item and Sequence Numbers (p. 1310) for more output.	-	Y
TEMP	Temperatures $T(I_1)$, $T(J_1)$, $T(K_1)$, $T(L_1)$, $T(I_2)$, $T(J_2)$, $T(K_2)$, $T(L_2)$, . . .	-	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	Y	Y
S:1, 2, 3	Principal stresses	-	Y
S:INT	Stress intensity	-	Y
S:EQV	Equivalent stress	Y	Y
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	Y	Y
EPEL:1, 2, 3	Principal elastic strains	-	Y
EPEL:EQV	Equivalent elastic strain [6]	Y	Y
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	2	2
EPTH:EQV	Equivalent thermal strain [6]	2	2
EPPL:X, Y, Z, XY, YZ, XZ	Plastic strains[7]	1	1
EPPL:EQV	Equivalent plastic strain [6]	1	1
EPCR:X, Y, Z, XY, YZ, XZ	Creep strains	1	1
EPCR:EQV	Equivalent creep strains [6]	1	1
EPTO:X, Y, Z, XY, YZ, XZ	Total mechanical strains (EPEL + EPPL + EPCR)	Y	-
EPTO:EQV	Total equivalent mechanical strains (EPEL + EPPL + EPCR)	Y	-
NL:EPEQ	Accumulated equivalent plastic strain	1	1
NL:CREQ	Accumulated equivalent plastic strain	1	1
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	1	1
NL:PLWK	Plastic work/volume	1	1
NL:HPRES	Hydrostatic pressure	1	1
SEND:ELASTIC, PLASTIC, CREEP	Strain energy densities	-	1
LOCI:X, Y, Z	Integration point locations	-	4
SVAR:1, 2, . . . , N	State variables	-	5

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Nonlinear solution, output only if the element has a nonlinear material.
2. Output only if element has a thermal load.
3. Available only at centroid as a ***GET** item.
4. Available only if **OUTRES**,LOCI is used.
5. Available only if the `UserMat` subroutine and **TB**,STATE command are used.
6. The equivalent strains use an effective Poisson's ratio: for elastic and thermal strains, this value is set by the user (**MP**,PRXY); for plastic and creep strains, this value is set at 0.5.
7. For the shape memory alloy material model, transformation strains are reported as plasticity strain EPPL.

Table 272.2: SOLID272 Item and Sequence Numbers (p. 1310) lists output available through **ETABLE** using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this document for more information. The following notation is used in the output table:

Name
output quantity as defined in the Table 272.1: SOLID272 Element Output Definitions (p. 1308)

Item
predetermined Item label for **ETABLE**

I_1, J_1, K_1, L_1, I_2, J_2, K_2, L_2, . . .
sequence number for data at nodes I_1, J_1, K_1, L_1, I_2, J_2, K_2, L_2, . . .

Table 272.2: SOLID272 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input													
	Item	I_1	J_1	K_1	L_1	I_2	J_2	K_2	L_2	---	I_n	J_n	K_n	L_n
P_1	SMISC	2	1							---				
P_2	SMISC		4	3						---				
P_3	SMISC			6	5					---				
P_4	SMISC	7			8					---				
P_5	SMISC					10	9			---				
P_6	SMISC						12	11		---				
P_7	SMISC							14	13	---				
P_8	SMISC					15			16	---				
---	---	---	---	---	---	---	---	---	---	---	---	---	---	---
P_{4n-3}	SMISC									---	8n-6	8n-7		
P_{4n-2}	SMISC									---		8n-4	8n-5	
P_{4n-1}	SMISC									---			8n-2	8n-3

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input													
	Item	I_1	J_1	K_1	L_1	I_2	J_2	K_2	L_2	---	I_n	J_n	K_n	L_n
P_{4n}	SMISC									---	8n-1			8n

SOLID272 Assumptions and Restrictions

- The area of the base element must be nonzero.
- The base element must lie on one side of the axisymmetric axis, and the axisymmetric axis must be on the same plane as the base element (master plane).
- A base element or base node must be associated with one axisymmetric axis (defined via **SECTYPE** and **SECDATA** commands) *before* generating nodes for general axisymmetric element sections (**NAXIS**) or defining an element by node connectivity (**E**).
- You can form a triangular base element by defining duplicate K1 and L1 node numbers. (See Degenerated Shape Elements (p. 39) in this document.)
- You cannot apply a pressure load via the **SFA** command.
- Incompressible and nearly incompressible material behavior should be modeled with the mixed u-P formulation.
- If you specify mixed formulation (KEYOPT(6) = 1), you must use the sparse solver.
- The contribution of the element to the mass moment inertia of the whole model is calculated by element mass multiplied by the square of the coordinates of the elemental centroid. The moment of inertia may therefore be inaccurate.
- The element does not support the expansion pass of a superelement with large rotation.
- Issuing an **/ESHAPE**,1 command while PowerGraphics is active causes the program to plot the elements in 3-D and the results on both nodal planes and all integration planes in the circumferential direction; otherwise, the program plots the elements in 2-D and the results on the master plane.
- You cannot display surface load symbols (**/PSF**) when displaying this element in 3-D via the **/ESHAPE** command.
- When specifying more than one facet per element edge for PowerGraphics displays (**NAXIS**,EFACET,*NUM*, where *NUM* > 1), ANSYS plots additional results on some planes between the nodal and integration planes. The results on these planes are interpolated based on the nodal and integration plane values. and are therefore less accurate than the values on the nodal and interpolation planes. If you do not wish to plot the interpolated values, set *NUM* = 1 to plot only the values on nodal and integration planes.
- Print commands in postprocessing print the nodal plane results only.
- To model axisymmetric solid surface loads acting on this element, use general axisymmetric surface element SURF159. (You cannot use this element with surface-effect elements SURF153 and SURF154.)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You cannot generate surface-based contact pairs (contact elements CONTA171 through CONTA174 paired with target elements TARGE169 and TARGE170) on this element.
- You *can* generate node-to-surface contact pairs (contact elements CONTA175 paired with target elements TARGE170) and node-to-node contact elements (CONTA178) on this element, with the following restrictions:
  - When TARGE170 is on the surface of SOLID272, you may have accuracy and convergence issues if the loading causes large rotations about the axisymmetric axis of SOLID272; you may also have those issues if the two sides of the contact boundaries have different mesh patterns in the circumferential direction (caused by different KEYOPT(2) values of SOLID272).
  - You cannot define CONTA175 with the multipoint constraint (MPC) approach using a force-distributed constraint (that is, you cannot set KEYOPT(2) = 2, KEYOPT(4) = 1, and KEYOPT(12) = 5 or 6 simultaneously for the CONTA175 elements).
  - In the case of a true axisymmetric condition (SOLID272 with KEYOPT(2) = 1), you should use 2-D node-to-surface contact pairs (CONTA175 contact elements paired with TARGE169 target elements).
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). Prestress effects can be activated via the **PSTRES** command.

SOLID272 Product Restrictions

No product-specific restrictions exist for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID273

General Axisymmetric Solid with 8 Base Nodes

MP ME ST <> <> <> <> <> PP EME MFS
Product Restrictions

SOLID273 Element Description

Use SOLID273 to model axisymmetric solid structures. The element has quadratic displacement behavior on the master plane and is well suited to modeling irregular meshes on the master plane. It is defined by eight nodes on the master plane, and nodes created automatically in the circumferential direction based on the eight master plane nodes. The total number of nodes depends on the number of nodal planes (KEYOPT(2)). Each node has three degrees of freedom: translations in the nodal x, y and z directions. The element allows a triangle as the degenerated shape on the base plane to simulate irregular areas. The element has plasticity, hyperelasticity, stress stiffening, large deflection, and large strain capabilities. It also has mixed-formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and nearly and fully incompressible hyperelastic materials.

For more details about this element, see SOLID273 in the *Mechanical APDL Theory Reference*, and General Axisymmetric Elements (p. 76) in this document.

Figure 273.1: SOLID273 Geometry (KEYOPT(2) = 3)

SOLID273 Input Data

The geometry and node locations for this element (when KEYOPT(2) = 3) are shown in Figure 273.1: SOLID273 Geometry (KEYOPT(2) = 3) (p. 1313). The element input data includes nodes and the orthotropic material properties. The total number of nodes is the 8 base nodes times the number of nodal planes.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(For information about how Fourier node are generated, see the **NAXIS** command documentation.) The default element coordinate system is the cylindrical coordinate system with the Z axis as the axisymmetric axis (defined via the **SECDATA** command) and the circumferential direction as θ. (See General Axisymmetric Elements (p. 76) for details.) Use the **ESYS** command to define an element coordinate system, which forms the basis for orthotropic material directions.

Element loads are described in Nodal Loading (p. 49). Pressures must be input as element surface loads on the element edge of nodal planes as shown by the circled numbers in Figure 273.1: SOLID273 Geometry (KEYOPT(2) = 3) (p. 1313). Positive pressures act into the element and the maximum face edge is 4n, where n is the number of nodal planes. If pressure is applied on the element edge with face numbers less that or equal to 4 and no load on other edges, the pressure loads are the same on the 360 degrees of circumferential surfaces. (If pressure is applied on a single element edge with a face number *greater* than 4, the pressure is ignored.) If pressure is applied on the element edges with faces p and 4q+p (where q = 1 . . . n-1), the pressure changes linearly with respect to θ within the part of the surface bounded by the edges p and 4q+p; on the rest of the surface, the pressure is zero.

Temperatures may be input as element body loads at the nodes. For the eight nodes on the master plane, the node I_1 temperature $T(I_1)$ defaults to TUNIF. If all other temperatures are unspecified, they default to $T(I_1)$. If all corner node temperatures are specified, each midside node temperature defaults to the average temperature of its adjacent corner nodes. For any other input pattern, unspecified temperatures default to TUNIF. For the nodes generated in circumferential direction based on the master node, if all other temperatures are unspecified, they default to the value of their base nodes ($T(I_1)$, $T(J_1)$, $T(K_1)$, $T(L_1)$, $T(M_1)$, $T(N_1)$, $T(O_1)$ and $T(P_1)$, depending on their location). For any other input pattern, unspecified temperatures default to TUNIF.

KEYOPT(6) = 1 sets the element for using mixed formulation. For details on the use of mixed formulation, see Applications of Mixed u-P Formulations in this document.

As described in Coordinate Systems, you can use the **ESYS** command to orient the material properties and strain/stress output. Use **RSYS** to choose output that follows the material coordinate system or the global coordinate system.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, issue a **NROPT**,UNSYM command. For a geometric nonlinear analysis when convergence is an issue, use an unsymmetric matrix.

"SOLID273 Input Summary" (p. 1314) contains a summary of the element input. See Element Input (p. 43) in this document for a general description of element input.

SOLID273 Input Summary

Nodes

I_1, J_1, K_1, L_1, M_1, N_1, O_1, P_1 , I_2, J_2, K_2, L_2, M_2, N_2, O_2, P_2, . . . , I_n, J_n, K_n, L_n, M_n, N_n, O_n, P_n (where n = KEYOPT(2), the number of nodal planes)

Degrees of Freedom

UX, UY, UZ

Real Constants

None

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.

MP command: EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ),

ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ),

DENS, GXY, GYZ, GXZ, ALPD, BETD

Surface Loads

Pressures --
edge 1 (J_1-I_1), edge 2 (K_1-J_1), edge 3 (L_1-K_1), edge 4 (I_1-L_1), edge 5 (J_2-I_2), edge 6 (K_2-J_2), . . . , edge 4n-3(J_n-I_n), edge 4n-2 (K_n-J_n), edge 4n-1 (L_n-K_n), edge 4n (I_n-L_n)

Body Loads

Temperatures --
T(I_1), T(J_1), T(K_1), T(L_1), T(M_1), T(N_1), T(O_1), T(P_1), T(I_2), T(J_2), T(K_2), T(L_2), T(M_2), T(N_2), T(O_2), T(P_2), . . . , T(I_n), T(J_n), T(K_n), T(L_n), T(M_n), T(N_n), T(O_n), T(P_n)

Special Features

Birth and death
Element technology autoselect
Large deflection
Large strain
Nonlinear stabilization
Stress stiffening

KEYOPT(2)

Number of Fourier nodes in the circumferential direction (that is, the number of nodal planes):

1 --
Axisymmetric deformation (may have torsion)

3 - 12 --
General 3-D deformation

This KEYOPT has no default. You must specify a valid nonzero value.

For large-rotation not about the axisymmetric axis, KEYOPT(2) $\geq$ 5 is recommended.

For information about specifying the number of Fourier nodes, see General Axisymmetric Elements in this document.

For information about how Fourier nodes are generated, see the **NAXIS** command documentation.

KEYOPT(6)

Element formulation:

0 --
Use pure displacement formulation (default)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --
Use mixed u-P formulation

SOLID273 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 273.1: SOLID273 Element Output Definitions (p. 1317)

As shown in Figure 273.2: SOLID273 Stress Output (p. 1316), the element stress directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

To view 3-D mode shapes for a modal or eigenvalue buckling analysis, expand the modes with element results calculation active (via the **MXPAND** command's `Elcalc` = YES option).

Figure 273.2: SOLID273 Stress Output

Element stress directions SX, SY, and SZ shown in the global coordinate system.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 273.1: SOLID273 Element Output Definitions

Name	Definition	O	R
EL	Element number	-	Y
NODES	Nodes - I_1, J_1, K_1, L_1, I_2, J_2, K_2, L_2, . . .	-	Y
MAT	Material number	-	Y
VOLU	Volume	-	Y
XC, YC, ZC	Location where results are reported	-	3
PRES	Pressures P1 at nodes J,I; P2 at K,J; P3 at L,K; P4 at I,L See Table 273.2: SOLID273 Item and Sequence Numbers (p. 1318) for more output.	-	Y
TEMP	Temperatures $T(I_1)$, $T(J_1)$, $T(K_1)$, $T(L_1)$, $T(I_2)$, $T(J_2)$, $T(K_2)$, $T(L_2)$, . . .	-	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	Y	Y
S:1, 2, 3	Principal stresses	-	Y
S:INT	Stress intensity	-	Y
S:EQV	Equivalent stress	Y	Y
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	Y	Y
EPEL:1, 2, 3	Principal elastic strains	-	Y
EPEL:EQV	Equivalent elastic strain [6]	Y	Y
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	2	2
EPTH:EQV	Equivalent thermal strain [6]	2	2
EPPL:X, Y, Z, XY, YZ, XZ	Plastic strains[7]	1	1
EPPL:EQV	Equivalent plastic strain [6]	1	1
EPCR:X, Y, Z, XY, YZ, XZ	Creep strains	1	1
EPCR:EQV	Equivalent creep strains [6]	1	1
EPTO:X, Y, Z, XY, YZ, XZ	Total mechanical strains (EPEL + EPPL + EPCR)	Y	-
EPTO:EQV	Total equivalent mechanical strains (EPEL + EPPL + EPCR)	Y	-
NL:EPEQ	Accumulated equivalent plastic strain	1	1
NL:CREQ	Accumulated equivalent plastic strain	1	1
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	1	1

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
NL:PLWK	Plastic work/volume	1	1
NL:HPRES	Hydrostatic pressure	1	1
SEND:ELASTIC, PLASTIC, CREEP	Strain energy densities	-	1
LOCI:X, Y, Z	Integration point locations	-	4
SVAR:1, 2, ... , N	State variables	-	5

1. Nonlinear solution, output only if the element has a nonlinear material.
2. Output only if element has a thermal load.
3. Available only at centroid as a ***GET** item.
4. Available only if **OUTRES**,LOCI is used.
5. Available only if the `UserMat` subroutine and **TB**,STATE command are used.
6. The equivalent strains use an effective Poisson's ratio: for elastic and thermal strains, this value is set by the user (**MP**,PRXY); for plastic and creep strains, this value is set at 0.5.
7. For the shape memory alloy material model, transformation strains are reported as plasticity strain EPPL.

Table 273.2: SOLID273 Item and Sequence Numbers (p. 1318) lists output available through **ETABLE** using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this document for more information. The following notation is used in the output table:

Name
output quantity as defined in the Table 273.1: SOLID273 Element Output Definitions (p. 1317)

Item
predetermined Item label for **ETABLE**

I_1, J_1, K_1, L_1, M_1, N_1, O_1, P_1 , I_2, J_2, K_2, L_2, M_2, N_2, O_2, P_2 . . .
sequence number for data at nodes I_1, J_1, K_1, L_1, M_1, N_1, O_1, P_1 , I_2, J_2, K_2, L_2, M_2, N_2, O_2, P_2, . . .

Table 273.2: SOLID273 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input																	
	Item	I_1	J_1	K_1	L_1	M_1	N_1	O_1	P_1	---	I_n	J_n	K_n	L_n	M_n	N_n	O_n	P_n
P_1	SMISC	2	1							---								
P_2	SMISC		4	3						---								
P_3	SMISC			6	5					---								
P_4	SMISC	7			8					---								
P_5	SMISC									---								
P_6	SMISC									---								
P_7	SMISC									---								

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input																	
	Item	I_1	J_1	K_1	L_1	M_1	N_1	O_1	P_1	---	I_n	J_n	K_n	L_n	M_n	N_n	O_n	P_n
P_8	SMISC									---								
---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---
P_{4n-3}	SMISC									---	8n-6	8n-7						
P_{4n-2}	SMISC									---		8n-4	8n-5					
P_{4n-1}	SMISC									---			8n-2	8n-3				
P_{4n}	SMISC									---	8n-1			8n				

SOLID273 Assumptions and Restrictions

- The area of the base element must be nonzero.
- The base element must lie on one side of the axisymmetric axis, and the axisymmetric axis must be on the same plane as the base element (master plane).
- A base element or base node must be associated with one axisymmetric axis (defined via **SECTYPE** and **SECDATA** commands) *before* generating nodes for general axisymmetric element sections (**NAXIS**) or defining an element by node connectivity (**E**).
- An edge with a removed midside node implies that the displacement varies linearly, rather than parabolically, along that edge. For more information about the use of midside nodes, see Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide*.
- Use at least two elements to avoid the hourglass effect.
- You can form a triangular base element by defining duplicate K_1 - L_1 - O_1 node numbers. (See Degenerated Shape Elements (p. 39) in this document.)
- You cannot apply a pressure load via the **SFA** command.
- If you specify mixed formulation (KEYOPT(6) = 1), do not miss any midside nodes. You must also use the sparse solver.
- The contribution of the element to the mass moment inertia of the whole model is calculated by element mass multiplied by the square of the coordinates of the elemental centroid. The moment of inertia may therefore be inaccurate.
- Issuing an **/ESHAPE**,1 command while PowerGraphics is active causes the program to plot the elements in 3-D and the results on both nodal planes and all integration planes in the circumferential direction; otherwise, the program plots the elements in 2-D and the results on the master plane.
- You cannot display surface load symbols (**/PSF**) when displaying this element in 3-D via the **/ESHAPE** command.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When specifying more than one facet per element edge for PowerGraphics displays (**NAXIS**,EFACET,*NUM*, where *NUM* > 1), ANSYS plots additional results on some planes between the nodal and integration planes. The results on these planes are interpolated based on the nodal and integration plane values and are therefore less accurate than the values on the nodal and interpolation planes. If you do not wish to plot the interpolated values, set *NUM* = 1 to plot only the values on nodal and integration planes.
- You cannot increase the number of planar facets per element edge (**/EFACET**) when displaying this element in 3-D via the **/ESHAPE** command.
- Print commands in postprocessing print the nodal plane results only.
- To model axisymmetric solid surface loads acting on this element, use general axisymmetric surface element SURF159. (You cannot use this element with surface-effect elements SURF153 and SURF154.)
- You cannot generate surface-based contact pairs (contact elements CONTA171 through CONTA174 paired with target elements TARGE169 and TARGE170) on this element.
- You *can* generate node-to-surface contact pairs (contact elements CONTA175 paired with target elements TARGE170) and node-to-node contact elements (CONTA178) on this element, with the following restrictions:
  - When TARGE170 is on the surface of SOLID273, you may have accuracy and convergence issues if the loading causes large rotations about the axisymmetric axis of SOLID273; you may also have those issues if the two sides of the contact boundaries have different mesh patterns in the circumferential direction (caused by different KEYOPT(2) values of SOLID273).
  - You cannot define CONTA175 with the multipoint constraint (MPC) approach using a force-distributed constraint (that is, you cannot set KEYOPT(2) = 2, KEYOPT(4) = 1, and KEYOPT(12) = 5 or 6 simultaneously for the CONTA175 elements).
  - In the case of a true axisymmetric condition (SOLID273 with KEYOPT(2) = 1), you should use 2-D node-to-surface contact pairs (CONTA175 contact elements paired with TARGE169 target elements).
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). Prestress effects can be activated via the **PSTRES** command.

SOLID273 Product Restrictions

No product-specific restrictions exist for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID278

3-D 8-Node Thermal Solid

MP ME <> PR PRN <> <> <> PP EME MFS
Product Restrictions

SOLID278 Element Description

SOLID278 has a 3-D thermal conduction capability. The element has eight nodes with a single degree of freedom, temperature, at each node. The element is applicable to a 3-D, steady-state or transient thermal analysis. If the model containing the conducting solid element is also to be analyzed structurally, the element should be replaced by an equivalent structural element (such as SOLID185). See SOLID279 for a similar thermal element, with mid-edge node capability.

SOLID278 is available in two forms:

- Homogeneous Thermal Solid (KEYOPT(3) = 0, the default) -- See "SOLID278 Homogeneous Thermal Solid Element Description" (p. 1321).
- Layered Thermal Solid (KEYOPT(3) = 1) -- See "SOLID278 Layered Thermal Solid Element Description" (p. 1326).

See SOLID278 in the *Mechanical APDL Theory Reference* for more details about this element.

A higher-order version of the SOLID278 element is SOLID279.

SOLID278 Homogeneous Thermal Solid Element Description

SOLID278 Thermal Solid is suitable for modeling general 3-D solid heat conduction. It allows for prism and tetrahedral degenerations when used in irregular regions. SOLID278 is designed to be a companion element for SOLID185.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 278.1: SOLID278 Homogeneous Thermal Solid Geometry

SOLID278 Homogeneous Thermal Solid Input Data

The geometry and node locations for this element are shown in Figure 278.1: SOLID278 Homogeneous Thermal Solid Geometry (p. 1322). The element is defined by eight nodes and the orthotropic material properties. The default element coordinate system is along global directions. You may define an element coordinate system using **ESYS**, which forms the basis for orthotropic material directions (namely, for thermal conductivity). Specific heat and density are ignored for steady-state solutions. Properties not input default as described in Linear Material Properties in the *Material Reference*.

As described in Coordinate Systems (p. 55), you can use **ESYS** to orient the material properties and the temperature gradient and heat flux output. Use **RSYS** to choose output that follows the material co-ordinate system or the global coordinate system.

Element loads are described in Nodal Loading (p. 49). Convection or heat flux (but not both) and radiation may be input as surface loads at the element faces as shown by the circled numbers on Figure 278.1: SOLID278 Homogeneous Thermal Solid Geometry (p. 1322). Heat generation rates may be input as element body loads at the nodes. If the node I heat generation rate HG(I) is input, and all others are unspecified, they default to HG(I).

"SOLID278 Homogeneous Thermal Solid Input Summary" (p. 1323) contains a summary of element input. For a general description of element input, see Element Input (p. 43).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID278 Homogeneous Thermal Solid Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
TEMP

Material Properties
MP command: KXX, KYY, KZZ, DENS, C

Surface Loads

Convection or Heat Flux (but not both) and Radiation (using Lab = RDSF) --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Heat Generations --
HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P)

Special Features
Birth and death

KEYOPT(3)
Layer construction:

0 --
homogeneous Solid (default) -- nonlayered

1 --
Layered Solid

KEYOPT(8)
Layer data storage. Only applicable for Layered Solid (KEYOPT(3) = 1).

SOLID278 Homogeneous Thermal Solid Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output as shown in Table 278.1: SOLID278 Homogeneous Thermal Solid Output Definitions (p. 1324)

Convection heat flux is positive out of the element; applied heat flux is positive into the element. The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 278.1: SOLID278 Homogeneous Thermal Solid Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	2
HGEN	Heat generations HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P)	Y	-
TG:X, Y, Z	Thermal gradient components	Y	Y
TF:X, Y, Z	Thermal flux (heat flow rate/cross-sectional area) components	Y	Y
FACE	Face label	1	-
AREA	Face area	1	1
NODES	Face nodes	1	-
HFILM	Film coefficient at each node of face	1	-
TBULK	Bulk temperature at each node of face	1	-
TAVG	Average face temperature	1	1
HEAT RATE	Heat flow rate across face by convection	1	1
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	1	-
HFAVG	Average film coefficient of the face	-	1
TBAVG	Average face bulk temperature	-	1
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	1
HFLUX	Heat flux at each node of face	1	-

1. Output if a surface load is input

2. Available only at centroid as a ***GET** item.

Table 278.2: SOLID278 homogeneous Thermal Solid Item and Sequence Numbers (p. 1325) lists output available via **ETABLE** using the Sequence Number method. See Element Table for Variables Identified By Sequence Number in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 278.2: SOLID278 homogeneous Thermal Solid Item and Sequence Numbers (p. 1325):

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name

output quantity as defined in the Table 278.1: SOLID278 Homogeneous Thermal Solid Output Definitions (p. 1324)

Item

predetermined Item label for **ETABLE** command

FC*n*

sequence number for solution items for element Face *n*

Table 278.2: SOLID278 homogeneous Thermal Solid Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input						
	Item	FC1	FC2	FC3	FC4	FC5	FC6
AREA	NMISC	1	7	13	19	25	31
HFAVG	NMISC	2	8	14	20	26	32
TAVG	NMISC	3	9	15	21	27	33
TBAVG	NMISC	4	10	16	22	28	34
HEAT RATE	NMISC	5	11	17	23	29	35
HFLXAVG	NMISC	6	12	18	24	30	36

SOLID278 Homogeneous Thermal Solid Assumptions and Restrictions

- Zero-volume elements are not allowed.
- Elements may be numbered either as shown in Figure 278.1: SOLID278 Homogeneous Thermal Solid Geometry (p. 1322) or may have the planes IJKL and MNOP interchanged. The element may not be twisted such that the element has two separate volumes (which occurs most frequently when the elements are not numbered properly).
- All elements must have eight nodes. You can form a prism-shaped element by defining duplicate K and L and duplicate O and P node numbers. (See Degenerated Shape Elements (p. 39).) A tetrahedron shape is also available.
- If the thermal element is to be replaced by a SOLID185 structural element with surface stresses requested, the thermal element should be oriented such that face I-J-N-M and/or face K-L-P-O is a free surface.
- A free surface of the element (that is, not adjacent to another element and not subjected to a boundary constraint) is assumed to be adiabatic.

This element has a layered option (KEYOPT(3) = 1). See "SOLID278 Layered Thermal Solid Assumptions and Restrictions" (p. 1330) for additional information.

SOLID278 Homogeneous Thermal Solid Product Restrictions

None.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID278 Layered Thermal Solid Element Description

Use SOLID278 Layered Thermal Solid to model heat conduction in layered thick shells or solids. The layered section definition is given by ANSYS section (SEC*xxx*) commands. A prism degeneration option is also available.

Figure 278.2: SOLID278 Layered Thermal Solid Geometry

x_o = Element x-axis if ESYS is not supplied.

x = Element x-axis if ESYS is supplied.

SOLID278 Layered Thermal Solid Input Data

The geometry and node locations for this element are shown in Figure 278.2: SOLID278 Layered Thermal Solid Geometry (p. 1326). The element is defined by eight nodes. A prism-shaped element may be formed by defining the same node numbers for nodes K and L, and O and P.

In addition to the nodes, the element input data includes the anisotropic material properties. Anisotropic material directions correspond to the layer coordinate directions which are based on the element coordinate system. The element coordinate system follows the shell convention where the z axis is normal to the surface of the shell. The nodal ordering must follow the convention that I-J-K-L and M-N-O-P element faces represent the bottom and top shell surfaces, respectively. You can change the orientation within the plane of the layers via the **SECDATA** command in the same way that you would for shell elements (as described in Coordinate Systems (p. 55)). To achieve the correct nodal ordering for a volume mapped (hexahedron) mesh, you can use the **VEORIENT** command to specify the desired volume orientation before executing the **VMESH** command. Alternatively, you can use the **EORIENT** command after automatic meshing to reorient the elements to be in line with the orientation of another element, or to be as parallel as possible to a defined **ESYS** axis.

Layered Section Definition Using Section Commands

You can associate SOLID278 Layered Solid with a shell section (**SECTYPE**). The layered composite specifications (including layer thickness, material, orientation, and number of integration points through

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the thickness of the layer) are specified via shell section (SEC*xxx*) commands. You can use the shell section commands even with a single-layered element. ANSYS obtains the actual layer thicknesses used for element calculations by scaling the input layer thickness so that they are consistent with the thickness between the nodes.

You can designate the number of integration points (1, 3, 5, 7, or 9) located through the thickness of each layer. Two points are located on the top and bottom surfaces respectively and the remaining points are distributed equal distance between the two points. The element requires at least two points through the entire thickness. When no shell section definition is provided, the element is treated as single-layered and uses two integration points through the thickness.

SOLID278 Layered Thermal Solid does not support real constant input for defining layer sections.

Other Input

The default orientation for this element has the S1 (shell surface coordinate) axis aligned with the first parametric direction of the element at the center of the element and is shown as x_o in Figure 278.2: SOLID278 Layered Thermal Solid Geometry (p. 1326).

The default first surface direction S1 can be reoriented in the element reference plane (as shown in Figure 278.2: SOLID278 Layered Thermal Solid Geometry (p. 1326)) via the **ESYS** command. You can further rotate S1 by angle THETA (in degrees) for each layer via the **SECDATA** command to create layer-wise coordinate systems. See Coordinate Systems (p. 55) for details.

The geometry, node locations, and the coordinate system for this element are shown in Figure 278.2: SOLID278 Layered Thermal Solid Geometry (p. 1326). The element is defined by eight nodes and the orthotropic material properties. A prism-shaped element may also be formed as shown in Figure 278.2: SOLID278 Layered Thermal Solid Geometry (p. 1326). Orthotropic material directions correspond to the layer coordinate directions. The element coordinate system orientation is as described in Coordinate Systems (p. 55). Specific heat and density are ignored for steady-state solutions. Properties not input default as described in Linear Material Properties in the *Material Reference*.

Element loads are described in Nodal Loading (p. 49). Convection or heat flux (but not both) and radiation may be input as surface loads at the element faces as shown by the circled numbers on Figure 278.2: SOLID278 Layered Thermal Solid Geometry (p. 1326). Heat generation rates may be input as element body loads on a per layer basis. One heat generation value is applied to the entire layer. If the first layer heat generation rate HG(1) is input, and all others are unspecified, they default to HG(1).

The following table summarizes the element input. Element Input (p. 43) provides a general description of element input.

SOLID278 Layered Thermal Solid Input Summary

Nodes
I, J, K, L, M, N, O, P

Degrees of Freedom
TEMP

Material Properties
KXX, KYY, KZZ, DENS, C

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface Loads

Convection or Heat Flux (but not both) and Radiation (using Lab = RDSF) --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Heat Generations --
HG(1), HG(2), HG(3), . . . , HG(number of layers)

Special Features
Birth and death

KEYOPT(3)
Layer construction:

0 --
homogeneous Solid (default) -- nonlayered

1 --
Layered Solid

KEYOPT(8)
Layer data storage:

0 --
Store data for bottom of bottom layer and top of top layer (default)

1 --
Store top and bottom data for all layers. (The volume of data may be considerable.)

SOLID278 Layered Thermal Solid Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output as shown in Table 278.3: SOLID278 Layered Thermal Solid Element Output Definitions (p. 1329)

Convection heat flux is positive out of the element; applied heat flux is positive into the element. The element output directions are parallel to the layer coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 278.3: SOLID278 Layered Thermal Solid Heat Flux/Temperature Gradient Output

Heat flux directions shown are for global directions.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 278.3: SOLID278 Layered Thermal Solid Element Output Definitions

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	2
HGEN	Heat generations HG(1), HG(2), HG(3), . . .	Y	-
TG:X, Y, Z	Thermal gradient components	Y	Y
TF:X, Y, Z	Thermal flux (heat flow rate/cross-sectional area) components	Y	Y
FACE	Face label	1	-
AREA	Face area	1	1
NODES	Face nodes	1	-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
HFILM	Film coefficient at each node of face	1	-
TBULK	Bulk temperature at each node of face	1	-
TAVG	Average face temperature	1	1
HEAT RATE	Heat flow rate across face by convection	1	1
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	1	-
HFAVG	Average film coefficient of the face	-	1
TBAVG	Average face bulk temperature	-	1
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	1
HFLUX	Heat flux at each node of face	1	-

1. Output if a surface load is input

2. Available only at centroid as a ***GET** item.

Table 278.4: SOLID278 Layered Thermal Solid Item and Sequence Numbers (p. 1330) lists output available via **ETABLE** using the Sequence Number method. See Element Table for Variables Identified By Sequence Number in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this manual for more information. The following notation is used in Table 278.4: SOLID278 Layered Thermal Solid Item and Sequence Numbers (p. 1330):

Name
output quantity as defined in Table 278.3: SOLID278 Layered Thermal Solid Element Output Definitions (p. 1329)

Item
predetermined Item label for **ETABLE** command

FC*n*
sequence number for solution items for element Face *n*

Table 278.4: SOLID278 Layered Thermal Solid Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input						
	Item	FC1	FC2	FC3	FC4	FC5	FC6
AREA	NMISC	1	7	13	19	25	31
HFAVG	NMISC	2	8	14	20	26	32
TAVG	NMISC	3	9	15	21	27	33
TBAVG	NMISC	4	10	16	22	28	34
HEAT RATE	NMISC	5	11	17	23	29	35
HFLXAVG	NMISC	6	12	18	24	30	36

SOLID278 Layered Thermal Solid Assumptions and Restrictions

- Zero-volume elements are not allowed.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Elements may be numbered either as shown in Figure 278.2: SOLID278 Layered Thermal Solid Geometry (p. 1326) or may have the planes IJKL and MNOP interchanged. The element may not be twisted such that the element has two separate volumes (which occurs most frequently when the elements are not numbered properly).
- All elements must have eight nodes. You can form a prism-shaped element by defining duplicate K and L and duplicate O and P node numbers. (See Degenerated Shape Elements (p. 39).)
- If the thermal element is to be replaced by a SOLID185 structural element with surface stresses requested, the thermal element should be oriented such that face I-J-N-M and/or face K-L-P-O is a free surface.
- A free surface of the element (that is, not adjacent to another element and not subjected to a boundary constraint) is assumed to be adiabatic.
- This element is primarily intended for conveniently modeling the in-plane effects in layered thick shells or solids. The in-plane conductivity is the average of the individual layer conductivities. For complicated through-thickness behaviors, consider using multiple layers of homogeneous (non-layered) SOLID278 elements, or use SHELL131, the layered shell with through-thickness degrees of freedom.

SOLID278 Layered Thermal Solid Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID279

3-D 20-Node Thermal Solid

MP ME <> PR PRN <> <> <> PP EME MFS
Product Restrictions

SOLID279 Element Description

SOLID279 is a higher order 3-D 20-node solid element that exhibits quadratic thermal behavior. The element is defined by 20 nodes with a temperature degree of freedom at each node.

SOLID279 is available in two forms:

- homogeneous Thermal Solid (KEYOPT(3) = 0, the default) -- See "SOLID279 homogeneous Thermal Solid Element Description " (p. 1333).
- Layered Thermal Solid (KEYOPT(3) = 1) -- See "SOLID279 Layered Thermal Solid Element Description" (p. 1338).

A lower-order version of the SOLID279 element is SOLID278.

SOLID279 homogeneous Thermal Solid Element Description

SOLID279 homogeneous Thermal Solid is well suited to modeling irregular meshes (such as those produced by various CAD/CAM systems). The element may have any spatial orientation.

Various printout options are available. See SOLID279 in the *Mechanical APDL Theory Reference* for more details.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 279.1: SOLID279 homogeneous Thermal Solid Geometry

SOLID279 homogeneous Thermal Solid Input Data

The geometry, node locations, and the element coordinate system for this element are shown in Figure 279.1: SOLID279 homogeneous Thermal Solid Geometry (p. 1334). A prism-shaped element may be formed by defining the same node numbers for nodes K, L, and S; nodes A and B; and nodes O, P, and W. A tetrahedral-shaped element and a pyramid-shaped element may also be formed as shown in Figure 279.1: SOLID279 homogeneous Thermal Solid Geometry (p. 1334). SOLID278 is a similar element, without mid-side nodes.

In addition to the nodes, the element input data includes the anisotropic material properties. Anisotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems (p. 55).

As described in Coordinate Systems (p. 55), you can use **ESYS** to orient the material properties and the temperature gradient and heat flux output. Use **RSYS** to choose output that follows the material coordinate system or the global coordinate system.

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems (p. 55). Specific heat and density are ignored for steady-state solutions. Properties not input default as described in Linear Material Properties in the *Material Reference*.

Element loads are described in Nodal Loading (p. 49). Convection or heat flux (but not both) and radiation may be input as surface loads at the element faces as shown by the circled numbers on Figure 279.1: SOLID279 homogeneous Thermal Solid Geometry (p. 1334). Heat generation rates may be input as element body loads at the nodes. If the node I heat generation rate HG(I) is input, and all others are unspecified,

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

they default to HG(I). If all corner node heat generation rates are specified, each midside node heat generation rate defaults to the average heat generation rate of its adjacent corner nodes.

The following table summarizes the element input. Element Input (p. 43) provides a general description of element input.

SOLID279 homogeneous Thermal Solid Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B

Degrees of Freedom
TEMP

Real Constants
None

Material Properties
MP command: KXX, KYY, KZZ, DENS, C

Surface Loads

Convection or Heat Flux (but not both) and Radiation (using Lab = RDSF) --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Heat Generations --

HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P), HG(Q), HG(R),
HG(S), HG(T), HG(U), HG(V), HG(W), HG(X), HG(Y), HG(Z), HG(A), HG(B)

Special Features
Birth and death

KEYOPT(3)
Layer construction:

0 --
homogeneous Solid (default) -- nonlayered

1 --
Layered Solid

KEYOPT(8)
Layer data storage. Only applicable for Layered Solid (KEYOPT(3) = 1).

SOLID279 homogeneous Thermal Solid Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Additional element output as shown in Table 279.1: SOLID279 Element Output Definitions (p. 1336)

Figure 279.2: SOLID279 homogeneous Thermal Solid Temperature Gradient/Heat Flux Output

The element heat flux directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 279.1: SOLID279 Element Output Definitions

Label	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	2
HGEN	Heat generations HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P), HG(Q), ..., HG(Z), HG(A), HG(B)	Y	-
TG:X, Y, Z	Thermal gradient components	Y	Y
TF:X, Y, Z	Thermal flux (heat flow rate/cross-sectional area) components	Y	Y
FACE	Face label	1	-
NODES	Corner nodes on this face	1	-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Label	Definition	O	R
AREA	Face area	1	1
HFILM	Film coefficient	1	-
TAVG	Average face temperature	1	1
TBULK	Fluid bulk temperature	1	-
HEAT RATE	Heat flow rate across face by convection	1	1
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	1	-
HFLUX	Heat flux at each node of face	1	-
HFAVG	Average film coefficient of the face	-	1
TBAVG	Average face bulk temperature	-	1
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	1

1. Output only if a surface load is input

2. Available only at centroid as a ***GET** item.

Table 279.2: SOLID279 Item and Sequence Numbers (p. 1337) lists output available through **ETABLE** using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this document for more information. The following notation is used in Table 279.2: SOLID279 Item and Sequence Numbers (p. 1337):

Name
output quantity as defined in Table 279.1: SOLID279 Element Output Definitions (p. 1336)

Item
predetermined Item label for **ETABLE**

FC*n*
sequence number for solution items for element Face *n*

Table 279.2: SOLID279 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input						
	Item	FC1	FC2	FC3	FC4	FC5	FC6
AREA	NMISC	1	7	13	19	25	31
HFAVG	NMISC	2	8	14	20	26	32
TAVG	NMISC	3	9	15	21	27	33
TBAVG	NMISC	4	10	16	22	28	34
HEAT RATE	NMISC	5	11	17	23	29	35
HFLXAVG	NMISC	6	12	18	24	30	36

SOLID279 homogeneous Thermal Solid Assumptions and Restrictions

- The element must not have a zero volume. This occurs most frequently when the element is not numbered properly.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Elements may be numbered either as shown in Figure 279.1: SOLID279 homogeneous Thermal Solid Geometry (p. 1334) or may have the planes IJKL and MNOP interchanged.
- The condensed face of a prism-shaped element should not be defined as a convection face.
- The specific heat is evaluated at each integration point to allow for abrupt changes (such as melting) within a coarse grid of elements.
- If the thermal element is to be replaced by a SOLID186 structural element with surface stresses requested, the thermal element should be oriented such that face IJNM and/or face KLPO is a free surface.
- A free surface of the element (i.e., not adjacent to another element and not subjected to a boundary constraint) is assumed to be adiabatic.
- Thermal transients having a fine integration time step and a severe thermal gradient at the surface will also require a fine mesh at the surface.
- An edge with a removed midside node implies that the temperature varies linearly, rather than parabolically, along that edge.
- See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.
- For transient solutions using the **THOPT**,QUASI option, the program removes the midside nodes from any face with a convection load. A temperature solution is not available for them. Do not use the midside nodes on these faces in constraint equations or with contact. If you use these faces for those situations, remove the midside nodes first.
- Degeneration to the form of pyramid should be used with caution.
- The element sizes, when degenerated, should be small in order to minimize the field gradients.
- Pyramid elements are best used as filler elements or in meshing transition zones.

This element has a layered option (KEYOPT(3) = 1). See "SOLID279 Layered Thermal Solid Assumptions and Restrictions" (p. 1344) for additional information.

SOLID279 homogeneous Thermal Solid Product Restrictions

None.

SOLID279 Layered Thermal Solid Element Description

Use SOLID279 Layered Thermal Solid to model heat conduction in layered thick shells or solids. The layered section definition is given by ANSYS section (SEC*xxx*) commands. A prism degeneration option is also available.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 279.3: SOLID279 Layered Thermal Solid Geometry

x_o = Element x-axis if ESYS is not supplied.

x = Element x-axis if ESYS is supplied.

SOLID279 Layered Thermal Solid Input Data

The geometry, node locations, and the element coordinate system for this element are shown in Figure 279.3: SOLID279 Layered Thermal Solid Geometry (p. 1339). A prism-shaped element may be formed by defining the same node numbers for nodes K, L, and S; nodes A and B; and nodes O, P, and W.

In addition to the nodes, the element input data includes the anisotropic material properties. Anisotropic material directions correspond to the layer coordinate directions which are based on the element coordinate system. The element coordinate system follows the shell convention where the z axis is normal to the surface of the shell. The nodal ordering must follow the convention that I-J-K-L and M-N-O-P element faces represent the bottom and top shell surfaces, respectively. You can change the orientation within the plane of the layers via the **ESYS** command in the same way that you would for shell elements (as described in Coordinate Systems (p. 55)). To achieve the correct nodal ordering for a volume mapped (hexahedron) mesh, you can use the **VEORIENT** command to specify the desired volume orientation before executing the **VMESH** command. Alternatively, you can use the **EORIENT** command after automatic meshing to reorient the elements to be in line with the orientation of another element, or to be as parallel as possible to a defined **ESYS** axis.

Layered Section Definition Using Section Commands

You can associate SOLID279 Layered Thermal Solid with a shell section (**SECTYPE**). The layered composite specifications (including layer thickness, material, orientation, and number of integration points through the thickness of the layer) are specified via shell section (SEC*xxx*) commands. You can use the shell section commands even with a single-layered element. ANSYS obtains the actual layer thicknesses used for element calculations by scaling the input layer thickness so that they are consistent with the thickness between the nodes.

You can designate the number of integration points (1, 3, 5, 7, or 9) located through the thickness of each layer. Two points are located on the top and bottom surfaces respectively and the remaining

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

points are distributed equal distance between the two points. The element requires at least two points through the entire thickness. When no shell section definition is provided, the element is treated as single-layered and uses two integration points through the thickness.

SOLID279 Layered Thermal Solid does not support real constant input for defining layer sections.

Other Input

The default orientation for this element has the S1 (shell surface coordinate) axis aligned with the first parametric direction of the element at the center of the element and is shown as x_o in Figure 279.3: SOLID279 Layered Thermal Solid Geometry (p. 1339).

The default first surface direction S1 can be reoriented in the element reference plane (as shown in Figure 279.3: SOLID279 Layered Thermal Solid Geometry (p. 1339)) via the **ESYS** command. You can further rotate S1 by angle THETA (in degrees) for each layer via the **SECDATA** command to create layer-wise coordinate systems. See Coordinate Systems (p. 55) for details.

Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems (p. 55). Specific heat and density are ignored for steady-state solutions. Properties not input default as described in Linear Material Properties in the *Material Reference*.

Element loads are described in Nodal Loading (p. 49). Convection or heat flux (but not both) and radiation may be input as surface loads at the element faces as shown by the circled numbers on Figure 279.3: SOLID279 Layered Thermal Solid Geometry (p. 1339). Heat generation rates may be input as element body loads on a per layer basis. One heat generation value is applied to the entire layer. If the first layer heat generation rate HG(1) is input, and all others are unspecified, they default to HG(1).

As described in Coordinate Systems (p. 55), you can use the **ESYS** command to orient the material properties and temperature gradient/heat flux output. Use **RSYS** to choose output that follows the material coordinate system or the global coordinate system.

The following table summarizes the element input. Element Input (p. 43) provides a general description of element input.

SOLID279 Layered Thermal Solid Input Summary

Nodes
I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B

Degrees of Freedom
TEMP

Real Constants
None

Material Properties
KXX, KYY, KZZ, DENS, C, ENTH

Surface Loads

Convection or Heat Flux (but not both) and Radiation (using Lab = RDSF) --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Body Loads

Heat Generations --
HG(1), HG(2), HG(3), . . . , HG(number of layers)

Special Features
Birth and death

KEYOPT(3)
Layer construction:

0 --
homogeneous Solid (default) -- nonlayered

1 --
Layered Solid

KEYOPT(8)
Layer data storage:

0 --
Store data for bottom of bottom layer and top of top layer

1 --
Store top and bottom data for all layers. (The volume of data may be excessive.)

SOLID279 Layered Thermal Solid Output Data

The solution output associated with the element is in two forms:

- Nodal temperatures included in the overall nodal solution
- Additional element output as shown in Figure 279.4: SOLID279 Layered Thermal Solid Temperature Gradient/Heat Flux Output (p. 1342)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 279.4: SOLID279 Layered Thermal Solid Temperature Gradient/Heat Flux Output

The element heat flux directions are parallel to the layer coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

Convection heat flux is positive out of the element; applied heat flux is positive into the element. The element output directions are parallel to the element coordinate system. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to view results.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 279.3: SOLID279 Element Output Definitions

Label	Definition	O	R
EL	Element Number	Y	Y
NODES	Nodes - I, J, K, L, M, N, O, P	Y	Y
MAT	Material number	Y	Y
VOLU:	Volume	Y	Y
XC, YC, ZC	Location where results are reported	Y	2
TG:X, Y, Z	Thermal gradient components	Y	Y
TF:X, Y, Z	Thermal flux (heat flow rate/cross-sectional area) components	Y	Y
FACE	Face label	1	-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Label	Definition	O	R
NODES	Corner nodes on this face	1	-
AREA	Face area	1	1
HFILM	Film coefficient	1	-
TAVG	Average face temperature	1	1
TBULK	Fluid bulk temperature	1	-
HEAT RATE	Heat flow rate across face by convection	1	1
HEAT RATE/AREA	Heat flow rate per unit area across face by convection	1	-
HFLUX	Heat flux at each node of face	1	-
HFAVG	Average film coefficient of the face	-	1
TBAVG	Average face bulk temperature	-	1
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux	-	1

1. Output only if a surface load is input

2. Available only at centroid as a ***GET** item.

Table 279.4: SOLID279 Item and Sequence Numbers (p. 1343) lists output available via **ETABLE** using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this document for more information. The following notation is used in Table 279.4: SOLID279 Item and Sequence Numbers (p. 1343):

Name
output quantity as defined in Table 279.3: SOLID279 Element Output Definitions (p. 1342)

Item
predetermined Item label for **ETABLE**

FC*n*
sequence number for solution items for element Face *n*

Table 279.4: SOLID279 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input						
	Item	FC1	FC2	FC3	FC4	FC5	FC6
AREA	NMISC	1	7	13	19	25	31
HFAVG	NMISC	2	8	14	20	26	32
TAVG	NMISC	3	9	15	21	27	33
TBAVG	NMISC	4	10	16	22	28	34
HEAT RATE	NMISC	5	11	17	23	29	35
HFLXAVG	NMISC	6	12	18	24	30	36

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID279 Layered Thermal Solid Assumptions and Restrictions

- The element must not have a zero volume. This occurs most frequently when the element is not numbered properly.
- Elements may be numbered either as shown in Figure 279.3: SOLID279 Layered Thermal Solid Geometry (p. 1339) or may have the planes IJKL and MNOP interchanged.
- This element is primarily intended for conveniently modeling the in-plane effects in layered thick shells or solids. The in-plane conductivity is the average of the individual layer conductivities. For complicated through-thickness behaviors, consider using multiple layers of homogeneous (non-layered) SOLID279 elements, or use SHELL132, the layered shell with through-thickness degrees of freedom.
- The condensed face of a prism-shaped element should not be defined as a convection face.
- The specific heat is evaluated at each integration point to allow for abrupt changes (such as melting) within a coarse grid of elements.
- If the thermal element is to be replaced by a SOLID186 structural element with surface stresses requested, the thermal element should be oriented such that face IJNM and/or face KLPO is a free surface.
- A free surface of the element (i.e., not adjacent to another element and not subjected to a boundary constraint) is assumed to be adiabatic.
- Thermal transients having a fine integration time step and a severe thermal gradient at the surface will also require a fine mesh at the surface.
- An edge with a removed midside node implies that the temperature varies linearly, rather than parabolically, along that edge.
- See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.
- For transient solutions using the **THOPT**,QUASI option, the program removes the midside nodes from any face with a convection load. A temperature solution is not available for them. Do not use the midside nodes on these faces in constraint equations or with contact. If you use these faces for those situations, remove the midside nodes first.

SOLID279 Layered Thermal Solid Product Restrictions

None.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SHELL281

8-Node Structural Shell

MP ME ST PR PRN DS <> <> PP EME MFS
Product Restrictions

SHELL281 Element Description

SHELL281 is suitable for analyzing thin to moderately-thick shell structures. The element has eight nodes with six degrees of freedom at each node: translations in the x, y, and z axes, and rotations about the x, y, and z-axes. (When using the membrane option, the element has translational degrees of freedom only.)

SHELL281 is well-suited for linear, large rotation, and/or large strain nonlinear applications. Change in shell thickness is accounted for in nonlinear analyses. The element accounts for follower (load stiffness) effects of distributed pressures.

SHELL281 may be used for layered applications for modeling composite shells or sandwich construction. The accuracy in modeling composite shells is governed by the first-order shear-deformation theory (usually referred to as Mindlin-Reissner shell theory).

The element formulation is based on logarithmic strain and true stress measures. The element kinematics allow for finite membrane strains (stretching). However, the curvature changes within a time increment are assumed to be small.

See SHELL281 in the *Mechanical APDL Theory Reference* for more details about this element.

SHELL281 Input Data

The following figure shows the geometry, node locations, and the element coordinate system for this element. The element is defined by shell section information and by eight nodes (I, J, K, L, M, N, O and P).

Midside nodes may not be removed from this element. See Quadratic Elements (Midside Nodes) in the *Modeling and Meshing Guide* for more information about the use of midside nodes.

A triangular-shaped element may be formed by defining the same node number for nodes K, L and O.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 281.1: SHELL281 Geometry

x_o = Element x-axis if element orientation (**ESYS**) is not provided.

x = Element x-axis if element orientation is provided.

Single-Layer Definition

To define the thickness (and other information), use section definition, as follows:

SECTYPE,,SHELL
SECDATA,*THICKNESS*, ...

A single-layer shell section definition provides flexible options. For example, you can specify the number of integration points used and the material orientation.

Multilayer Definition

The shell section commands allow for layered shell definition. Options are available for specifying the thickness, material, orientation, and number of integration points through the thickness of the layers.

You can designate the number of integration points (1, 3, 5, 7, or 9) located through the thickness of each layer when using section input. When only one, the point is always located midway between the top and bottom surfaces. If three or more points, two points are located on the top and bottom surfaces respectively and the remaining points are distributed at equal distances between the two points. The default number of integration points for each layer is three; however, when a single layer is defined and plasticity is present, the number of integration points is changed to a minimum of five during solution.

The following additional capabilities are available when defining shell layers:

- SHELL281 accepts the preintegrated shell section type (**SECTYPE**,,GENS).

  When the element is associated with the GENS section type, thickness or material definitions are not required.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You can use the function tool to define thickness as a function of global/local coordinates or node numbers (**SECFUNCTION**).
- You can specify offsets (**SECOFFSET**).
- A section can be partially defined using data from a FiberSIM `.xml` file (**SECTYPE**,,SHELL,FIBERSIM).

Other Input

The default orientation for this element has the S1 (shell surface coordinate) axis aligned with the first parametric direction of the element at the four in-plane integration points:

$$S_1 = \frac{\partial\{x\}}{\partial s} \Big/ \left(\left| \frac{\partial\{x\}}{\partial s} \right| \right)$$

Where element geometry {x} is given as follows:

$$\{x\} = \sum_{i=1}^{8} h^i(s,r)\{x\}^i$$

(s,r) = two isoparametric element coordinates

h^I (s,r) = shape functions associated with eight element nodes (see *Mechanical APDL Theory Reference* for more details)

$\{x\}^i$ = global coordinates of eight element nodes

For undistorted elements, the default orientation is the same as described in Coordinate Systems (p. 55) (the first surface direction is aligned with the IJ side).

The first surface direction S_1 can be rotated by angle θ (in degrees) for the layer via the **SECDATA** command. For an element, you can specify a single value of orientation in the plane of the element. Layer-wise orientation is supported.

You can also define element orientation via the **ESYS** command. For more information, see Coordinate Systems (p. 55).

The element supports degeneration into a triangular form. For better accuracy, however, ANSYS, Inc. recommends quadrilateral shaped elements. The triangle form is generally more robust when using the membrane option with large deflections.

To evaluate stresses and strains on exterior surfaces, use KEYOPT(1) = 2. When used as overlaid elements on the faces of 3-D elements, this option is similar to the surface stress option (described in the *Mechanical APDL Theory Reference*), but is more general and applicable to nonlinear analysis. The element used with this option does not provide any stiffness, mass, or load contributions. This option should only be used in single-layer shells. Irrespective of other settings, SHELL281 provides stress and strain output at the four in-plane integration points of the layer.

SHELL281uses a penalty method to relate the independent rotational degrees of freedom about the normal (to the shell surface) with the in-plane components of displacements. The program chooses an appropriate penalty stiffness by default. A drill stiffness factor can be specified via the **SECCONTROL** command.

Element loads are described in Nodal Loading (p. 49). Pressures may be input as surface loads on the element faces as shown by the circled numbers in Figure 281.1: SHELL281 Geometry (p. 1346). Positive

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

pressures act into the element. Because shell edge pressures are input on a per-unit-length basis, per-unit-area quantities must be multiplied by the shell thickness.

Temperatures may be input as element body loads at the corners of the outside faces of the element and at the corners of the interfaces between layers. The first corner temperature T1 defaults to TUNIF. If all other temperatures are unspecified, they default to T1. If KEYOPT(1) = 0 and if exactly NL+1 temperatures are input, one temperature is used for the four bottom corners of each layer, and the last temperature is used for the four top corner temperatures of the top layer. If KEYOPT(1) = 1 and if exactly NL temperatures are input, one temperature is used for the four corners of each layer. That is, T1 is used for T1, T2, T3, and T4; T2 (as input) is used for T5, T6, T7, and T8, etc. For any other input pattern, unspecified temperatures default to TUNIF.

SHELL281 includes the effects of transverse shear deformation. The transverse shear stiffness of the element is a 2x2 matrix as shown below:

$$E = \begin{bmatrix} E_{11} & E_{12} \\ \text{sym} & E_{22} \end{bmatrix}$$

To define transverse shear stiffness values, use the **SECCONTROL** command .

For a single-layer shell with isotropic material, default transverse shear stiffnesses are:

$$E = \begin{bmatrix} kGh & 0 \\ 0 & kGh \end{bmatrix}$$

In the above matrix, k = 5/6, G = shear modulus, and h = thickness of the shell.

SHELL281 can be associated with linear elastic, elastoplastic, creep, or hyperelastic material properties. Only isotropic, anisotropic, and orthotropic linear elastic properties can be input for elasticity. The von Mises isotropic hardening plasticity models can be invoked with BISO (bilinear isotropic hardening), MISO (multilinear isotropic hardening), and NLISO (nonlinear isotropic hardening) options. The kinematic hardening plasticity models can be invoked with BKIN (bilinear kinematic hardening), MKIN and KINH (multilinear kinematic hardening), and CHABOCHE (nonlinear kinematic hardening). Invoking plasticity assumes that the elastic properties are isotropic (that is, if orthotropic elasticity is used with plasticity, ANSYS assumes the isotropic elastic modulus = EX and Poisson's ratio = NUXY).

Hyperelastic material properties (2, 3, 5, or 9 parameter Mooney-Rivlin material model, Neo-Hookean model, Polynomial form model, Arruda-Boyce model, and user-defined model) can be used with this element. Poisson's ratio is used to specify the compressibility of the material. If less than 0, Poisson's ratio is set to 0; if greater than or equal to 0.5, Poisson's ratio is set to 0.5 (fully incompressible).

Both isotropic and orthotropic thermal expansion coefficients can be input using **MP**,ALPX. When used with hyperelasticity, isotropic expansion is assumed.

Use the **TREF** command to specify the global value of reference temperature. If **MP**,REFT is defined for the material number of the element, it is used for the element instead of the value from the **TREF** command. But if **MP**,REFT is defined for the material number of the layer, it is used instead of either the global or element value.

SHELL281 uses an advanced shell formulation that accurately incorporates initial curvature effects. The calculation for effective shell curvature change accounts for both shell-membrane and thickness strains.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The new formulation generally offers improved accuracy in curved shell structure simulations, especially when thickness strain is significant or the material anisotropy in the thickness direction cannot be ignored.

KEYOPT(8) = 2 is used to store midsurface results in the results file for single or multi-layer shell elements. If you use **SHELL**,MID, you will see these calculated values, rather than the average of the TOP and BOTTOM results. Use this option to access these correct midsurface results (membrane results) for those analyses where averaging TOP and BOTTOM results is inappropriate; examples include midsurface stresses and strains with nonlinear material behavior, and midsurface results after mode combinations that involve squaring operations such as in spectrum analyses.

KEYOPT(9) = 1 is used to read initial thickness data from a user subroutine.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

A summary of the element input is given in "SHELL281 Input Summary" (p. 1349). A general description of element input is given in Element Input (p. 43).

SHELL281 Input Summary

Nodes

I, J, K, L, M, N, O, P

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ if KEYOPT(1) = 0

UX, UY, UZ if KEYOPT(1) = 1

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, EY, EZ, (PRXY, PRYZ, PRXZ, or NUXY, NUYZ, NUXZ), ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ), DENS, GXY, GYZ, GXZ, ALPD
Specify BETD, ALPD, and DMPR for the element (all layers) by issuing the **MAT** command to assign the material property set. REFT may be specified once for the element, or may be assigned on a per-layer basis. See the discussion in "SHELL281 Input Summary" (p. 1349) for more information.

Surface Loads

Pressures --

face 1 (I-J-K-L) (bottom, in +N direction),
face 2 (I-J-K-L) (top, in -N direction),
face 3 (J-I), face 4 (K-J), face 5 (L-K), face 6 (I-L)

Body Loads

Temperatures --

For KEYOPT(1) = 0 (Bending and membrane stiffness):

T1, T2, T3, T4 (at bottom of layer 1), T5, T6, T7, T8 (between layers 1-2); similarly for between next layers, ending with temperatures at top of layer NL(4*(NL+1) maximum). Hence, for one-layer elements, eight temperatures are used.

For KEYOPT(1) = 1 (Membrane stiffness only):

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

T1, T2, T3, T4 for layer 1, T5, T6, T7, T8 for layer 2, similarly for all layers (4*NL maximum). Hence, for one-layer elements, four temperatures are used.

Special Features

Birth and death
Element technology autoselect
Initial state
Large deflection
Large strain
Linear perturbation
Nonlinear stabilization
Section definition for layered shells and preintegrated general shell sections for input of homogeneous section stiffnesses

KEYOPT(1)
Element stiffness:

0 --
Bending and membrane stiffness (default)

1 --
Membrane stiffness only

2 --
Stress/strain evaluation only

KEYOPT(8)
Specify layer data storage:

0 --
Store data for bottom of bottom layer and top of top layer (multi-layer elements) (default)

1 --
Store data for TOP and BOTTOM, for all layers (multi-layer elements). (The volume of data may be considerable.)

2 --
Store data for TOP, BOTTOM, and MID for all layers; applies to single- and multi-layer elements

KEYOPT(9)
User thickness option:

0 --
No user subroutine to provide initial thickness (default)

1 --
Read initial thickness data from user subroutine UTHICK

See the *Guide to User-Programmable Features* for information about user-written subroutines

SHELL281 Output Data

The solution output associated with the element is in two forms:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Nodal displacements included in the overall nodal solution
- Additional element output as shown in Table 281.1: SHELL281 Element Output Definitions (p. 1352)

Several items are illustrated in Figure 281.2: SHELL281 Stress Output (p. 1352).

KEYOPT(8) controls the amount of data output to the results file for processing with the **LAYER** command. Interlaminar shear stress is available as SYZ and SXZ evaluated at the layer interfaces. KEYOPT(8) must be set to either 1 or 2 to output these stresses in the POST1 postprocessor. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to review results.

The element stress resultants (N11, M11, Q13, etc.) are parallel to the element coordinate system, as are the membrane strains and curvatures of the element. Such generalized strains are available through the SMISC option at the element centroid only. The transverse shear forces Q13, Q23 are available only in resultant form: that is, use SMISC,7 (or 8). Likewise, the transverse shear strains, γ_{13} and γ_{23}, are constant through the thickness and are only available as SMISC items (SMISC,15 and SMISC,16, respectively).

ANSYS computes moments (M11, M22, M12) with respect to the shell reference plane. By default, ANSYS adopts the shell midplane as the reference plane. To offset the reference plane to any other specified location, issue the **SECOFFSET** command. When there is a nonzero offset (L) from the reference plane to the midplane, moments with respect to the midplane ($\overline{M11}, \overline{M22}, \overline{M12}$) can be recovered from stress resultants with respect to the reference plane as follows:

$$\overline{M11} = M11 - L \times N11$$

$$\overline{M22} = M22 - L \times N22$$

$$\overline{M12} = M12 - L \times N12$$

SHELL281 does not support extensive basic element printout. POST1 provides more comprehensive output processing tools; therefore, ANSYS suggests issuing the **OUTRES** command to ensure that the required results are stored in the database.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 281.2: SHELL281 Stress Output

x_o = Element x-axis if ESYS is not provided.
x = Element x-axis if ESYS is provided.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 281.1: SHELL281 Element Output Definitions

Name	Definition	O	R
EL	Element number and name	Y	Y
NODES	Nodes - I, J, K, L	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
MAT	Material number	-	Y
THICK	Average thickness	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	-	4
PRES	Pressures P1 at nodes I, J, K, L; P2 at I, J, K, L; P3 at J,I; P4 at K,J; P5 at L,K; P6 at I,L	-	Y
TEMP	T1, T2, T3, T4 at bottom of layer 1, T5, T6, T7, T8 between layers 1-2, similarly for between next layers, ending with temperatures at top of layer NL(4*(NL+1) maximum)	-	Y
LOC	TOP, MID, BOT, or integration point location	-	1
S:X, Y, Z, XY, YZ, XZ	Stresses	3	1
S:1, 2, 3	Principal stresses	-	1
S:INT	Stress intensity	-	1
S:EQV	Equivalent stress	-	1
EPEL:X, Y, Z, XY	Elastic strains	3	1
EPEL:EQV	Equivalent elastic strains [7]	-	1
EPTH:X, Y, Z, XY	Thermal strains	3	1
EPTH:EQV	Equivalent thermal strains [7]	-	1
EPPL:X, Y, Z, XY	Average plastic strains	3	2
EPPL:EQV	Equivalent plastic strains [7]	-	2
EPCR:X, Y, Z, XY	Average creep strains	3	2
EPCR:EQV	Equivalent creep strains [7]	-	2
EPTO:X, Y, Z, XY	Total mechanical strains (EPEL + EPPL + EPCR)	3	-
EPTO:EQV	Total equivalent mechanical strains (EPEL + EPPL + EPCR)	-	-
NL:SEPL	Plastic yield stress	-	2
NL:EPEQ	Accumulated equivalent plastic strain	-	2
NL:CREQ	Accumulated equivalent creep strain	-	2
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	-	2
NL:PLWK	Plastic work/volume	-	2
NL:HPRES	Hydrostatic pressure	-	2
SEND:ELASTIC, PLASTIC, CREEP	Strain energy densities	-	2
N11, N22, N12	In-plane forces (per unit length)	-	Y
M11, M22, M12	Out-of-plane moments (per unit length)	-	8
Q13, Q23	Transverse shear forces (per unit length)	-	8
ε_{11}, ε_{22}, ε_{12}	Membrane strains	-	Y
k_{11}, k_{22}, k_{12}	Curvatures	-	8

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
γ_{13}, γ_{23}	Transverse shear strains	-	8
LOCI:X, Y, Z	Integration point locations	-	5
SVAR:1, 2, ... , N	State variables	-	6
ILSXZ	SXZ interlaminar shear stress	-	Y
ILSYZ	SYZ interlaminar shear stress	-	Y
ILSUM	Magnitude of the interlaminar shear stress vector	-	Y
ILANG	Angle of interlaminar shear stress vector (measured from the element x-axis toward the element y-axis in degrees)	-	Y
Sm: 11, 22, 12	Membrane stresses	-	Y
Sb: 11, 22, 12	Bending stresses	-	Y
Sp: 11, 22, 12	Peak stresses	-	Y
St: 13, 23	Averaged transverse shear stresses	-	Y

1. The following stress solution repeats for top, middle, and bottom surfaces.
2. Nonlinear solution output for top, middle, and bottom surfaces, if the element has a nonlinear material, or if large-deflection effects are enabled (**NLGEOM**,ON) for SEND.
3. Stresses, total strains, plastic strains, elastic strains, creep strains, and thermal strains in the element coordinate system are available for output (at all section points through thickness). If layers are in use, the results are in the layer coordinate system.
4. Available only at centroid as a ***GET** item.
5. Available only if **OUTRES**,LOCI is used.
6. Available only if the `UserMat` subroutine and **TB**,STATE command are used.
7. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.
8. Not available if the membrane element option is used (KEYOPT(1) = 1).

Table 281.2: SHELL281 Item and Sequence Numbers (p. 1355) lists output available through **ETABLE** using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this document for more information. The following notation is used in Table 281.2: SHELL281 Item and Sequence Numbers (p. 1355):

Name
output quantity as defined in the Table 281.1: SHELL281 Element Output Definitions (p. 1352)

Item
predetermined *`Item`* label for **ETABLE**

E
sequence number for single-valued or constant element data

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

I,J,K,L

sequence number for data at nodes I, J, K, L

Table 281.2: SHELL281 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
N11	SMISC	1	-	-	-	-
N22	SMISC	2	-	-	-	-
N12	SMISC	3	-	-	-	-
M11	SMISC	4	-	-	-	-
M22	SMISC	5	-	-	-	-
M12	SMISC	6	-	-	-	-
Q13	SMISC	7	-	-	-	-
Q23	SMISC	8	-	-	-	-
ε_{11}	SMISC	9	-	-	-	-
ε_{22}	SMISC	10	-	-	-	-
ε_{12}	SMISC	11	-	-	-	-
k_{11}	SMISC	12	-	-	-	-
k_{22}	SMISC	13	-	-	-	-
k_{12}	SMISC	14	-	-	-	-
γ_{13}	SMISC	15	-	-	-	-
γ_{23}	SMISC	16	-	-	-	-
THICK	SMISC	17	-	-	-	-
P1	SMISC	-	18	19	20	21
P2	SMISC	-	22	23	24	25
P3	SMISC	-	27	26	-	-
P4	SMISC	-	-	29	28	-
P5	SMISC	-	-	-	31	30
P6	SMISC	-	32	-	-	33
P1	SMISC	-	18	19	20	21
P2	SMISC	-	22	23	24	25
P3	SMISC	-	27	26	-	-
P4	SMISC	-	-	29	28	-
P5	SMISC	-	-	-	31	30
P6	SMISC	-	32	-	-	33
Sm: 11	SMISC	34	-	-	-	-
Sm: 22	SMISC	35	-	-	-	-
Sm: 12	SMISC	36	-	-	-	-
Sb: 11	SMISC	37	-	-	-	-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	E	I	J	K	L
Sb: 22	SMISC	38	-	-	-	-
Sb: 12	SMISC	39	-	-	-	-
Sp: 11 (at shell bottom)	SMISC	40	-	-	-	-
Sp: 22 (at shell bottom)	SMISC	41	-	-	-	-
Sp: 12 (at shell bottom)	SMISC	42	-	-	-	-
Sp: 11 (at shell top)	SMISC	43	-	-	-	-
Sp: 22 (at shell top)	SMISC	44	-	-	-	-
Sp: 12 (at shell top)	SMISC	45	-	-	-	-
St: 13	SMISC	46	-	-	-	-
St: 23	SMISC	47	-	-	-	-

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	Bottom of Layer i	Top of Layer NL
ILSXZ	SMISC	8 * (i - 1) + 51	8 * (NL - 1) + 52
ILSYZ	SMISC	8 * (i - 1) + 53	8 * (NL - 1) + 54
ILSUM	SMISC	8 * (i - 1) + 55	8 * (NL - 1) + 56
ILANG	SMISC	8 * (i - 1) + 57	8 * (NL - 1) + 58

SHELL281 Assumptions and Restrictions

- ANSYS, Inc. recommends against using this element in triangular form, except as a filler element. Avoid triangular form especially in areas with high stress gradients.
- Zero-area elements are not allowed. (Zero-area elements occur most often whenever the elements are numbered improperly.)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Zero thickness elements or elements tapering down to a zero thickness at any corner are not allowed (but zero thickness layers are allowed).
- If multiple load steps are used, the number of layers may not change between load steps.
- When the element is associated with preintegrated shell sections (**SECTYPE**,,GENS), additional restrictions apply. For more information, see Considerations for Using Preintegrated Shell Sections.
- No slippage is assumed between the element layers. Shear deflections are included in the element; however, normals to the center plane before deformation are assumed to remain straight after deformation.
- Transverse shear stiffness of the shell section is estimated by an energy equivalence procedure (of the generalized section forces and strains vs. the material point stresses and strains). The accuracy of this calculation may be adversely affected if the ratio of material stiffnesses (Young's moduli) between adjacent layers is very high.
- The calculation of interlaminar shear stresses is based on simplifying assumptions of unidirectional, un-coupled bending in each direction. If accurate edge interlaminar shear stresses are required, shell-to-solid submodeling should be used.
- The section definition permits use of hyperelastic material models and elastoplastic material models in laminate definition. However, the accuracy of the solution is primarily governed by fundamental assumptions of shell theory. The applicability of shell theory in such cases is best understood by using a comparable solid model.
- The layer orientation angle has no effect if the material of the layer is hyperelastic.
- The through-thickness stress, SZ, is always zero.
- This element works best with full Newton-Raphson solution scheme (**NROPT**,FULL,,ON).
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). Prestress effects can be activated via the **PSTRES** command.
- In a nonlinear analysis, the solution process terminates if the thickness at any integration point that was defined with a nonzero thickness vanishes (within a small numerical tolerance).
- If a shell section has only one layer and the number of section integration points is equal to one, or if KEYOPT(1) = 1, then the shell has no bending stiffness, a condition that can result in solver and convergence problems.

SHELL281 Product Restrictions

ANSYS Professional

- The only special features allowed are stress stiffening and large deflections.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID285

3-D 4-Node Tetrahedral Structural Solid with Nodal Pressures

MP ME ST <> <> <> <> <> PP EME MFS
Product Restrictions

SOLID285 Element Description

SOLID285 element is a lower-order 3-D, 4-node mixed u-P element. The element has a linear displacement and hydrostatic pressure behavior. The element is suitable for modeling irregular meshes (such as those generated by various CAD/CAM systems) and general materials (including incompressible materials).

The element is defined by four nodes having four degrees of freedom at each node: three translations in the nodal x, y, and z directions, and one hydrostatic pressure (HDSP) for all materials *except* nearly incompressible hyperelastic materials. For nearly incompressible materials, instead of hydrostatic pressure, the volume change rate is used at each node together with the three translation degrees of freedom. In a nonlinear analysis, you can control the tolerance of HDSP separately via the **CNVTOL** command.

The element has plasticity, hyperelasticity, creep, stress stiffening, large deflection, and large strain capabilities. It is capable of simulating deformations of nearly incompressible elastoplastic materials, nearly incompressible hyperelastic materials, and fully incompressible hyperelastic materials.

For more details about this element, see SOLID285 in the *Mechanical APDL Theory Reference*.

Figure 285.1: SOLID285 Geometry

SOLID285 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 285.1: SOLID285 Geometry (p. 1359).

In addition to the nodes, the element input data includes the orthotropic or anisotropic material properties. Orthotropic and anisotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Linear Material Properties in the *Material Reference*.

Element loads are described in Nodal Loading (p. 49). Pressure loads may be input as surface loads on the element faces as shown by the circled numbers on Figure 285.1: SOLID285 Geometry (p. 1359). Positive

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

pressures act into the element. Temperatures may be input as element body loads at the nodes. The node I temperature T(I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). For any other input temperature pattern, unspecified temperatures default to TUNIF.

As described in Coordinate Systems (p. 55), you can use **ESYS** to orient the material properties and strain/stress output. Use **RSYS** to choose output that follows the material coordinate system or the global coordinate system. For the case of hyperelastic materials, the output of stress and strain is always with respect to the global Cartesian coordinate system rather than following the material/element coordinate system.

You can apply an initial stress state to this element via the **INISTATE** command. For more information, see Initial State in the *Basic Analysis Guide*.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, use **NROPT**,UNSYM.

The next table summarizes the element input. Element Input (p. 43) gives a general description of element input.

SOLID285 Input Summary

Nodes
I, J, K, L

Degrees of Freedom
UX, UY, UZ, HDSP

Real Constants
None

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, EY, EZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ), PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), DENS, GXY, GYZ, GXZ, ALPD, BETD

Surface Loads

Pressures --
face 1 (J-I-K), face 2 (I-J-L), face 3 (J-K-L), face 4 (K-I-L)

Body Loads

Temperatures --
T(I), T(J), T(K), T(L)

Body force densities --
The element values in the global X, Y, and Z directions.

Special Features

Birth and death
Element technology autoselect
Initial state
Large deflection

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Large strain
Linear perturbation
Nonlinear stabilization
Rezoning

Solid 285 Element Technology

This element has only a mixed u-P formulation with pressure stabilization. For more information, see Element Technologies.

SOLID285 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements and hydrostatic pressure included in the overall nodal solution
- Additional element output as shown in Table 285.1: SOLID285 Element Output Definitions (p. 1361)

Several items are illustrated in Figure 285.2: SOLID285 Stress Output (p. 1361). The element stress directions are parallel to the element coordinate system. A general description of solution output is given in The Item and Sequence Number Table (p. 52). See the *Basic Analysis Guide* for ways to view results.

Figure 285.2: SOLID285 Stress Output

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 285.1: SOLID285 Element Output Definitions

Name	Definition	O	R
EL	Element Number	-	Y
NODES	Nodes - I, J, K, L	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
MAT	Material number	-	Y
VOLU:	Volume	-	Y
XC, YC, ZC	Location where results are reported	Y	3
PRES	Pressures P1 at nodes J, I, K; P2 at I, J, L; P3 at J, K, L; P4 at K, I, L	-	Y
TEMP	Temperatures T(I), T(J), T(K), T(L)	-	Y
S:X, Y, Z, XY, YZ, XZ	Stresses	Y	Y
S:1, 2, 3	Principal stresses	-	Y
S:INT	Stress intensity	-	Y
S:EQV	Equivalent stress	-	Y
EPEL:X, Y, Z, XY, YZ, XZ	Elastic strains	Y	Y
EPEL:EQV	Equivalent elastic strains [6]	-	Y
EPTH:X, Y, Z, XY, YZ, XZ	Thermal strains	1	1
EPTH: EQV	Equivalent thermal strains [6]	1	1
EPPL:X, Y, Z, XY, YZ, XZ	Plastic strains [7]	1	1
EPPL:EQV	Equivalent plastic strains [6]	1	1
EPCR:X, Y, Z, XY, YZ, XZ	Creep strains	1	1
EPCR:EQV	Equivalent creep strains [6]	1	1
EPTO:X, Y, Z, XY, YZ, XZ	Total mechanical strains (EPEL + EPPL + EPCR)	Y	-
EPTO:EQV	Total equivalent mechanical strains (EPEL + EPPL + EPCR)	Y	-
NL:EPEQ	Accumulated equivalent plastic strain	1	1
NL:CREQ	Accumulated equivalent creep strain	1	1
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	1	1
NL:HPRES	Hydrostatic pressure	1	1
SEND: ELASTIC, PLASTIC, CREEP	Strain energy density	-	1
LOCI:X, Y, Z	Integration point locations	-	4
SVAR:1, 2, ... , N	State variables	-	5

1. Nonlinear solution, output only if the element has a nonlinear material, or if large-deflection effects are enabled (**NLGEOM**,ON) for SEND.

2. Output only if element has a thermal load.

3. Available only at centroid as a ***GET** item.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Available only if **OUTRES**,LOCI is used.

5. Available only if the `UserMat` subroutine and **TB**,STATE command are used.

6. The equivalent strains use an effective Poisson's ratio: for elastic and thermal this value is set by the user (**MP**,PRXY); for plastic and creep this value is set at 0.5.

7. For the shape memory alloy material model, transformation strains are reported as plasticity strain EPPL.

Table 285.2: SOLID285 Item and Sequence Numbers (p. 1363) lists output available through **ETABLE** using the Sequence Number method. See The General Postprocessor (POST1) in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this document for more information. The following notation is used in Table 285.2: SOLID285 Item and Sequence Numbers (p. 1363):

Name
output quantity as defined in Table 285.1: SOLID285 Element Output Definitions (p. 1361)

Item
predetermined Item label for **ETABLE** command

I,J,...,R
sequence number for data at nodes I, J, ..., R

Table 285.2: SOLID285 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input					
	Item	I	J	K	L	M,...,R
P1	SMISC	2	1	3	-	-
P2	SMISC	4	5	-	6	-
P3	SMISC	-	7	8	9	-
P4	SMISC	11	-	10	12	-

SOLID285 Assumptions and Restrictions

- The element must not have a zero volume.
- Elements may be numbered either as shown in Figure 285.1: SOLID285 Geometry (p. 1359) or may have node L below the I, J, K plane.
- Only the sparse solver is valid when using this element.
- Support is available for static and transient analyses.
- The element may not offer sufficient accuracy for bending-dominant problems, especially if the mesh is not fine enough.
- On the interfaces of different materials, the elements should not share nodes because the hydrostatic pressure value is not continuous at those nodes. This behavior can be overcome in either of two ways:
  - Coupling the displacements of the nodes on the interface but leaving HDSP unconstrained.
  - Adding bonded contact elements on the interfaces.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The element is not computationally efficient when the model uses compressible material. In such cases, ANSYS, Inc. recommends using a more suitable (pure displacement) element such as SOLID185 or SOLID187.
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). Prestress effects can be activated by the **PSTRES** command.

SOLID285 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PIPE288

3-D 2-Node Pipe

MP ME ST PR PRN <> <> <> PP EME MFS
Product Restrictions

PIPE288 Element Description

The PIPE288 element is suitable for analyzing slender to moderately stubby/thick pipe structures. The element is based on Timoshenko beam theory. Shear-deformation effects are included.

PIPE288 is a linear, quadratic, or cubic two-node pipe element in 3-D. The element has six degrees of freedom at each node (the translations in the x, y, and z directions and rotations about the x, y, and z directions). The element is well-suited for linear, large rotation, and/or large strain nonlinear applications.

PIPE288 includes stress stiffness terms, by default, in any analysis with **NLGEOM**,ON. The provided stress-stiffness terms enable the elements to analyze flexural, lateral, and torsional stability problems (using eigenvalue buckling, or collapse studies with arc length methods or nonlinear stabilization).

Elasticity, hyperelasticity, plasticity, creep, and other nonlinear material models are supported. Internal fluid and external insulation are supported. Added mass, hydrodynamic added mass and loading, and buoyant loading are available.

Figure 288.1: PIPE288 Geometry

PIPE288 Element Technology and Usage Recommendations

PIPE288 is based on Timoshenko beam theory, a first-order shear-deformation theory. Transverse-shear strain is constant through the cross-section; that is, cross-sections remain plane and undistorted after deformation.

The element can be used for slender or stout pipes. Due to the limitations of first-order shear-deformation theory, only moderately "thick" pipes can be analyzed. The slenderness ratio of a pipe structure (GAL^2 / (EI)) can be used to judge the applicability of the element, where:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

G
Shear modulus

A
Area of the cross-section

L
Length of the member (not the element length)

EI
Flexural rigidity

For pipes, (GAL^2 / EI) can be reduced to: $2L^2 / ((1 + \nu)(R_o^2 + R_i^2))$, or for thin-walled pipes: $L^2 / ((1 + \nu) R^2)$, where ν = Poisson's ratio, R_o = outer radius, R_i = inner radius, and R = average radius.

The following illustration provides an estimate of transverse-shear deformation in a cantilever pipe subjected to a tip load. Although the results cannot be extrapolated to other applications, the example serves generally. ANSYS, Inc. recommends a slenderness ratio greater than 30.

Figure 288.2: Transverse-Shear-Deformation Estimation

Slenderness Ratio (GAL^2/(EI))	δ Timoshenko / δ Euler-Bernoulli
25	1.120
50	1.060
100	1.030
1000	1.003

The PIPE288 element supports an elastic relationship between transverse-shear forces and transverse-shear strains.

When KEYOPT(3) = 0 (linear, default), PIPE288 is based on linear shape functions. It uses one point of integration along the length; therefore, all element solution quantities are constant along the length. For example, when SMISC quantities are requested at nodes I and J, the centroidal values are reported for both end nodes. This option is recommended if the element is used as stiffener and it is necessary to maintain compatibility with a first-order shell element (such as SHELL181). Only constant bending moments can be represented exactly with this option. Mesh refinement is generally required in typical applications.

When KEYOPT(3) = 2 (quadratic), PIPE288 has an internal node in the interpolation scheme, effectively making this a beam element based on quadratic shape functions. Two points of integration are used, resulting in linear variation of element solution quantities along the length. Linearly varying bending moments are represented exactly.

When KEYOPT(3) = 3 (cubic), PIPE288 has two internal nodes and adopts cubic shape functions. Quadratically varying bending moments are represented exactly. Three points of integration along the length

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

are used, resulting in quadratic variation of element solution quantities along the length. Unlike typical cubic (Hermitian) formulations, cubic interpolation is used for all displacements and rotations.

In general, the more complex the element, the fewer elements are needed. Quadratic and cubic options are recommended when higher-order element interpolations are desired in situations where:

- Nonuniform loads (including tapered distributed loads) exist within the element; in this case, the cubic option gives superior results over the quadratic option.

  (For partially distributed loads and non-nodal point loads, only the cubic option is valid.)

- The element may undergo highly nonuniform deformation (for example, when individual frame members in civil engineering structures are modeled with single elements).

PIPE288 supports both the thin-pipe (KEYOPT(4) = 1) and the thick-pipe (KEYOPT(4) = 2) options. The thin-pipe option assumes a plane stress state in the pipe wall and ignore the stress in the wall thickness direction. The thick-pipe option accounts for the full 3-D stress state and generally leads to more accurate results in thick-walled pipes where through-the-thickness stress can be significant. The element allows change in cross-sectional area in large-deflection analysis. While the thick-pipe option can accurately determine the cross-section area change from the actual material constitutive properties, the thin-pipe option calculates the approximate area change based on a simple material incompressibility assumption. The thin pipe option runs slightly faster than the thick pipe option. The thick pipe option should generally be avoided for very thin-walled pipes (for example, Do/Tw > 1000.0), and the thin pipe option should generally be avoided for very thick-walled pipes (for example, Do/Tw < 10.0). The thick pipe option is required for a “solid” pipe section (Do/Tw = 2.0). For more information, see the **SECDATA** command.

Two limitations are associated with the quadratic and cubic options in PIPE288:

- Although the elements employ higher-order interpolations, the initial geometry of PIPE288 is treated as straight.

- Because the internal nodes are inaccessible, no boundary/loading/initial conditions are allowed on these internal nodes.

As a result of the limitations associated with the quadratic and cubic options, you will notice discrepancies in the results between PIPE289 and the quadratic option of PIPE288 if the midside nodes of the PIPE289 model have specified boundary/loading/initial conditions and/or the midside nodes are not located exactly at the element midpoint. Similarly, the cubic option of PIPE288 may not be identical to a traditional cubic (Hermitian) beam element.

For the mass matrix and load vectors, a higher order integration rule than that used for stiffness matrix is employed. The elements support both consistent and lumped mass matrices. The **LUMPM**,ON command activates lumped mass matrix. Consistent mass matrix is the default behavior. You can add mass per unit length via the **SECCONTROL** command's *ADDMAS* values. See "PIPE288 Input Summary" (p. 1371).

When ocean loading is applied, the loading is nonlinear (that is, based on the square of the relative velocity between the structure and the water). Accordingly, the full Newton-Raphson option (**NROPT**,FULL) may be necessary to achieve optimal results. (Full Newton-Raphson is applied automatically in an analysis involving large-deflection effects [**NLGEOM**,ON].)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PIPE288 Input Data

The geometry, node locations, coordinate system, and pressure directions for this element are shown in Figure 288.1: PIPE288 Geometry (p. 1365). PIPE288 is defined by nodes I and J in the global coordinate system. If ocean loading is present, the global origin is normally at the mean sea level, with the global Z-axis pointing away from the center of the earth; however, the vertical location can be adjusted via *Zmsl* (*Val6*) on the **OCDATA** command (following the **OCTYPE**,BASIC command).

Because the section is round, the element orientation is important only for defining offsets and temperatures, and interpreting bending moment directions and stress locations.

Node K is the preferred way to define the orientation of the element. For information about orientation nodes and beam meshing, see Generating a Beam Mesh With Orientation Nodes in the *Modeling and Meshing Guide*. See the **LMESH** and **LATT** command descriptions for details on generating the K node automatically.

PIPE288 can also be defined without the orientation node. The element x-axis is oriented from node I toward node J. When no orientation node is used, the default orientation of the element y-axis is automatically calculated to be parallel to the global X-Y plane. If the element is parallel to the global Z-axis (or within a 0.01 percent slope of it), the element y-axis is oriented parallel to the global Y-axis. To control the element orientation about the element x-axis, use the orientation-node option. If both are defined, the orientation-node option takes precedence. The orientation node K, if used, defines a plane (with I and J) containing the element x and z-axes (as shown). If this element is used in a large-deflection analysis, the location of the orientation node K only *initially* orients the element.

The pipe element is a one-dimensional line elements in space. The cross-section details are provided separately via the **SECTYPE** and **SECDATA** commands. A section is associated with the pipe elements by specifying the section ID number (**SECNUM**). A section number is an independent element attribute.

PIPE288 Cross-Sections

PIPE288 can be associated only with the pipe cross-section (**SECTYPE**,,PIPE). The material of the pipe is defined as an element attribute (MAT).

PIPE288 is provided with section-relevant quantities (area of integration, position, etc.) automatically at a number of section points via the **SECDATA** command. Each section is assumed to be an assembly of a predetermined number of cells and is numerically integrated.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 288.3: Typical Cross-Section Cell

Section Flexibility

To apply section flexibility factors, use the **SFLEX** command. The command is valid only for linear material properties and small strain analyses, and does not support offsets, temperature loading, or initial state loading.

PIPE288 Loads

Element loads are described in Element Loading (p. 46). Internal fluid and external insulation are supported. Added mass, hydrodynamic added mass, and hydrodynamic and buoyant loading, are available via the **OCDATA** and **OCTABLE** commands. For ocean loading, you can define added mass directly via the **OCDATA** and **OCTABLE** commands (rather than by issuing the **SECCONTROL** command as is typical with other load types.)

Forces are applied at the nodes I and J. If the centroidal axis is not colinear with the element x-axis because of offsets, applied axial forces will cause bending. The nodes should therefore be located at the desired points where you want to apply the forces. Use the *OFFSETY* and *OFFSETZ* arguments of the **SECOFFSET** command appropriately. By default, the program uses the centroid as the reference axis for the pipe elements.

Pressure Input

To input surface-load information on the element faces, issue the **SFE** command

On faces 1 and 2, internal and external pressures are input, respectively, on an average basis over the element.

On face 3, the input (*VAL1*) is the global Z coordinate of the free surface of the internal fluid of the pipe. Specify this value with **ACEL**,0,0,*ACEL_Z*, where *ACEL_Z* is a positive number. The free surface coordinate is used for the internal mass and pressure effects. If *VAL1* on the **SFE** command is zero, no fluid inside of the pipe is considered. If the internal fluid free surface should be at Z = 0, use a very small number instead. The pressure calculation presumes that the fluid density above an element to the free surface is constant. For cases where the internal fluid consists of more than one type of fluid (such as oil over water), the free surface coordinate of the elements in the fluids below the top fluid need to be adjusted (lowered) to account for the less dense fluid(s) above them.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The internal fluid mass is assumed to be lumped along the centerline of the pipe, so as the centerline of a horizontal pipe moves across the free surface, the mass changes in step-function fashion. The free surface location is stepped, even if you specify ramped loading (**KBC**,0).

On face 4 and face 5, lateral pressures are input as force-per-unit-length.

On face 6 and face 7, end "pressures" are input as forces.

Faces 4 through 8 are shown by the circled numbers in Figure 288.1: PIPE288 Geometry (p. 1365).

Temperature Input

When KEYOPT(1) = 0, temperatures can be input as element body loads at the inner and outer surfaces at both ends of the pipe element so that the temperature varies linearly through the wall thickness. If only two temperatures are specified, those two temperatures are used at both ends of the pipe element (that is, there is no gradient along the length). If only the first temperature is specified, all others default to the first. The following graphic illustrates temperature input at a node when KEYOPT(1) = 0:

When KEYOPT(1) = 1, temperatures can be input as element body loads at three locations at both nodes of the pipe element so that the temperature varies linearly in the element y and z directions. At either end of the element, temperatures can be input at these locations:

- At the element x-axis (T(0,0))
- At the outer radius from the x-axis in the element y-direction (T(R_o,0))
- At the outer radius from the x-axis in the element z-direction (T(0,R_o))

The following graphic illustrates temperature input at a node when KEYOPT(1) = 1:

Element locations (T(Y,Z)) are given according to the convention used in Figure 288.1: PIPE288 Geometry (p. 1365).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For pipe elements, element body load commands (**BFE**) accept an element number and a list of values, 1 through 6 for temperatures $T_I(0,0)$, $T_I(1,0)$, $T_I(0,1)$, $T_J(0,0)$, $T_J(1,0)$, and $T_J(0,1)$. This input can be used to specify temperature gradients that vary linearly both over the cross section and along the length of the element.

The following defaults apply to element temperature input:

- If all temperatures after the first are unspecified, they default to the first. This pattern applies a uniform temperature over the entire element. (The first coordinate temperature, if unspecified, defaults to TUNIF.)
- If all three temperatures at node I are input, and all temperatures at node J are unspecified, the node J temperatures default to the corresponding node I temperatures. This pattern applies a temperature gradient that varies linearly over the cross section but remains constant along the length of the element.
- For any other input pattern, unspecified temperatures default to TUNIF.

Alternatively, temperatures at nodes I and J can be defined using nodal body loads (**BF**,*NODE*,TEMP,*VAL1*). This specifies a uniform temperature over the cross section at the specified node.

Other Input

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, issue an **NROPT**,UNSYM command.

PIPE288 Input Summary

Nodes

I, J, K (an optional orientation node)

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ

Section Information

Accessed via **SECTYPE**,,PIPE and **SECDATA** commands.

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ), DENS, GXY, GYZ, GXZ, ALPD, BETD

Surface Loads

Pressure --

face 1 - Internal pressure
face 2 - External pressure
face 3 - Z coordinate of free surface of fluid on inside of pipe
face 4 (I-J) (-z normal direction)
face 5 (I-J) (-y normal direction)
face 6 (I-J) (+x tangential direction)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

face 7 (I) (+x axial direction)
face 8 (J) (-x axial direction)

I and J denote end nodes.
Use a negative value for loading in the opposite direction.
Input pressure values for faces 1, 2, and 3 via the **SFE** command. Input pressure values for faces 4 through 8 via the **SFBEAM** command.
For faces 4, 5, and 6, offsets apply only if you are using the cubic option (KEYOPT(3) = 3).

Body Loads

Temperatures --
TOUT(I), TIN(I), TOUT(J), TIN(J) if KEYOPT(1) = 0.

TAVG(I), T_y(I), T_z(I), TAVG(J), T_y(J), T_z(J) if KEYOPT(1) = 1.

Special Features

Birth and death
Element technology autoselect
Initial state
Large deflection
Linear perturbation
Nonlinear stabilization
Ocean loading
Stress stiffening

KEYOPT(1)

Temperature input

0 --
Through wall gradient

1 --
Diametral gradient

KEYOPT(3)

Shape functions along the length:

0 --
Linear

2 --
Quadratic

3 --
Cubic

KEYOPT(4)

Hoop strain treatment

1 --
Thin pipe theory

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2 --
Thick pipe theory

KEYOPT(6)
End cap loads

0 --
Internal and external pressures cause loads on end caps

1 --
Internal and external pressures do not cause loads on end caps

KEYOPT(7), KEYOPT(9), KEYOPT(11), and KEYOPT(12)
Active only when **OUTPR**,ESOL is active:

KEYOPT(7)
Output control for section forces/moments and strains/curvatures:

0 --
Output section forces/moments, strains/curvatures, internal and external pressures, effective tension, and maximum hoop stress (default)

1 --
Same as KEYOPT(7) = 0 plus current section area

2 --
Same as KEYOPT(7) = 1 plus element basis directions (X,Y,Z)

3 --
Output section forces/moments, strains/curvatures, internal and external pressures, effective tension, and maximum hoop stress extrapolated to the element nodes

KEYOPT(8)
Shear stress output:

0 --
Output a combined state of the following two types (default)

1 --
Output only torsion-related shear stresses

2 --
Output only flexure-related transverse-shear stresses

KEYOPT(9)
Output control at integration points:

0 --
None (default)

1 --
Maximum and minimum stresses/strains

2 --
Same as KEYOPT(9) = 1 plus stresses and strains at each section node

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(11)

Output control for values extrapolated to the element and section nodes:

0 --

None (default)

1 --

Maximum and minimum stresses/strains

2 --

Same as KEYOPT(11) = 1 plus stresses and strains along the exterior boundary of the cross-section

3 --

Same as KEYOPT(11) = 1 plus stresses and strains at all section nodes

KEYOPT(12)

Hydrodynamic output (not available in harmonic analyses that include ocean wave effects (**HROCEAN**)):

0 --

None (default)

1 --

Additional centroidal hydrodynamic printout

KEYOPT(15)

Results file format:

0 --

Store averaged results at each section corner node (default).

1 --

Store non-averaged results at each section integration point. (The volume of data may be excessive.)

PIPE288 Output Data

The solution output associated with these elements is in two forms:

- Nodal displacements and reactions included in the overall nodal solution
- Additional element output as described in Table 288.1: PIPE288 Element Output Definitions (p. 1375)

For ways to view results, see the *Basic Analysis Guide*.

To view 3-D deformed shapes for PIPE288, issue an **OUTRES**,MISC or **OUTRES**,ALL command for static or transient analyses. To view 3-D mode shapes for a modal or eigenvalue buckling analysis, expand the modes with element results calculation active (via the **MXPAND** command's `Elcalc` = YES option).

Linearized Stress

It is customary in pipe design to employ components of axial stress that contribute to axial loads and bending in each direction separately; therefore, PIPE288 provides a linearized stress output as part of its SMISC output record, as indicated in the following definitions:

SDIR is the stress component due to axial load.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SDIR = Fx/A, where Fx is the axial load (SMISC quantities 1 and 14) and A is the area of the cross-section.

SByT and SByB are bending stress components.

$$SByT = -Mz * R_0 / I$$
$$SByB = Mz * R_0 / I$$
$$SBzT = My * R_0 / I$$
$$SBzB = -My * R_0 / I$$

where My, Mz are bending moments in the beam coordinate system (SMISC quantities 2,15,3,16) as shown in Figure 288.1: PIPE288 Geometry (p. 1365), R_0 is the outside radius, and I is the moment of inertia of the cross-section. The program uses the maximum and minimum cross-section dimensions.

Corresponding definitions for the component strains are:

$$EPELDIR = Ex$$
$$EPELByT = -Kz * R_0$$
$$EPELByB = Kz * R_0$$
$$EPELBzT = Ky * R_0$$
$$EPELBzB = -Ky * R_0$$

where Ex, Ky, and Kz are generalized strains and curvatures (SMISC quantities 7,8,9, 20,21 and 22).

The reported stresses are strictly valid only for elastic behavior of members. PIPE288 always employs combined stresses in order to support nonlinear material behavior. When the elements are associated with nonlinear materials, the component stresses can at best be regarded as linearized approximations and should be interpreted with caution.

When using KEYOPT(9) with the cubic option (KEYOPT(3) = 3), the integration point at the middle of the element is reported *last* in the integration-point printout.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

For the stress and strain components, X refers to axial, Y refers to hoop, and Z refers to radial.

Table 288.1: PIPE288 Element Output Definitions

Name	Definition	O	R
EL	Element number	Y	Y
NODES	Element connectivity	Y	Y
MAT	Material number	Y	Y
C.G.:X, Y, Z	Element center of gravity	Y	1
Area	Area of cross-section	2	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
S:x, y, z, xy, yz, xz	Section point stresses	3	Y
EPEL:x, y, z, xy, yz, xz	Elastic strains	3	Y
EPTO:x,y,z xy, yz, xz	Section point total mechanical strains (EPEL + EPPL + EPCR)	3	Y
EPTT:x,y,z xy, yz, xz	Section point total strains (EPEL + EPPL + EPCR + EPTH)	3	Y
EPPL:x, y, z, xy, yz, xz	Section point plastic strains	3	Y
EPCR:x, y, z, xy, yz, xz	Section point creep strains	3	Y
EPTH:x, y, z, xy, yz, xz	Section point thermal strains	3	Y
NL:EPEQ	Accumulated equivalent plastic strain	-	4
NL:CREQ	Accumulated equivalent creep strain	-	4
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	-	4
NL:PLWK	Plastic work/volume	-	4
SEND:ELASTIC, PLASTIC, CREEP	Strain energy densities	-	4
TQ	Torsional moment	Y	Y
TE	Torsional strain	Y	Y
SFy, SFz	Section shear forces	2	Y
SEy, SEz	Section shear strains	2	Y
Ky, Kz	Curvature	Y	Y
Ex	Axial strain	Y	Y
Fx	Axial force (excluding insulation)	Y	Y
My, Mz	Bending moments	Y	Y
INT PRESS	Internal pressure at integration point	5	5
EXT PRESS	External pressure at integration point	5	5
EFFECTIVE TENS	Effective tension on pipe	5	5
MAX HOOP STRESS	Maximum hoop stress at integration point	5	5
SDIR	Axial direct stress	-	2
SByT	Bending stress on the element +Y side of the pipe	-	2
SByB	Bending stress on the element -Y side of the pipe	-	2
SBzT	Bending stress on the element +Z side of the pipe	-	2

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
SBzB	Bending stress on the element -Z side of the pipe	-	2
EPELDIR	Axial strain at the end	-	Y
EPELByT	Bending strain on the element +Y side of the pipe	-	Y
EPELByB	Bending strain on the element -Y side of the pipe	-	Y
EPELBzT	Bending strain on the element +Z side of the pipe	-	Y
EPELBzB	Bending strain on the element -Z side of the pipe	-	Y
TEMP	Temperatures at all section corner nodes	-	Y
LOCI:X, Y, Z	Integration point locations	-	6
SVAR:1, 2, ... , N	State variables	-	7
The following values apply to ocean loading only:			
GLOBAL COORD	Element centroid location	8	Y
VR, VZ	Radial and vertical fluid particle velocities (VR is always > 0)	8	Y
AR, AZ	Radial and vertical fluid particle accelerations	8	Y
PHDYN	Dynamic fluid pressure head	8	Y
ETA	Wave amplitude over integration point	8	Y
TFLUID	Fluid temperature (printed if VISC is nonzero)	8	Y
VISC	Viscosity (output if VISC is nonzero)	8	Y
REN, RET	Normal and tangential Reynolds numbers (if VISC is nonzero)	8	Y
CT	Input tangential drag coefficients evaluated at Reynolds numbers	8	Y
CDY, CDZ	Input normal drag coefficients evaluated at Reynolds numbers	8	Y
CMY, CMZ	Input inertia coefficients evaluated at Reynolds numbers	8	Y
URT, URN	Tangential (parallel to element axis) and normal relative velocities	8	Y
ABURN	Vector sum of normal (URN) velocities	8	Y
AN	Accelerations normal to element	8	Y
FX, FY, FZ	Hydrodynamic tangential and normal forces in element coordinates	8	Y
ARGU	Effective position of wave (radians)	8	Y

1. Available only at the centroid as a ***GET** item, or on the NMISC record for ocean loading.
2. See KEYOPT(7) description.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. See KEYOPT(9) and KEYOPT(11) descriptions.
4. Available if the element has a nonlinear material, or if large-deflection effects are enabled (**NLGEOM**,ON) for SEND.
5. Available only if ocean loading is present.
6. Available only if **OUTRES**,LOCI is used.
7. Available only if the `UserMat` subroutine and the **TB**,STATE command are used.
8. Applies to ocean loading only.

More output is described via the **PRESOL** and ***GET**,,SECR commands in POST1.

Table 288.2: PIPE288 Item and Sequence Numbers (p. 1378) lists output available for the **ETABLE** command using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) for more information. The output tables use the following notation:

Name
output quantity as defined in Table 288.1: PIPE288 Element Output Definitions (p. 1375)

Item
predetermined Item label for **ETABLE**

E,I,J
sequence number for data at nodes E, I and J

Table 288.2: PIPE288 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
Fx	SMISC	--	1	14
My	SMISC	--	2	15
Mz	SMISC	--	3	16
TQ	SMISC	--	4	17
SFz	SMISC	--	5	18
SFy	SMISC	--	6	19
Ex	SMISC	--	7	20
Ky	SMISC	--	8	21
Kz	SMISC	--	9	22
TE	SMISC	--	10	23
SEz	SMISC	--	11	24
SEy	SMISC	--	12	25
Area	SMISC	--	13	26
BM	SMISC	--	27	29
BK	SMISC	--	28	30
SDIR	SMISC	--	31	36

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
SByT	SMISC	--	32	37
SByB	SMISC	--	33	38
SBzT	SMISC	--	34	39
SBzB	SMISC	--	35	40
EPELDIR	SMISC	--	41	46
EPELByT	SMISC	--	42	47
EPELByB	SMISC	--	43	48
EPELBzT	SMISC	--	44	49
EPELBzB	SMISC	--	45	50
TEMP	SMISC	--	51-53	54-56
INT PRESS [1]	SMISC	--	61	65
EXT PRESS [1]	SMISC	--	62	66
EFFECTIVE TENS [1]	SMISC	--	63	67
MAX HOOP STRESS [1]	SMISC	--	64	68
S:x, y, z, xy, yz, xz	LS	--	CI[2], DI[3]	CJ[2], DJ[3]
EPEL:x, y, z, xy, yz, xz	LEPEL	--	CI[2], DI[3]	CJ[2], DJ[3]
EPTH:x, y, z, xy, yz, xz	LEPTH	--	CI[2], DI[3]	CJ[2], DJ[3]
EPPL:x, y, z, xy, yz, xz	LEPPL	--	CI[2], DI[3]	CJ[2], DJ[3]
EPCR:x, y, z, xy, yz, xz	LEPCR	--	CI[2], DI[3]	CJ[2], DJ[3]
EPTO:x, y, z, xy, yz, xz	LEPTO	--	CI[2], DI[3]	CJ[2], DJ[3]
EPTT:x, y, z, xy, yz, xz	LEPTT	--	CI[2], DI[3]	CJ[2], DJ[3]
The following output quantities are valid for ocean loading only:				
GLOBAL COORD	NMISC	1, 2, 3	--	--
VR, VZ	NMISC	4, 5	--	--
AR, AZ	NMISC	6, 7 [4]	--	--
PHDYN	NMISC	8 [4]	--	--
ETA	NMISC	9 [4]	--	--
TFLUID	NMISC	10	--	--
VISC	NMISC	11	--	--
REN, RET	NMISC	12, 13 [5]	--	--
CT	NMISC	14	--	--
CDY, CDZ	NMISC	15, 16	--	--
CMY, CMZ	NMISC	17, 18 [4]	--	--
URT, URN	NMISC	19, 20, 21	--	--

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
ABURN	NMISC	22 [4]	--	--
AN	NMISC	23, 24 [4]	--	--
FX, FY, FZ	NMISC	25, 26, 27	--	--
ARGU	NMISC	28 [4]	--	--

1. Internal pressure (INT PRESS), external pressure (EXT PRESS), effective tension (EFFECTIVE TENS), and maximum hoop stress (MAX HOOP STRESS) occur at integration points, and not at end nodes.

2. CI and CJ are the sequence numbers for accessing the averaged line element solution quantities (LS, LEPEL, LEPTH, LEPPL, LEPCR, LEPTO, and LEPTT) at RST section nodes (section corner nodes where results are available), at element Node I and J respectively. CI and CJ are applicable only when KEYOPT(15) = 0. For a given section corner node *nn*, CI and CJ are given as follows:

   CI = (*nn* - 1) * 6 + COMP

   CJ = (*nnMax* + *nn* - 1) * 6 + COMP

   Where *nnMax* is the total number of RST section nodes, and COMP is the stress or strain component (1 - x, 2 - y, 3 - z, 4 - xy, 5 - yz, 6 - xz). Locations of RST section nodes can be visualized with **SECPLOT**,,6.

3. DI and DJ are the sequence numbers for accessing the non-averaged line element solution quantities (LS, LEPEL, LEPTH, LEPPL, LEPCR, LEPTO, and LEPTT) at RST section integration points (section integration points where results are available), at element Node I and J respectively. DI and DJ are applicable only when KEYOPT(15) = 1. For the ith integration point (i = 1, 2, 3, or 4) in section cell *nc*, DI and DJ are given as follows:

   DI = (*nc* - 1) * 24 + (i - 1) * 6 + COMP

   DJ = (*ncMax* + *nc* - 1) * 24 + (i - 1) * 6 + COMP

   Where *ncMax* is the total number of RST section cells, and COMP is the stress or strain component (1 - x, 2 - x, 3 - z, 4 - xy, 5 - yz, 6 - xz). Locations of RST section cells can be visualized with **SECPLOT**,,7.

4. These quantities are output only if wave loading is used.

5. These quantities are output only if a Reynold's number dependency is used.

For more usage details, see Plot and Review the Section Results and Example Problem with Cantilever Beams.

Transverse-Shear Stress Output

The shear stresses are caused by torsional and transverse loads. PIPE288 is based on first-order shear-deformation theory, also popularly known as Timoshenko beam theory. The transverse-shear strain is constant for the cross-section; therefore, the shear energy is based on a transverse-shear force. This shear force is redistributed by predetermined shear stress distribution coefficients across the pipe cross-

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

section, and made available for output purposes. Use KEYOPT(8) to activate output of shear stresses caused by flexure or transverse loading.

By default, the program uses a mesh density (for cross-section modeling) that provides accurate results for torsional rigidity, warping rigidity, inertia properties, and shear center determination. The default mesh employed is also appropriate for nonlinear material calculations; however, more refined cross-section models may be necessary if the shear stress distribution due to transverse loads must be captured very accurately. Use the **SECDATA** command to adjust cross-section mesh density.

The traction-free state at the edges of the cross-section is met only in a well-refined model of the cross-section.

The transverse-shear distribution calculation ignores the effects of Poisson's ratio. The Poisson's ratio affects the shear correction factor and shear stress distribution slightly, and this effect is ignored.

PIPE288 Assumptions and Restrictions

- The pipe cannot have zero length.
- Cross-section distortion or collapse is not considered.
- Rotational degrees of freedom are not included in the lumped mass matrix if offsets are present.
- The element works best with the full Newton-Raphson solution scheme (the default option in solution control).
- Only moderately "thick" pipes can be analyzed. See "PIPE288 Element Technology and Usage Recommendations" (p. 1365) for more information.
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). Prestress effects can be activated via the **PSTRES** command.
- Hydrodynamic output via KEYOPT(12) is not available in harmonic analyses that include ocean wave effects (**HROCEAN**).
- For a random vibration (PSD) analysis, equivalent stress is not calculated.
- Temperature-dependent density of the internal fluid is evaluated at an average temperature of the element. Temperature-dependent density of the insulation is evaluated at an average temperature at each integration point along the length.

PIPE288 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The only special features allowed are stress stiffening and large deflections.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

PIPE289

3-D 3-Node Pipe

MP ME ST PR PRN <> <> <> PP EME MFS
Product Restrictions

PIPE289 Element Description

The PIPE289 element is suitable for analyzing slender to moderately stubby/thick pipe structures. The element is based on Timoshenko beam theory. Shear-deformation effects are included.

PIPE289 is a quadratic three-node pipe element in 3-D. The element has six degrees of freedom at each node (the translations in the x, y, and z directions and rotations about the x, y, and z directions). The element is well-suited for linear, large rotation, and/or large strain nonlinear applications.

PIPE289 includes stress stiffness terms, by default, in any analysis with **NLGEOM**,ON. The provided stress-stiffness terms enable the elements to analyze flexural, lateral, and torsional stability problems (using eigenvalue buckling, or collapse studies with arc length methods or nonlinear stabilization).

Elasticity, hyperelasticity, plasticity, creep, and other nonlinear material models are supported. Internal fluid and external insulation are supported. Added mass, hydrodynamic added mass and loading, and buoyant loading are available.

For more detailed information about this element, see PIPE289 - 3-D 3-Node Pipe in the *Mechanical APDL Theory Reference*.

Figure 289.1: PIPE289 Geometry

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PIPE289 Element Technology and Usage Recommendations

PIPE289 is based on Timoshenko beam theory, a first-order shear-deformation theory. Transverse-shear strain is constant through the cross-section; that is, cross-sections remain plane and undistorted after deformation. (For cases where cross-section distortion must be considered, it is preferable to use EL-BOW290.)

The element can be used for slender or stout pipes. Due to the limitations of first-order shear-deformation theory, only moderately "thick" pipes can be analyzed. The slenderness ratio of a pipe structure (GAL^2 / (EI)) can be used to judge the applicability of the element, where:

G
Shear modulus

A
Area of the cross-section

L
Length of the member (not the element length)

EI
Flexural rigidity

For pipes, (GAL^2 / EI) can be reduced to: $2L^2 / ((1 + \nu)(R_o^2 + R_i^2))$, or for thin-walled pipes: $L^2 / ((1 + \nu) R^2)$, where ν = Poisson's ratio, R_o = outer radius, R_i = inner radius, and R = average radius.

The following illustration provides an estimate of transverse-shear deformation in a cantilever pipe subjected to a tip load. Although the results cannot be extrapolated to other applications, the example serves generally. ANSYS, Inc. recommends a slenderness ratio greater than 30.

Figure 289.2: Transverse-Shear-Deformation Estimation

Slenderness Ratio (GAL^2/(EI))	δ Timoshenko / δ Euler-Bernoulli
25	1.120
50	1.060
100	1.030
1000	1.003

The element supports an elastic relationship between transverse-shear forces and transverse-shear strains.

Unlike other cubic (Hermitian) polynomial-based elements, PIPE289 is based on quadratic polynomials; therefore, offsets in specification of distributed pressure loads are not allowed. The element has linear bending-moment variation. Refinement of the mesh is recommended in order to accommodate such

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

loading. The element is computationally efficient and has super-convergence properties with respect to mesh refinement. For example, the quadratic beam with a two point Gaussian integration is known to be of same accuracy as a Hermitian element.

PIPE289 supports both the thin-pipe (KEYOPT(4) = 1) and the thick-pipe (KEYOPT(4) = 2) options. The thin-pipe option assumes a plane stress state in the pipe wall and ignore the stress in the wall thickness direction. The thick-pipe option accounts for the full 3-D stress state and generally leads to more accurate results in thick-walled pipes where through-the-thickness stress can be significant. The element allows change in cross-sectional area in large-deflection analysis. While the thick-pipe option can accurately determine the cross-section area change from the actual material constitutive properties, the thin-pipe option calculates the approximate area change based on a simple material incompressibility assumption. The thin pipe option runs slightly faster than the thick pipe option. The thick pipe option should generally be avoided for very thin-walled pipes (for example, Do/Tw > 1000.0), and the thin pipe option should generally be avoided for very thick-walled pipes (for example, Do/Tw < 10.0). The thick pipe option is required for a "solid" pipe section (Do/Tw = 2.0). For more information, see the **SECDATA** command.

For the mass matrix and load vectors, a higher order integration rule than that used for stiffness matrix is employed. The elements support both consistent and lumped mass matrices. Avoid using **LUMPM**,ON as PIPE289 is a higher-order element. Consistent mass matrix is the default behavior. You can add mass per unit length using the **SECCONTROL** command's *ADDMAS* values. See "PIPE289 Input Summary" (p. 1389).

When ocean loading is applied, the loading is nonlinear (that is, based on the square of the relative velocity between the structure and the water). Accordingly, the full Newton-Raphson option (**NROPT**,FULL) may be necessary to achieve optimal results. (Full Newton-Raphson is applied automatically in an analysis involving large-deflection effects [**NLGEOM**,ON].)

PIPE289 Input Data

The geometry, node locations, coordinate system, and pressure directions for this element are shown in Figure 289.1: PIPE289 Geometry (p. 1383). PIPE289 is defined by nodes I, J, and K in the global coordinate system. If ocean loading is present, the global origin is normally at the mean sea level, with the global Z-axis pointing away from the center of the earth; however, the vertical location can be adjusted via `Zmsl` (`Val6`) on the **OCDATA** command (following the **OCTYPE**,BASIC command).

Because the section is round, the element orientation is important only for defining offsets and temperatures, and interpreting bending moment directions and stress locations.

Node L is the preferred way to define the orientation of the element. For information about orientation nodes and beam meshing, see Generating a Beam Mesh With Orientation Nodes in the *Modeling and Meshing Guide*. See the **LMESH** and **LATT** command descriptions for details on generating the K node automatically.

You can define PIPE289 without the orientation node. The element x-axis is oriented from node I toward node J. When no orientation node is used, the default orientation of the element y-axis is automatically calculated to be parallel to the global X-Y plane. If the element is parallel to the global Z-axis (or within a 0.01 percent slope of it), the element y-axis is oriented parallel to the global Y-axis. To control the element orientation about the element x-axis, use the L (orientation) node option. If both are defined, the orientation node option takes precedence.

The orientation node L, if used, defines a plane (with I and J) containing the element x and z-axes (as shown). If this element is used in a large-deflection analysis, the location of the orientation node only *initially* orients the element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The pipe element is a one-dimensional line elements in space. The cross-section details are provided separately via the **SECTYPE** and **SECDATA** commands. A section is associated with the pipe elements by specifying the section ID number (**SECNUM**). A section number is an independent element attribute.

Internal fluid and external insulation are supported. Added mass, hydrodynamic added mass, and hydrodynamic and buoyant loading, are available via the **OCDATA** and **OCTABLE** commands. See the **SECCONTROL** command for defining added mass.

PIPE289 Cross-Sections

PIPE289 can be associated only with the pipe cross-section (**SECTYPE**,,PIPE). The material of the pipe is defined as an element attribute (MAT).

PIPE289 is provided with section-relevant quantities (area of integration, position, etc.) automatically at a number of section points using the **SECDATA** command. Each section is assumed to be an assembly of a predetermined number of cells and is numerically integrated.

Figure 289.3: Typical Cross-Section Cell

Section Flexibility

To apply section flexibility factors, use the **SFLEX** command. The command is valid only for linear material properties and small strain analyses, and does not support offsets, temperature loading, or initial state loading.

PIPE289 Loads

Element loads are described in Element Loading (p. 46). Internal fluid and external insulation are supported. Added mass, hydrodynamic added mass, and hydrodynamic and buoyant loading, are available via the **OCDATA** and **OCTABLE** commands. For ocean loading, you can define added mass directly via the **OCDATA** and **OCTABLE** commands (rather than by issuing the **SECCONTROL** command as is typical with other load types.)

Forces are applied at nodes I, J, and K. If the centroidal axis is not colinear with the element x-axis because of node-location offsets, applied axial forces will cause bending. The nodes should therefore be located at the desired points where you want to apply the forces. Use the *OFFSETY* and *OFFSETZ* arguments

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of the **SECOFFSET** command appropriately. By default, the program uses the centroid as the reference axis for the pipe elements.

Internal and external pressures are input on an average basis over the element. Lateral pressures are input as force per unit length. End "pressures" are input as forces.

Pressure Input

To input surface-load information on the element faces, issue the **SFE** command

On faces 1 and 2, internal and external pressures are input, respectively, on an average basis over the element.

On face 3, the input (*VAL1*) is the global Z coordinate of the free surface of the internal fluid of the pipe. Specify this value with **ACEL**,0,0,*ACEL_Z*, where *ACEL_Z* is a positive number. The free surface coordinate is used for the internal mass and pressure effects. If *VAL1* on the **SFE** command is zero, no fluid inside of the pipe is considered. If the internal fluid free surface should be at Z = 0, use a very small number instead. The pressure calculation presumes that the fluid density above an element to the free surface is constant. For cases where the internal fluid consists of more than one type of fluid (such as oil over water), the free surface coordinate of the elements in the fluids below the top fluid need to be adjusted (lowered) to account for the less dense fluid(s) above them.

The internal fluid mass is assumed to be lumped along the centerline of the pipe, so as the centerline of a horizontal pipe moves across the free surface, the mass changes in step-function fashion. The free surface location is stepped, even if you specify ramped loading (**KBC**,0).

On face 4 and face 5, lateral pressures are input as force-per-unit-length.

On face 6 and face 7, end "pressures" are input as forces.

Faces 4 through 8 are shown by the circled numbers in Figure 289.1: PIPE289 Geometry (p. 1383).

Temperature Input

When KEYOPT(1) = 0, temperatures can be input as element body loads at the inner and outer surfaces at both ends of the pipe element so that the temperature varies linearly through the wall thickness. If only two temperatures are specified, those two temperatures are used at both ends of the pipe element (that is, there is no gradient along the length). If only the first temperature is specified, all others default to the first. The following graphic illustrates temperature input at a node when KEYOPT(1) = 0:

When KEYOPT(1) = 1, temperatures can be input as element body loads at three locations at both nodes of the pipe element so that the temperature varies linearly in the element y and z directions. At either end of the element, temperatures can be input at these locations:

- At the element x-axis (T(0,0))

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- At the outer radius from the x-axis in the element y-direction ($T(R_o,0)$)
- At the outer radius from the x-axis in the element z-direction ($T(0,R_o)$)

The following graphic illustrates temperature input at a node when KEYOPT(1) = 1:

Element locations (T(Y,Z)) are given according to the convention used in Figure 289.1: PIPE289 Geometry (p. 1383).

For pipe elements, element body load commands (**BFE**) accept an element number and a list of values, 1 through 6 for temperatures $T_I(0,0)$, $T_I(1,0)$, $T_I(0,1)$, $T_J(0,0)$, $T_J(1,0)$, and $T_J(0,1)$. This input can be used to specify temperature gradients that vary linearly both over the cross section and along the length of the element.

The following defaults apply to element temperature input:

- If all temperatures after the first are unspecified, they default to the first. This pattern applies a uniform temperature over the entire element. (The first coordinate temperature, if unspecified, defaults to TUNIF.)
- If all three temperatures at node I are input, and all temperatures at node J are unspecified, the node J temperatures default to the corresponding node I temperatures. This pattern applies a temperature gradient that varies linearly over the cross section but remains constant along the length of the element.
- For any other input pattern, unspecified temperatures default to TUNIF.

Alternatively, temperatures at nodes I and J can be defined using nodal body loads (**BF**,*NODE*,TEMP,*VAL1*). This specifies a uniform temperature over the cross section at the specified node. (**BF** command input is not accepted at node K.)

Other Input

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, issue an **NROPT**,UNSYM command.

The end-cap pressure effect is included by default. The end-cap effect can be deactivated via KEYOPT(6). When subjected to internal and external pressures, PIPE289 with end caps (KEYOPT(6) = 0) is always in equilibrium; that is, no net forces are produced. Because the element curvature is not considered for

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the end-cap orientations, the element is also in equilibrium without end caps (KEYOPT(6) = 1), even when the element is curved.

PIPE289 Input Summary

Nodes

I, J, K, and L (the optional, but recommended, orientation node)

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ

Section Information

Accessed via **SECTYPE**,,PIPE and **SECDATA** commands.

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ), DENS, GXY, GYZ, GXZ, ALPD, BETD

Surface Loads

Pressure --

face 1- Internal pressure
face 2 - External pressure
face 3 - Z coordinate of free surface of fluid on inside of pipe
face 4 (I-J) (-z normal direction)
face 5 (I-J) (-y normal direction)
face 6 (I-J) (+x tangential direction)
face 7 (I) (+x axial direction)
face 8 (J) (-x axial direction)

I and J denote end nodes.
Use a negative value for loading in the opposite direction.
Input pressure values for faces 1, 2, and 3 via the **SFE** command. Input pressure values for faces 4 through 8 via the **SFBEAM** command
Distributed pressure offsets are not available for faces 4, 5, and 6.

Body Loads

Temperatures --

TOUT(I), TIN(I), TOUT(J), TIN(J) if KEYOPT(1) = 0.

TAVG(I), T_y(I), T_z(I), TAVG(J), T_y(J), T_z(J) if KEYOPT(1) = 1.

Special Features

Birth and death
Element technology autoselect
Initial state
Large deflection
Linear perturbation

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Nonlinear stabilization
Ocean loading
Stress stiffening

KEYOPT(1)
Temperature input

0 --
Through wall gradient

1 --
Diametral gradient

KEYOPT(4)
Hoop strain treatment

1 --
Thin pipe theory

2 --
Thick pipe theory

KEYOPT(6)
End cap loads

0 --
Internal and external pressures cause loads on end caps

1 --
Internal and external pressures do not cause loads on end caps

KEYOPT(7), KEYOPT(9), KEYOPT(11), and KEYOPT(12)
Active only when **OUTPR**,ESOL is active:

KEYOPT(7)
Output control for section forces/moments and strains/curvatures:

0 --
Output section forces/moments, strains/curvatures, internal and external pressures, effective tension, and maximum hoop stress (default)

1 --
Same as KEYOPT(7) = 0 plus current section area

2 --
Same as KEYOPT(7) = 1 plus element basis directions (X,Y,Z)

3 --
Output section forces/moments, strains/curvatures, internal and external pressures, effective tension, and maximum hoop stress extrapolated to the element nodes

KEYOPT(8)
Shear stress output:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0 --
Output a combined state of the following two types (default)

1 --
Output only torsion-related shear stresses

2 --
Output only flexure-related transverse-shear stresses

KEYOPT(9)
Output control at integration points:

0 --
None (default)

1 --
Maximum and minimum stresses/strains

2 --
Same as KEYOPT(9) = 1 plus stresses and strains at each section node

KEYOPT(11)
Output control for values extrapolated to the element and section nodes:

0 --
None (default)

1 --
Maximum and minimum stresses/strains

2 --
Same as KEYOPT(11) = 1 plus stresses and strains along the exterior boundary of the cross-section

3 --
Same as KEYOPT(11) = 1 plus stresses and strains at all section nodes

KEYOPT(12)
Hydrodynamic output (not available in harmonic analyses that include ocean wave effects (**HROCEAN**)):

0 --
None (default)

1 --
Additional centroidal hydrodynamic printout

KEYOPT(15)
Results file format:

0 --
Store averaged results at each section corner node (default).

1 --
Store non-averaged results at each section integration point. (The volume of data may be excessive.)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PIPE289 Output Data

The solution output associated with these elements is in two forms:

- Nodal displacements and reactions included in the overall nodal solution
- Additional element output as described in Table 289.1: PIPE289 Element Output Definitions (p. 1393)

For ways to view results, see the *Basic Analysis Guide*.

To view 3-D deformed shapes for PIPE289, issue an **OUTRES**,MISC or **OUTRES**,ALL command for static or transient analyses. To view 3-D mode shapes for a modal or eigenvalue buckling analysis, expand the modes with element results calculation active (via the **MXPAND** command's `Elcalc` = YES option).

Linearized Stress

It is customary in pipe design to employ components of axial stress that contribute to axial loads and bending in each direction separately; therefore, PIPE289 provides a linearized stress output as part of its SMISC output record, as indicated in the following definitions:

SDIR is the stress component due to axial load.

SDIR = Fx/A, where Fx is the axial load (SMISC quantities 1 and 14) and A is the area of the cross-section.

SByT and SByB are bending stress components.

$$SByT = -Mz * R_0 / I$$
$$SByB = Mz * R_0 / I$$
$$SBzT = My * R_0 / I$$
$$SBzB = -My * R_0 / I$$

where My, Mz are bending moments (SMISC quantities 2,15,3,16), R_0 is the outside radius, and I is the moment of inertia of the cross-section. The program uses the maximum and minimum cross-section dimensions.

Corresponding definitions for the component strains are:

$$EPELDIR = Ex$$
$$EPELByT = -Kz * R_0$$
$$EPELByB = Kz * R_0$$
$$EPELBzT = Ky * R_0$$
$$EPELBzB = -Ky * R_0$$

where Ex, Ky, and Kz are generalized strains and curvatures (SMISC quantities 7,8,9, 20,21 and 22).

The reported stresses are strictly valid only for elastic behavior of members. PIPE289 always employs combined stresses in order to support nonlinear material behavior. When the elements are associated with nonlinear materials, the component stresses can at best be regarded as linearized approximations and should be interpreted with caution.

When using KEYOPT(9) with the cubic option (KEYOPT(3) = 3), the integration point at the middle of the element is reported *last* in the integration-point printout.

The Element Output Definitions table uses the following notation:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

For the stress and strain components, X refers to axial, Y refers to hoop, and Z refers to radial.

Table 289.1: PIPE289 Element Output Definitions

Name	Definition	O	R
EL	Element number	Y	Y
NODES	Element connectivity	Y	Y
MAT	Material number	Y	Y
C.G.:X, Y, Z	Element center of gravity	Y	1
Area	Area of cross-section	2	Y
S:x, y, z, xy, yz, xz	Section point stresses	3	Y
EPEL:x, y, z, xy, yz, xz	Elastic strains	3	Y
EPTO:x, y, z, xy, yz, xz	Section point total mechanical strains (EPEL + EPPL + EPCR)	3	Y
EPTT:x, y, z, xy, yz, xz	Section point total strains (EPEL + EPPL + EPCR + EPTH)	3	Y
EPPL:x, y, z, xy, yz, xz	Section point plastic strains	3	Y
EPCR:x, y, z, xy, yz, xz	Section point creep strains	3	Y
EPTH:x, y, z, xy, yz, xz	Section point thermal strains	3	Y
NL:EPEQ	Accumulated equivalent plastic strain	-	4
NL:CREQ	Accumulated equivalent creep strain	-	4
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	-	4
NL:PLWK	Plastic work/volume	-	4
SEND:ELASTIC, PLASTIC, CREEP	Strain energy densities	-	4
TQ	Torsional moment	Y	Y
TE	Torsional strain	Y	Y
SFy, SFz	Section shear forces	2	Y
SEy, SEz	Section shear strains	2	Y
Ky, Kz	Curvature	Y	Y
Ex	Axial strain	Y	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
Fx	Axial force (excluding insulation)	Y	Y
My, Mz	Bending moments	Y	Y
INT PRESS	Internal pressure at integration point	5	5
EXT PRESS	External pressure at integration point	5	5
EFFECTIVE TENS	Effective tension on pipe	5	5
MAX HOOP STRESS	Maximum hoop stress at integration point	5	5
SDIR	Axial direct stress	-	Y
SByT	Bending stress on the element +Y side of the pipe	-	Y
SByB	Bending stress on the element -Y side of the pipe	-	Y
SBzT	Bending stress on the element +Z side of the pipe	-	Y
SBzB	Bending stress on the element -Z side of the pipe	-	Y
EPELDIR	Axial strain at the end	-	Y
EPELByT	Bending strain on the element +Y side of the pipe	-	Y
EPELByB	Bending strain on the element -Y side of the pipe	-	Y
EPELBzT	Bending strain on the element +Z side of the pipe	-	Y
EPELBzB	Bending strain on the element -Z side of the pipe	-	Y
TEMP	Temperatures at all section corner nodes	-	Y
LOCI:X, Y, Z	Integration point locations	-	6
SVAR:1, 2, ... , N	State variables	-	7
The following values apply to ocean loading only:			
GLOBAL COORD	Element centroid location	8	Y
VR, VZ	Radial and vertical fluid particle velocities (VR is always > 0)	8	Y
AR, AZ	Radial and vertical fluid particle accelerations	8	Y
PHDYN	Dynamic fluid pressure head	8	Y
ETA	Wave amplitude over integration point	8	Y
TFLUID	Fluid temperature (printed if VISC is nonzero)	8	Y
VISC	Viscosity (output if VISC is nonzero)	8	Y
REN, RET	Normal and tangential Reynolds numbers (if VISC is nonzero)	8	Y
CT	Input tangential drag coefficients evaluated at Reynolds numbers	8	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
CDY, CDZ	Input normal drag coefficients evaluated at Reynolds numbers	8	Y
CMY, CMZ	Input inertia coefficients evaluated at Reynolds numbers	8	Y
URT, URN	Tangential (parallel to element axis) and normal relative velocities	8	Y
ABURN	Vector sum of normal (URN) velocities	8	Y
AN	Accelerations normal to element	8	Y
FX, FY, FZ	Hydrodynamic tangential and normal forces in element coordinates	8	Y
ARGU	Effective position of wave (radians)	8	Y

1. Available only at the centroid as a ***GET** item, or on the NMISC record for ocean loading.
2. See KEYOPT(7) description.
3. See KEYOPT(9) and KEYOPT(11) descriptions.
4. Available if the element has a nonlinear material, or if large-deflection effects are enabled (**NLGEOM**,ON) for SEND.
5. Available only if ocean loading is present.
6. Available only if **OUTRES**,LOCI is used.
7. Available only if the `UserMat` subroutine and the **TB**,STATE command are used.
8. Applies to ocean loading only.

More output is described via the **PRESOL** and ***GET**,,SECR commands in POST1.

Table 289.2: PIPE289 Item and Sequence Numbers (p. 1395) lists output available for the **ETABLE** command using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this document for more information. The output tables use the following notation:

Name
output quantity as defined in Table 289.1: PIPE289 Element Output Definitions (p. 1393).

Item
predetermined Item label for **ETABLE**

E,I,J
sequence number for data at nodes I and J

Table 289.2: PIPE289 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
Fx	SMISC	--	1	14

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
My	SMISC	--	2	15
Mz	SMISC	--	3	16
TQ	SMISC	--	4	17
SFz	SMISC	--	5	18
SFy	SMISC	--	6	19
Ex	SMISC	--	7	20
Ky	SMISC	--	8	21
Kz	SMISC	--	9	22
TE	SMISC	--	10	23
SEz	SMISC	--	11	24
SEy	SMISC	--	12	25
Area	SMISC	--	13	26
BM	SMISC	--	27	29
BK	SMISC	--	28	30
SDIR	SMISC	--	31	36
SByT	SMISC	--	32	37
SByB	SMISC	--	33	38
SBzT	SMISC	--	34	39
SBzB	SMISC	--	35	40
EPELDIR	SMISC	--	41	46
EPELByT	SMISC	--	42	47
EPELByB	SMISC	--	43	48
EPELBzT	SMISC	--	44	49
EPELBzB	SMISC	--	45	50
TEMP	SMISC	--	51-53	54-56
INT PRESS [1]	SMISC	--	61	65
EXT PRESS [1]	SMISC	--	62	66
EFFECTIVE TENS [1]	SMISC	--	63	67
MAX HOOP STRESS [1]	SMISC	--	64	68
S:x, y, z, xy, yz, xz	LS	--	CI[2], DI[3]	CJ[2], DJ[3]
EPEL:x, y, z, xy, yz, xz	LEPEL	--	CI[2], DI[3]	CJ[2], DJ[3]
EPTH:x, y, z, xy, yz, xz	LEPTH	--	CI[2], DI[3]	CJ[2], DJ[3]
EPPL:x, y, z, xy, yz, xz	LEPPL	--	CI[2], DI[3]	CJ[2], DJ[3]
EPCR:x, y, z, xy, yz, xz	LEPCR	--	CI[2], DI[3]	CJ[2], DJ[3]

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input			
	Item	E	I	J
EPTO:x, y, z, xy, yz, xz	LEPTO	--	CI[2], DI[3]	CJ[2], DJ[3]
EPTT:x, y, z, xy, yz, xz	LEPTT	--	CI[2], DI[3]	CJ[2], DJ[3]
The following output quantities are valid for ocean loading only:				
GLOBAL COORD	NMISC	1, 2, 3	--	--
VR, VZ	NMISC	4, 5	--	--
AR, AZ	NMISC	6, 7 [4]	--	--
PHDYN	NMISC	8 [4]	--	--
ETA	NMISC	9 [4]	--	--
TFLUID	NMISC	10	--	--
VISC	NMISC	11	--	--
REN, RET	NMISC	12, 13 [5]	--	--
CT	NMISC	14	--	--
CDY, CDZ	NMISC	15, 16	--	--
CMY, CMZ	NMISC	17, 18 [4]	--	--
URT, URN	NMISC	19, 20, 21	--	--
ABURN	NMISC	22 [4]	--	--
AN	NMISC	23, 24 [4]	--	--
FX, FY, FZ	NMISC	25, 26, 27	--	--
ARGU	NMISC	28 [4]	--	--

1. Internal pressure (INT PRESS), external pressure (EXT PRESS), effective tension (EFFECTIVE TENS), and maximum hoop stress (MAX HOOP STRESS) occur at integration points, and not at end nodes.

2. CI and CJ are the sequence numbers for accessing the averaged line element solution quantities (LS, LEPEL, LEPTH, LEPPL, LEPCR, LEPTO, and LEPTT) at RST section nodes (section corner nodes where results are available), at element Node I and J respectively. CI and CJ are applicable only when KEYOPT(15) = 0. For a given section corner node *nn*, CI and CJ are given as follows:

   CI = (*nn* - 1) * 6 + COMP

   CJ = (*nnMax* + *nn* - 1) * 6 + COMP

   Where *nnMax* is the total number of RST section nodes, and COMP is the stress or strain component (1 - x, 2 - y, 3 - z, 4 - xy, 5 - yz, 6 - xz). Locations of RST section nodes can be visualized with **SECPLOT**,,6.

3. DI and DJ are the sequence numbers for accessing the non-averaged line element solution quantities (LS, LEPEL, LEPTH, LEPPL, LEPCR, LEPTO, and LEPTT) at RST section integration points (section integration points where results are available), respectively at element Node I and J. DI and DJ are applicable only when KEYOPT(15) = 1. For the ith integration point (i = 1, 2, 3, or 4) in section cell *nc*, DI and DJ are given as follows:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DI = (*nc* - 1) * 24 + (i - 1) * 6 + COMP

DJ = (*ncMax* + *nc* - 1) * 24 + (i - 1) * 6 + COMP

Where *ncMax* is the total number of RST section cells, and COMP is the stress or strain component (1 - x, 2 - x, 3 - z, 4 - xy, 5 - yz, 6 - xz). Locations of RST section cells can be visualized with **SECPLOT**,,7.

4. These quantities are output only if wave loading is used.

5. These quantities are output only if a Reynold's number dependency is used.

For more usage details, see Plot and Review the Section Results and Example Problem with Cantilever Beams.

Transverse-Shear Stress Output

The shear stresses are caused by torsional and transverse loads. PIPE289 is based on first-order shear-deformation theory, also popularly known as Timoshenko beam theory. The transverse-shear strain is constant for the cross-section; therefore, the shear energy is based on a transverse-shear force. This shear force is redistributed by predetermined shear stress distribution coefficients across the pipe cross-section, and made available for output purposes. Use KEYOPT(8) to output shear stresses caused by flexure or transverse loading.

By default, the program uses a mesh density (for cross-section modeling) that provides accurate results for torsional rigidity, warping rigidity, inertia properties, and shear center determination. The default mesh employed is also appropriate for nonlinear material calculations; however, more refined cross-section models may be necessary if the shear stress distribution due to transverse loads must be captured very accurately. Use the **SECDATA** command to adjust cross-section mesh density.

The traction-free state at the edges of the cross-section is met only in a well-refined model of the cross-section.

The transverse-shear distribution calculation ignores the effects of Poisson's ratio. The Poisson's ratio affects the shear-correction factor and shear-stress distribution slightly, and this effect is ignored.

PIPE289 Assumptions and Restrictions

- The pipe cannot have zero length.
- Cross-section distortion or collapse is not considered; therefore, section ovalization associated with a curved pipe undergoing bending cannot be modeled. Curved pipes are best modeled using ELBOW290.
- Rotational degrees of freedom are not included in the lumped mass matrix if node-location offsets are present.
- The element works best with the full Newton-Raphson solution scheme (that is, the default choice in solution control).
- Only moderately "thick" pipes can be analyzed. See "PIPE289 Element Technology and Usage Recommendations" (p. 1384) for more information.
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). Prestress effects can be activated via the **PSTRES** command.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Hydrodynamic output via KEYOPT(12) is not available in harmonic analyses that include ocean wave effects (**HROCEAN**).
- For a random vibration (PSD) analysis, equivalent stress is not calculated.
- Temperature-dependent density of the internal fluid is evaluated at an average temperature of the element. Temperature-dependent density of the insulation is evaluated at an average temperature at each integration point along the length.

PIPE289 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The only special features allowed are stress stiffening and large deflections.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

ELBOW290

3-D 3-Node Elbow

MP ME ST PR PRN <> <> <> PP EME MFS
Product Restrictions

ELBOW290 Element Description

The ELBOW290 element is suitable for analyzing pipe structures with initially circular cross-sections and thin to moderately thick pipe walls. The element accounts for cross-section distortion, which can be commonly observed in curved pipe structures under loading.

ELBOW290 is a quadratic (three-node) pipe element in 3-D. The element has six degrees of freedom at each node (the translations in the x, y, and z directions and rotations about the x, y, and z directions). The element is well-suited for linear, large rotation, and/or large strain nonlinear applications. Change in pipe thickness is accounted for in geometrically nonlinear analyses. The element accounts for follower (load stiffness) effects of distributed pressures.

ELBOW290 can be used in layered applications for modeling laminated composite pipes. The accuracy in modeling composite pipes is governed by the first-order shear-deformation theory (generally referred to as Mindlin-Reissner shell theory).

ELBOW290 supports the pipe cross-section defined via **SECTYPE**, **SECDATA**, and **SECOFFSET** commands.

For more detailed information about this element, see ELBOW290 - 3-D 3-Node Elbow in the *Mechanical APDL Theory Reference*.

Figure 290.1: ELBOW290 Geometry

A general description of the element coordinate system is available in Coordinate Systems (p. 55) in this document. Following is information about specific coordinate systems as they apply to ELBOW290.

Beam Coordinate Systems

The beam coordinate systems (x-y-z) are used for defining beam offsets and diametral temperature gradients.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

No orientation node (default)

Using orientation node L

The x axis is always the axial direction pointing from node I to node J. The optional orientation node L, if used, defines a plane containing the x and z axes at node K. If this element is used in a large-deflection analysis, the location of the orientation node L is used only to *initially* orient the element.

When no orientation node is used, z is perpendicular to the curvature plane, uniquely determined by the I, J, and K nodes. If I, J, and K are colinear, the y axis is automatically calculated to be parallel to the global X-Y plane. In cases where the element is parallel to the global Z axis (or within a 0.01 percent slope of the axis), the element y axis is oriented parallel to the global Y axis.

For information about orientation nodes and beam meshing, see Generating a Beam Mesh With Orientation Nodes in the *Modeling and Meshing Guide*. See Quadratic Elements (Midside Nodes) in the same document for information about midside nodes. For details about generating the optional orientation node L automatically, see the **LMESH** and **LATT** command descriptions.

Local Cylindrical Coordinate Systems

The cylindrical coordinate systems (A-R-T) are used for defining internal section motions (that is, axial-A, radial-R, and hoop-T displacements and rotations).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The cylindrical systems are always created from the default beam coordinate systems (beam system without orientation node L), with A being the same as beam axis x, and an angle α ($0 < \alpha < 360$ degrees) from R to the beam axis y.

Element and Layer Coordinate Systems

The element coordinate systems (e1-e2-e3) are defined at the mid-surfaces of the pipe wall. The e1, e2, and e3 axes are parallel respectively to cylindrical axes A, T, and R in the undeformed configuration. Each element coordinate system is updated independently to account for large material rotation during a geometrically nonlinear analysis. Support is not available for user-defined element coordinate systems.

The layer coordinate systems (L1-L2-L3) are identical to the element coordinate system if no layer orientation angles are specified; otherwise, the layer coordinate system can be generated by rotating the corresponding element coordinate system about the shell normal (axis e3). Material properties are defined in the layer systems; therefore, the layer system is also called the material coordinate system.

ELBOW290 Input Data

The geometry and node locations for ELBOW290 are shown in Figure 290.1: ELBOW290 Geometry (p. 1401). The element is defined by nodes I, J, and K in the global coordinate system.

When using ELBOW290, the subtended angle φ should not exceed 45 degrees:

ELBOW290 Cross Sections

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The element is a one-dimensional line element in space. The cross-section details are provided separately (via the **SECDATA** command). A section is associated with the element by specifying the section ID number (**SECNUM**). A section number is an independent element attribute.

ELBOW290 can only be associated with the pipe cross section (**SECTYPE**,,PIPE). For elements with homogeneous materials, the material of the pipe is defined as an element attribute (MAT).

The layup of a composite pipe can be defined with a shell section (**SECTYPE**). Shell section commands provide the input options for specifying the thickness, material, orientation and number of integration points through the thickness of the layers. ANSYS obtains the actual layer thicknesses used for ELBOW290 element calculations (by scaling the input layer thickness) so that they are consistent with the total wall thickness given by the pipe section. A single-layer shell section definition is possible, allowing flexibility with regard to the number of integration points used and other options.

For shell section input, you can designate the number of integration points (1, 3, 5, 7, or 9) located through the thickness of each layer. When only one integration point is specified, the point is always located midway between the top and bottom surfaces. If three or more points are specified, one point is located on the top surface, one point is located on the bottom surface, and the remaining points are distributed at equal distances between the top and bottom points. The default number of integration points for each layer is 3. When a single layer is defined and plasticity is present, however, the number of integration points is changed to a minimum of five during solution.

In "Element and Layer Coordinate Systems" (p. 1403), the layer coordinate system can be obtained by rotating the corresponding element coordinate system about the shell normal (axis e3) by angle θ (in degrees). The value of θ for each layer is given by the **SECDATA** command input for the shell section.

For details about associating a shell section with a pipe section, see the **SECDATA** command documentation.

Cross-Section Deformation

The level of accuracy in elbow cross-sectional deformation is given by the number of Fourier terms around the circumference of the cross section. The accuracy can be adjusted via KEYOPT(2) = *n*, where *n* is an integer value from 0 through 8, as follows:

KEYOPT(2) = 0 -- Only uniform radial expansion and transverse shears through the pipe wall are allowed. Suitable for simulating straight pipes without undergoing bending.
KEYOPT(2) = 1 -- Radial expansion and transverse shears are allowed to vary along the circumference to account for bending. Suitable for straight pipes in small-deformation analysis.
KEYOPT(2) = 2 through 8 -- Allow general section deformation, including radial expansion, ovalization, and warping. Suitable for curved pipes or straight pipes in large-deformation analysis. The default is KEYOPT(2) = 2. Higher values for KEYOPT(2) may be necessary for pipes with thinner walls, as they are more susceptible to complex cross-section deformation than are pipes with thicker walls.

Element computation becomes more intensive as the value of KEYOPT(2) increases. Use a KEYOPT(2) value that offers an optimal balance between accuracy and computational cost.

Cross-Section Constraints

The constraints on the elbow cross-section can be applied at the element nodes I, J, and K with the following section degrees of freedom labels:

SE – section radial motion (as occurs during expansion or ovalization, for example)
SO – section tangential motion (as occurs during ovalization, for example)

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SW – section axial motion (as occurs during warping, for example)
SRA – local shell normal rotation about cylindrical axis A
SRT – local shell normal rotation about cylindrical axis T
SECT – all section deformation

Only fixed cross-section constraints are allowed via the **D** command. Delete section constraints via the **DDELE** command. For example, to constrain the warping and ovalization of the cross-section at node *n*, issue this command:

D,*n*,SW,,,,,SO

To allow only the radial expansion of the cross-section, use the following commands:

D,*n*,SECT
DDELE,*n*,SE

It is not practical to maintain the continuity of cross-section deformation between two adjacent elements joined at a sharp angle. For such cases, ANSYS, Inc. recommends coupling the nodal displacements and rotations but leaving the cross-section deformation uncoupled. The **ELBOW** command can automate the process by uncoupling the cross-section deformation for any adjacent elements with cross-sections intersecting at an angle greater than 20 degrees.

Element loads are described in Nodal Loading (p. 49). Forces are applied at the nodes. By default, ELBOW290 element nodes are located at the center of the cross-section. Use the **SECOFFSET** command's OFFSETY and OFFSETZ arguments for the pipe section to define locations other than the centroid for force application.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pressures may be input as surface loads on the element faces as shown by the circled numbers in the following illustration. Positive pressures act into the pipe wall.

The end-cap pressure effect is included by default. The end-cap effect on one or both ends of the element can be deactivated via KEYOPT(6). When subjected to internal and external pressures, ELBOW290 with end caps (KEYOPT(6) = 0) is always in equilibrium; that is, no net forces are produced. Without end caps (KEYOPT(6) = 1), the element is also in equilibrium except for the case when the element is curved. With end caps only at one end (KEYOPT(6) = 2 or 3), the element is obviously not in equilibrium.

Pressure Load Stiffness

The effects of pressure load stiffness are included by default for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, issue an **NROPT**,UNSYM command.

Temperatures

When KEYOPT(1) = 0, a layer-wise pattern is used. T1 and T2 are temperatures at inner wall, T3 and T4 and the interface temperatures between layer 1 and layer 2, ending with temperatures at the exterior of the pipe. All undefined temperatures are default to TUNIF. If exactly (*NL* + 1) temperatures are given (where *NL* is the number of layers), then one temperature is taken as the uniform temperature at the bottom of each layer, with the last temperature for the exterior of the pipe.

When KEYOPT(1) = 1, temperatures can be input as element body loads at three locations at both end nodes of the element so that the temperature varies linearly in the beam y axis and z axis directions. At both ends, the element temperatures are input at the section centroid (TAVG), at the outer radius from the centroid in the element y direction (Ty), and at the outer radius from the centroid in the element z-direction (Tz). The first coordinate temperature TAVG defaults to TUNIF. If all temperatures after the first are unspecified, they default to the first. If all temperatures at node I are input, and all temperatures at node J are unspecified, the node J temperatures default to the corresponding node I temperatures. For any other input pattern, unspecified temperatures default to TUNIF. The following graphic illustrates temperature input when KEYOPT(1) = 1:

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Transverse Shear Stiffness

ELBOW290 includes the effects of transverse shear deformation through the pipe wall. The transverse shear stiffness of the element is a 2 x 2 matrix, as shown:

$$E = \begin{bmatrix} E_{11} & E_{12} \\ sym & E_{22} \end{bmatrix}$$

For a single-layer elbow with isotropic material, default transverse shear stiffnesses are as follows:

$$E = \begin{bmatrix} kGh & 0 \\ 0 & kGh \end{bmatrix}$$

where k = 5/6, G = shear modulus, and h = pipe wall thickness.

You can override the default transverse shear stiffness values by assigning different values via the **SECCONTROL** command for the shell section.

ELBOW290 Input Summary

Nodes

I, J, K, and L (the optional orientation node)

Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ

Section Information

Accessed via **SECTYPE**,,PIPE and **SECDATA** commands.

Material Properties

TB command: See Element Support for Material Models (p. 59) for this element.
MP command: EX, EY, EZ, (PRXY, PRYZ, PRXZ, *or* NUXY, NUYZ, NUXZ), ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ *or* THSX, THSY, THSZ), DENS, GXY, GYZ, GXZ, ALPD
Specify BETD only once for the element. (Issue the **MAT** command to assign the material property set.) REFT may be specified once for the element, or it may be assigned on a per-layer basis.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface Loads

Pressure --

Internal pressure
External pressure

Body Loads

Temperatures --

For KEYOPT(1) = 0 -- T1, T2 (at bottom of layer 1), T3, T4 (between layers 1-2); similarly for between next layers, ending with temperatures at top of layer *NL* (2 * (*NL* + 1) maximum).
For KEYOPT(1) = 1 -- TAVG(I), Ty(I), Tz(I), TAVG(J), Ty(J), Tz(J)

Special Features

Birth and death
Large deflection
Large strain
Nonlinear stabilization
Stress stiffening

KEYOPT(1)
Temperature input

0 --
Layerwise input

1 --
Diametral gradient

KEYOPT(2)
Number of Fourier terms (used for cross-sectional flexibility)

0 --
Uniform radial expansion

1 --
Nonuniform radial expansion to account for bending

2 through 8 --
General section deformation (default = 2)

KEYOPT(6)
End cap loads

0 --
Internal and external pressures cause loads on end caps

1 --
Internal and external pressures do not cause loads on end caps

2 --
Internal and external pressures cause loads on element node I

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3 --
Internal and external pressures cause loads on element node J

KEYOPT(8)
Specify layer data storage:

0 --
Store data for bottom of bottom layer and top of top layer (multilayer elements) (default)

1 --
Store data for TOP and BOTTOM, for all layers (multilayer elements)

2 --
Store data for TOP, BOTTOM, and MID for all layers; applies to single-layer and multilayer elements. (The volume of data may be considerable.)

ELBOW290 Output Data

The solution output associated with these elements is in two forms:

- Nodal displacements and reactions included in the overall nodal solution
- Additional element output as described in Table 290.1: ELBOW290 Element Output Definitions (p. 1411)

Integration Stations

Integration stations along the length and within the cross-section of the elbow are shown in Figure 290.2: ELBOW290 Element Integration Stations (p. 1409).

Figure 290.2: ELBOW290 Element Integration Stations

Element solution is available at all integration points through element printout (**OUTPR**). Solution via the POST1 postprocessor is available at element nodes and selected section integration locations (see KEYOPT(8) settings for more details).

Stress Output

Several items are illustrated in Figure 290.3: ELBOW290 Stress Output (p. 1410):

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 290.3: ELBOW290 Stress Output

KEYOPT(8) controls the amount of data output to the results file for processing with the LAYER command. Interlaminar shear stress is available at the layer interfaces. KEYOPT(8) must be set to either 1 or 2 to output these stresses in the POST1 postprocessor. A general description of solution output is given in Solution Output (p. 50). See the *Basic Analysis Guide* for ways to review results.

The element shell stress resultants (N11, M11, Q13, etc.) are parallel to the element coordinate system (e1-e2-e3), as are the shell membrane strains and curvatures (ε_{11}, κ_{11}, γ_{13}, etc.) of the element. Shell stress resultants and generalized shell strains are available via the SMISC option at the element end nodes I and J only.

ELBOW290 also outputs beam-related stress resultants (Fx, My, TQ, etc) and linearized stresses (SDIR, SByT, SByB, etc) at two element end nodes I and J to SMISC records. Beam stress resultants and linearized stresses are parallel to the beam coordinate system (x-y-z).

Linearized Stress

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

It is customary in pipe design to employ components of axial stress that contribute to axial loads and bending in each direction separately. Therefore, ELBOW290 provides a linearized stress output as part of its SMISC output record, as indicated in the following definitions:

SDIR is the stress component due to axial load.

SDIR = Fx/A, where Fx is the axial load (SMISC quantities 1 and 36) and A is the area of the cross section (SMISC quantities 7 and 42).

SByT and SByB are bending stress components.

$SByT = -Mz * y_{max} / Izz$
$SByB = -Mz * y_{min} / Izz$
$SBzT = My * z_{max} / Iyy$
$SBzB = My * z_{min} / Iyy$

where My, Mz are bending moments (SMISC quantities 2,37,3,38). Coordinates y_{max}, y_{min}, z_{max}, and z_{min} are the maximum and minimum y, z coordinates in the cross section measured from the centroid. Values Iyy and Izz are moments of inertia of the cross section.

The reported stresses are strictly valid only for elastic behavior of members. ELBOW290 always employs combined stresses in order to support nonlinear material behavior. When the elements are associated with nonlinear materials, the component stresses can at best be regarded as linearized approximations and should be interpreted with caution.

ELBOW290 does not provide extensive element printout. Because the POST1 postprocessor provides more comprehensive output processing tools, ANSYS suggests issuing the **OUTRES** command to ensure that the required results are stored in the database. To view 3-D deformed shapes for ELBOW290, issue an **OUTRES**,MISC or **OUTRES**,ALL command for static or transient analyses. To view 3-D mode shapes for a modal or eigenvalue buckling analysis, expand the modes with element results calculation active via the **MXPAND** command's `Elcalc` = YES option.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates that the item can be accessed by the Component Name method (**ETABLE**, **ESOL**). The O column indicates the availability of the items in the file `Jobname.OUT`. The R column indicates the availability of the items in the results file.

In either the O or R columns, "Y" indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and "-" indicates that the item is *not* available.

Table 290.1: ELBOW290 Element Output Definitions

Name	Definition	O	R
EL	Element number	Y	Y
NODES	Element connectivity	-	Y
MAT	Material number	-	Y
THICK	Average wall thickness	-	Y
AREA	Area of cross-section	-	Y
XC, YC, ZC	Location where results are reported	-	4

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
LOCI:X, Y, Z	Integration point locations	-	5
TEMP	T1, T2 at bottom of layer 1, T3, T4 between layers 1-2, similarly for between next layers, ending with temperatures at top of layer *NL* (2 * (*NL* + 1) maximum)	-	Y
LOC	TOP, MID, BOT, or integration point location	-	1
S:X, Y, Z, XY, YZ, XZ	Stresses	3	1
S:INT	Stress intensity	-	1
S:EQV	Equivalent stress	-	1
EPEL:X, Y, Z, XY	Elastic strains	3	1
EPEL:EQV	Equivalent elastic strains [7]	-	1
EPTH:X, Y, Z, XY	Thermal strains	3	1
EPTH:EQV	Equivalent thermal strains [7]	-	1
EPPL:X, Y, Z, XY	Average plastic strains	3	2
EPPL:EQV	Equivalent plastic strains [7]	-	2
EPCR:X, Y, Z, XY	Average creep strains	3	2
EPCR:EQV	Equivalent creep strains [7]	-	2
EPTO:X, Y, Z, XY	Total mechanical strains (EPEL + EPPL + EPCR)	3	-
EPTO:EQV	Total equivalent mechanical strains (EPEL + EPPL + EPCR)	-	-
NL:EPEQ	Accumulated equivalent plastic strain	-	2
NL:CREQ	Accumulated equivalent creep strain	-	2
NL:SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)	-	2
NL:PLWK	Plastic work/volume	-	2
NL:HPRES	Hydrostatic pressure	-	2
SEND:ELASTIC, PLASTIC, CREEP	Strain energy densities	-	2
Fx	Section axial force	-	Y
My, Mz	Section bending moments	-	Y
TQ	Section torsional moment	-	Y
SFy, SFz	Section shear forces	-	Y
SDIR	Axial direct stress	-	Y
SByT	Bending stress on the element +y side of the pipe	-	Y
SByB	Bending stress on the element -y side of the pipe	-	Y
SBzT	Bending stress on the element +z side of the pipe	-	Y

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Definition	O	R
SBzB	Bending stress on the element -z side of the pipe	-	Y
N11, N22, N12	Wall in-plane forces (per unit length)	-	Y
M11, M22, M12	Wall out-of-plane moments (per unit length)	-	Y
Q13, Q23	Wall transverse shear forces (per unit length)	-	Y
ε_{11}, ε_{22}, ε_{12}	Wall membrane strains	-	Y
κ_{11}, κ_{22}, κ_{12}	Wall curvatures	-	Y
γ_{13}, γ_{23}	Wall transverse shear strains	-	Y
SVAR:1, 2, ... , *N*	State variables	-	6

1. The subsequent stress solution repeats for top, middle, and bottom surfaces.
2. Nonlinear solution output for top, middle, and bottom surfaces, if the element has a nonlinear material, or if large-deflection effects are enabled (**NLGEOM**,ON) for SEND.
3. Stresses, total strains, plastic strains, elastic strains, creep strains, and thermal strains in the element co-ordinate system are available for output (at all section points through thickness). If layers are in use, the results are in the layer coordinate system.
4. Available only at the centroid as a ***GET** item.
5. Available via an **OUTRES**,LOCI command only.
6. Available only if the `UserMat` subroutine and **TB**,STATE command are used.
7. The equivalent strains use an effective Poisson's ratio. For elastic and thermal, you set the value (**MP**,PRXY). For plastic and creep, ANSYS sets the value at 0.5.

More output is described via the **PRESOL** and ***GET**,,SECR commands in POST1.

Table 290.2: ELBOW290 Item and Sequence Numbers (p. 1413) lists output available for the **ETABLE** command using the Sequence Number method. See Creating an Element Table in the *Basic Analysis Guide* and The Item and Sequence Number Table (p. 52) in this document for more information. The output tables use the following notation:

Name
output quantity as defined in Table 290.1: ELBOW290 Element Output Definitions (p. 1411)

Item
predetermined Item label for **ETABLE**

I,J
sequence number for data at nodes I and J

Table 290.2: ELBOW290 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	I	J
Fx	SMISC	1	36
My	SMISC	2	37

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	I	J
Mz	SMISC	3	38
TQ	SMISC	4	39
SFz	SMISC	5	40
SFy	SMISC	6	41
Area	SMISC	7	42
SDIR	SMISC	8	43
SByT	SMISC	9	44
SByB	SMISC	10	45
SBzT	SMISC	11	46
SBzB	SMISC	12	47
N11	SMISC	14	49
N22	SMISC	15	50
N12	SMISC	16	51
M11	SMISC	17	52
M22	SMISC	18	53
M12	SMISC	19	54
Q13	SMISC	20	55
Q23	SMISC	21	56
ε_{11}	SMISC	22	57
ε_{22}	SMISC	23	58
ε_{12}	SMISC	24	59
κ_{11}	SMISC	25	60
κ_{22}	SMISC	26	61
κ_{12}	SMISC	27	62
γ_{13}	SMISC	28	63
γ_{23}	SMISC	29	64
THICK	SMISC	30	65

ELBOW290 Assumptions and Restrictions

- The element cannot have zero length.
- Zero wall thickness is not allowed. (Zero thickness *layers* are allowed.)
- In a nonlinear analysis, the solution is terminated if the thickness at any integration point vanishes (within a small numerical tolerance).
- This element works best with the full Newton-Raphson solution scheme (the default behavior in solution control).

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- No slippage is assumed between the element layers. Shear deflections are included in the element; however, normals to the center wall surface before deformation are assumed to remain straight after deformation.
- If multiple load steps are used, the number of layers must remain unchanged between load steps.
- If the layer material is hyperelastic, the layer orientation angle has no effect .
- Stress stiffening is always included in geometrically nonlinear analyses (**NLGEOM**,ON). Apply prestress effects via a **PSTRES** command.
- The through-thickness stress SZ is always zero.
- The effects of fluid motion inside the pipe are ignored.

ELBOW290 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional

- The only special features allowed are stress stiffening and large deflections.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

USER300

User-Defined Element

MP ME ST PR PRN <> <> <> PP EME MFS
Product Restrictions

USER300 Element Description

ANSYS offers a convenient way create your own custom element named USER300. You can create virtually any element type.

The `UserElem` subroutine provides an interface to ANSYS code above the element level, passing all data needed to create your user-defined element, then returning all data and results from the element to update the database and files. Because access to database and file information occurs through the interface, an understanding of ANSYS database routines and file structures is rarely necessary. The interface also provides access to ANSYS code at the material level, allowing you to call ANSYS standard material subroutines from your element coding.

> For detailed instructions, see Creating a New Element via the User-Defined Element API in the *Guide to User-Programmable Features*.

USER300 Input Data

Use the **USRELEM** and **USRDOF** commands to input basic element characteristics. All other characteristics default automatically.

Special features include element convergence criteria and cutback control via the element.

The element supports parallel processing and Distributed ANSYS.

USER300 Assumptions and Restrictions

The following capabilities are not available for user-defined elements:

- Birth and death
- Superelement stress pass
- Initial stress
- Section input
- Input of fluences
- Swelling

USER300 Product Restrictions

There are no product-specific restrictions for this element.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Bibliography

[1] *Nuclear Systems Material Handbook*. Vol. 1: Design Data, Part 1: Structural Materials, Group 1: High Alloy Steels . U. S. Department of Energy, Office of Scientific and Technical Information. Oak Ridge, TN.

[2] *Nuclear Systems Material Handbook*. Vol. 1: Design Data, Part 1: Structural Materials, Group 2: Low Alloy Steels, Section 2-2 1/4 CR - 1 Mo. . U. S. Department of Energy, Office of Scientific and Technical Information. Oak Ridge, TN.

[3] F. Barlat and J. Lian. *"Plastic Behavior and Stretchability of Sheet Metals*. Part I: A Yield Function for Orthotropic Sheets Under Plane Stress Conditions". *Int. Journal of Plasticity, 5*. pg. 51-66.

[4] F. Barlat, D. J. Lege, and J. C. Brem. *"A Six-Component Yield Function for Anistropic Materials". Int. Journal of Plasticity, 7*. pg. 693-712.

[5] R. Hill. *"A Theory of the Yielding and Plastic Flow of Anisotropic Metals". Proceedings of the Royal Society of London, Series A., Vol. 193*. 1948.

[6] F. K. Chang and K. Y. Chang. *"A Progressive Damage Model for Laminated Composites Containing Stress Concentration". Journal of Composite Materials, 21*. pg. 834-855. 1987a.

[7] R. G. Dean. *Evaluation and Development of Water Wave Theories for Engineering Application*. Volume 2, Tabulation of Dimensionless Stream Function Theory Variables, Special Report No. 1, . U. S. Army Corps of Engineers, Coastal Engineering Research Center. Fort Belvoir, VA. November 1974.

[8] Michael E. McCormick. *Ocean Engineering Wave Mechanics*. Wiley & Sons. New York. 1973.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Index

A

axisymmetric elements
  general, 76
  harmonic, 83
  harmonic with nonaxisymmetric loads, 84

B

beam elements, 90
BEAM161, 573
BEAM188, 997
BEAM189, 1017

C

Cantilever beams, 1018
CIRCU124, 431
CIRCU125, 443
CIRCU94, 353
COMBI165, 611
COMBI214, 1123
COMBIN14, 147
COMBIN37, 211
COMBIN39, 225
COMBIN40, 233
CONTA171, 647
CONTA172, 665
CONTA173, 683
CONTA174, 701
CONTA175, 719
CONTA176, 739
CONTA177, 755
CONTA178, 769
CPT212, 1111
CPT213, 1117
CPT215, 1131
CPT216, 1137
CPT217, 1143

D

degenerated shape elements, 39

E

ELBOW290, 1401
element coordinate system, 55
element input, 43
  degrees of freedom, 44
  loading, 46
  material properties, 44
  name, 43
  nodal loading, 49
  nodes, 43
  properties and controls (KEYOPT settings), 50
  real constants, 46
  sections, 45
  special features, 45
element support for material properties, 59
element types
  user-defined, 41
elements
  acoustic, 104
  automatic selection of, 92
  coupled physics, 99
  current-technology , 37
  diffusion, 114
  electric and magnetic, 112
  legacy vs. current, 38
  material property support, 59
  supporting linear perturbation, 95
  thermal, 102
  user-defined, 41

F

failure criteria, 54
FLUID116, 401
FLUID129, 459
FLUID130, 463
FLUID136, 483
FLUID138, 495
FLUID139, 499
FLUID220, 1149
FLUID221, 1151
FLUID29, 179
FLUID30, 185
FLUID38, 221
FOLLW201, 1071

G

generalized plane strain, 68

H

HSFLD241, 1259
HSFLD242, 1267

I

INFIN110, 389
INFIN111, 395
INFIN47, 249
INFIN9, 129
INTER192, 1047
INTER193, 1051
INTER194, 1055

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

INTER195, 1061
INTER202, 1075
INTER203, 1079
INTER204, 1083
INTER205, 1087

L

linear perturbation
  material behavior, 96
  supported elements, 95
LINK11, 133
LINK160, 569
LINK167, 617
LINK180, 797
LINK31, 187
LINK33, 191
LINK34, 195
LINK68, 299

M

MASS166, 615
MASS21, 155
MASS71, 311
MATRIX27, 167
MATRIX50, 253
MESH200, 1065
mixed u-P formulation elements, 71
MPC184, 837
MPC184 cylindrical joint, 897
MPC184 general joint, 939
MPC184 orient joint, 927
MPC184 planar joint, 909
MPC184 point-in-plane joint, 881
MPC184 revolute joint, 853
MPC184 rigid link/beam, 843
MPC184 screw joint, 949
MPC184 slider, 849
MPC184 slot joint, 873
MPC184 spherical, 933
MPC184 translational joint, 889
MPC184 universal joint, 863
MPC184 weld joint, 921

O

output, 50
  element solution, 51
  file contents, 51
  nodal solution, 55

P

pipe elements, 90
PIPE288, 1365
PIPE289, 1383
plane strain
  generalized, 68
PLANE121, 413
PLANE13, 137
PLANE162, 587
PLANE182, 821
PLANE183, 829
PLANE223, 1153
PLANE230, 1201
PLANE233, 1217
PLANE238, 1247
PLANE25, 159
PLANE35, 201
PLANE53, 257
PLANE55, 267
PLANE75, 315
PLANE77, 321
PLANE78, 327
PLANE83, 333
PRETS179, 793
prism elements, 39

R

REINF263, 1281
REINF264, 1289
REINF265, 1297
reinforcing, 1281, 1289, 1297
reinforcing elements, 91
ROM144, 505

S

shell elements, 88
SHELL131, 467
SHELL132, 475
SHELL157, 553
SHELL163, 593
SHELL181, 805
SHELL208, 1091
SHELL209, 1101
SHELL28, 173
SHELL281, 1345
SHELL41, 241
SHELL61, 273
SOLID122, 419
SOLID123, 425
SOLID164, 605
SOLID168, 621
SOLID185, 959
SOLID186, 975
SOLID187, 991
SOLID226, 1169

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID227, 1185
SOLID231, 1207
SOLID232, 1213
SOLID236, 1227
SOLID237, 1237
SOLID239, 1251
SOLID240, 1255
SOLID272, 1305
SOLID273, 1313
SOLID278, 1321
SOLID279, 1333
SOLID285, 1359
SOLID5, 119
SOLID65, 289
SOLID70, 305
SOLID87, 341
SOLID90, 347
SOLID96, 361
SOLID97, 367
SOLID98, 379
SOLSH190, 1037
SOURC36, 207
substructuring, 41
superelements, 41
SURF151, 509
SURF152, 519
SURF153, 529
SURF154, 537
SURF156, 547
SURF159, 559
SURF251 element, 1273
SURF252 element, 1277

T

TARGE169, 627
TARGE170, 635
tetrahedral elements, 39
TRANS126, 449
triangle elements, 39

U

user-defined elements, 41
USER300, 1417
USER300 custom element, 41

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.