Marc® 2020

Feature Pack 1

Volume B: Element Library

Corporate
MSC Software Corporation
4675 MacArthur Court, Suite 900
Newport Beach, CA 92660
Telephone: (714) 540-8900
Toll Free Number: 1 855 672 7638
Email: americas.contact@mscsoftware.com

Europe, Middle East, Africa
MSC Software GmbH
Am Moosfeld 13
81829 Munich, Germany
Telephone: (49) 89 431 98 70
Email: europe@mscsoftware.com

Japan
MSC Software Japan Ltd.
Shinjuku First West 8F
23-7 Nishi Shinjuku
1-Chome, Shinjuku-Ku
Tokyo 160-0023, JAPAN
Telephone: (81) (3)-6911-1200
Email: MSCJ.Market@mscsoftware.com

Asia-Pacific
MSC Software (S) Pte. Ltd.
100 Beach Road
#16-05 Shaw Tower
Singapore 189702
Telephone: 65-6272-0082
Email: APAC.Contact@mscsoftware.com

Worldwide Web
www.mscsoftware.com

Support
http://www.mscsoftware.com/Contents/Services/Technical-Support/Contact-Technical-Support.aspx

Disclaimer

MA:V2020.1:Z:Z:Z:DC-VOL-B-PDF

Documentation Feedback

At MSC Software, we strive to produce the highest quality documentation and welcome your feedback. If you have comments or suggestions about our documentation, write to us at: documentation-feedback@mscsoftware.com.

Please include the following information with your feedback:

- Document name
- Release/Version number
- Chapter/Section name
- Topic title (for Online Help)
- Brief description of the content (for example, incomplete/incorrect information, grammatical errors, information that requires clarification or more details and so on).
- Your suggestions for correcting/improving documentation

You may also provide your feedback about MSC Software documentation by taking a short 5-minute survey at: http://msc-documentation.questionpro.com.

Note:	The above mentioned e-mail address is only for providing documentation specific feedback. If you have any technical problems, issues, or queries, please contact Technical Support.

C O N T E N T S

Marc Volume B: Element Library

Table of Contents

1
Introduction

- Library Elements, 9
- Incompressible and Nearly Incompressible Elements (Herrmann Formulation), 9
- Reduced Integration Elements, 10
- Reduced Integration with Hourglass Control, 10
- Assumed Strain Elements, 11
- Constant Dilatational Elements, 11
- Elements with Strain Smoothing, 11
- Solution Procedures for Large Strain Analysis, 11
  - Large Strain Elasticity, 11
  - Large Strain Plasticity, 12
- Distributed Loads, 13
  - Nontable Driven Input, 13
  - Table Driven Input, 15
- Shell Layer Convention, 16
  - Beams in Plane — Elements 16 and 45, 16
  - Axisymmetric Shells — Elements 1, 15, 87, 88, 89, and 90, 16
  - Curvilinear Coordinate Shell — Elements 4, 8, and 24, 16
  - Triangular Shell — Element 49 and 138, 16
  - Shell Elements 22, 72, 75, 85, 86, 139, and 140, 16
  - Shell Elements 50, 85, 86, 87, and 88, 17
- Transverse Shear for Thick Beams and Shells, 17
- Beam Elements, 17
  - Length of Element, 18
  - Orientation of Beam Axis, 18
  - Cross-section Properties, 19
  - Solid Sections, 24
  - Cross-section Orientation, 27
  - Location of the Local Cross-section Axis, 28
- Element Characteristics, 31
- Follow Force Stiffness Contribution, 59

- Explicit Dynamics, 59
- Local Adaptive Mesh Refinement, 59

2 Marc Element Classifications

- Class 1, 63
- Class 2, 64
- Class 3, 65
- Class 4, 67
- Class 5, 69
- Class 6, 71
- Class 7, 72
- Class 8, 74
- Class 9, 76
- Class 10, 78
- Class 11, 80
- Class 12, 81
- Class 13, 82
- Class 14, 83
- Class 15, 85
- Class 16, 87
- Class 17, 88
- Class 18, 89
- Class 19, 90
- Class 20, 91
- Class 21, 92
- Class 22, 93
- Special Elements, 94

3 Element Library

- Element 1, 110
- Element 2, 113
- Element 3, 116
- Element 4, 120
- Element 5, 125
- Element 6, 129
- Element 7, 132
- Element 8, 138
- Element 9, 144
- Element 10, 147
- Element 11, 151
- Element 12, 155
- Element 13, 160
- Element 14, 165
- Element 15, 171
- Element 16, 175
- Element 17, 179
- Element 18, 187
- Element 19, 191
- Element 20, 196
- Element 21, 200
- Element 22, 206
- Element 23, 211
- Element 24, 213
- Element 25, 219
- Element 26, 224
- Element 27, 228
- Element 28, 232
- Element 29, 236
- Element 30, 241
- Element 31, 245
- Element 32, 249
- Element 33, 254
- Element 34, 259
- Element 35, 264
- Element 36, 270
- Element 37, 272
- Element 38, 274
- Element 39, 276
- Element 40, 279
- Element 41, 282
- Element 42, 285
- Element 43, 288
- Element 44, 292
- Element 45, 296
- Element 46, 300
- Element 47, 302
- Element 48, 304
- Element 49, 307
- Element 50, 312
- Element 51, 316
- Element 52, 319
- Element 53, 324
- Element 54, 329
- Element 55, 333
- Element 56, 338
- Element 57, 343
- Element 58, 350
- Element 59, 355
- Element 60, 360
- Element 61, 366
- Element 62, 372
- Element 63, 376
- Element 64, 380

- Element 65, 383
- Element 66, 385
- Element 67, 390
- Element 68, 394
- Element 69, 396
- Element 70, 399
- Element 71, 402
- Element 72, 406
- Element 73, 412
- Element 74, 416
- Element 75, 420
- Element 76, 425
- Element 77, 431
- Element 78, 436
- Element 79, 441
- Element 80, 446
- Element 81, 450
- Element 82, 455
- Element 83, 459
- Element 84, 463
- Element 85, 469
- Element 86, 474
- Element 87, 479
- Element 88, 483
- Element 89, 487
- Element 90, 491
- Element 91, 495
- Element 92, 498
- Element 93, 501
- Element 94, 504
- Element 95, 507
- Element 96, 512
- Element 97, 517
- Element 98, 519
- Element 99, 525
- Element 100, 527
- Element 101, 529
- Element 102, 532
- Element 103, 535
- Element 104, 538
- Element 105, 541
- Element 106, 544
- Element 107, 547
- Element 108, 550
- Element 109, 553
- Element 110, 557
- Element 111, 560
- Element 112, 564
- Element 113, 568
- Element 114, 572
- Element 115, 576
- Element 116, 580
- Element 117, 584
- Element 118, 589
- Element 119, 593
- Element 120, 597
- Element 121, 602
- Element 122, 605
- Element 123, 608
- Element 124, 611
- Element 125, 615
- Element 126, 619
- Element 127, 623
- Element 128, 627
- Element 129, 631
- Element 130, 635
- Element 131, 640
- Element 132, 643
- Element 133, 646
- Element 134, 649
- Element 135, 653
- Element 136, 656
- Element 137, 660
- Element 138, 663
- Element 139, 668
- Element 140, 673
- Element 141, 679
- Element 142, 680
- Element 143, 683
- Element 144, 685
- Element 145, 688
- Element 146, 691
- Element 147, 693
- Element 148, 695
- Element 149, 697
- Element 150, 703
- Element 151, 709
- Element 152, 713
- Element 153, 717
- Element 154, 722
- Element 155, 727
- Element 156, 731
- Element 157, 735
- Element 158, 739
- Element 159, 742
- Element 160, 743
- Element 161, 748
- Element 162, 752
- Element 163, 756
- Element 164, 762
- Element 165, 766
- Element 166, 768
- Element 167, 770
- Element 168, 772

- Element 169, 774
- Element 170, 776
- Element 171, 778
- Element 172, 780
- Element 173, 782
- Element 174, 784
- Element 175, 786
- Element 176, 790
- Element 177, 793
- Element 178, 796
- Element 179, 799
- Element 180, 802
- Element 181, 805
- Element 182, 808
- Element 183, 811
- Element 184, 813
- Element 185, 817
- Element 186, 823
- Element 187, 826
- Element 188, 829
- Element 189, 833
- Element 190, 837
- Element 191, 840
- Element 192, 843
- Element 193, 847
- Element 194, 851
- Element 195, 854
- Element 196, 858
- Element 197, 861
- Element 198, 864
- Element 199, 868
- Element 200, 872
- Element 201, 875
- Element 202, 878
- Element 203, 882
- Element 204, 885
- Element 205, 888
- Element 206, 892
- Element 207, 895
- Element 208, 896
- Element 209, 897
- Element 210, 898
- Element 211, 899
- Element 212, 900
- Element 213, 901
- Element 214, 902
- Element 215, 903
- Element 216, 904
- Element 217, 908
- Element 218, 911
- Element 219, 915
- Element 220, 918
- Element 221, 921
- Element 222, 924
- Element 223, 927
- Element 224, 930
- Element 225, 933
- Element 226, 936
- Element 227, 939
- Element 228, 942
- Element 229, 945
- Element 230, 949
- Element 231, 952
- Element 232, 956
- Element 233, 959
- Element 234, 963
- Element 235, 967
- Element 236, 971
- Element 237, 976
- Element 238, 980
- Element 239, 984
- Element 240, 988
- Element 241, 992
- Element 242, 996
- Element 243, 1002
- Element 244, 1007
- Element 245, 1009
- Element 246, 1011
- Element 247, 1013

Preface

- About This Guide 5
- Purpose of This Guide 5
- Contents of This Guide 6
- Typographical Conventions 6
- Accessing Marc Manuals 7
- Training and Internet Resources 8
- Technical Support 11

About This Guide

The *Marc Volume B: Element Library*, the second in a series of five volumes documenting the Marc Finite Element program, discusses various elements and the information required to use them. The guide also explains categorization of these elements based on their respective node number, interpolation functions, and integration schemes. You will find references to the following documents throughout this manual:

Marc Documentation

TITLE	VOLUME
Theory and User Information	Volume A
Program Input	Volume C
User Subroutines and Special Routines	Volume D
Demonstration Problems	Volume E
Release Guide	
Installation and Operations Guide	
Python: Tutorial and Reference Manual	

Purpose of This Guide

The purpose of this volume is:

1. To give information about elements available in Marc program and data necessary to use them.
2. To provide classification of different elements embedded in Marc Finite Element Program.
3. To describe each element type with regard to geometric information that user requires at input along with the output associated with it.

Contents of This Guide

The principal categories of information are found under the following titles:

Chapter	Title	Description
Chapter 1	Introduction	Contains a short description of each element from the Marc's extensive element library and a summary of the data necessary for use of those elements.
Chapter 2	Marc Element Classifications	Demonstrates classification of the elements of Marc Finite Element Program based on their respective node number, interpolation functions, and integration schemes.
Chapter 3	Element Library	Describes each element type with regard to geometric information that user requires at input. The output associated with each element type is also discussed.

Typographical Conventions

The section provides a brief overview of the typographical conventions used in the document to help you better follow the *Marc and Mentat* documentation.

This section describes some syntax that will help you in understanding text in the various chapters and thus in facilitating your learning process. It contains stylistic conventions to denote user action, to emphasize particular aspects of Marc and Mentat to signal other differences within the text.

Adobe Garamond Pro	Body and general text
Courier New	■ Represents command-line options of Marc and Mentat. ■ Directory names and paths ■ File names and Paths ■ Linux terminal script Example: `lmreread -c <parent>/msc/MSC.Licensing/licenses/license.dat`
Bold Text	■ Highlights ■ Dialog box ■ Buttons ■ Menus ■ The commands/user inputs for all descriptions related to terminal commands. ■ Default values Example: `[root@vm-tmrhel73 MSC]#` **./msc_licensing_helium_linux64.bin**

HelveticaNeueLT Pro Cn 57	■ Hyperlinks ■ Weblinks Example: Appendix. A: Microsoft Windows: Marc Subdirectories and Installation
Italic Text	Represents references to books. Example: *Volume A: Theory and User Information*
20XX	Represents the latest version number. If the release is 2019, XX stands for 19.

Accessing Marc Manuals

This section describes how to access the Marc documentation outside of MSC Software. Marc documentation is available through PDF files. The PDF files can be obtained from the following sources:

- Marc documentation installer
- SimCompanion
- Combined documentation

The PDF documentation files are appropriate for viewing and printing with Adobe Acrobat Reader (version 5.0 or higher), which is available for most Windows and Linux systems. These files are identified by a .pdf suffix in their file names.

Downloading the PDF Documentation Files

You can download the PDF documentation from SimCompanion (http://simcompanion.mscsoftware.com).

Navigating the PDF Files

For the purpose of easier online document navigation, the PDF files contain hyperlinks in the table of contents and index. In addition, links to other guides, hyperlinks to all cross-references to chapters, sections, figures, tables, bibliography, and index entries have been applied.

Printing the PDF Files

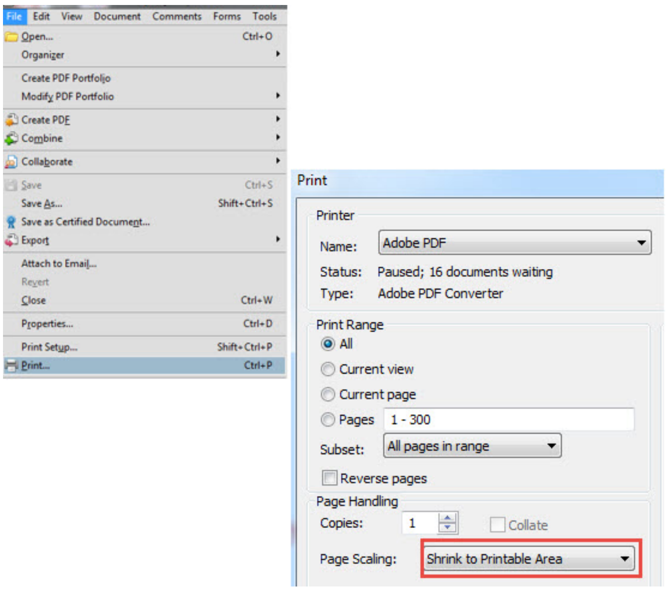
Adobe Acrobat PDF files are provided for printing all or part of the manuals. You can select the paper size to which you are printing in Adobe Acrobat Reader by doing the following:

1. Click **File**.
2. Select the **Print**.... option. The **Print** dialog box is displayed.
3. Select **Page Setup**....
4. Choose the required paper size in the **Page Setup** menu.

The PDF files are recommended when printing long sections since the printout will have a higher quality.

If the page is too large to fit on your paper size, you can reduce it by doing the following:

1. Select the **File -> Print**.
2. Under **Page Scaling,** choose the **Shrink to Printable Area** option.



Training and Internet Resources

Information about MSC's products, services and latest events is available on our website www.mscsoftware.com.

The information about MSC Seminars is available on the training link http://www.mscsoftware.com/msc-training. You can also use this link to schedule the seminars.

If you are a new Marc user, we recommend the following courses:

MAR101 - Basic Nonlinear Analysis using Marc and Mentat

The purpose of this course is to introduce the new Marc user to both Marc and Mentat by lectures and hands on modeling of nonlinear problems.

Pre-requisites:

A basic knowledge of statics and strength of materials is highly recommended. Previous finite element analysis experience is recommended.

Topics:

- Introduction to Mentat
- Nonlinear Finite Element Analysis

- Resolving Convergence Problems
- Numerical Analysis of Nonlinear Problems

MAR102 - Advanced Nonlinear Analysis using Marc and Mentat

The purpose of this course is to enhance the current Marc user's understanding of modeling nonlinear problems. Lectures are supported by hands-on modeling of nonlinear problems.

Pre-requisites:

A basic knowledge of nonlinear simulations - Familiarity with Mentat 2011 - Completion of MAR101 (Basic Nonlinear Analysis using Marc and Mentat) or equivalent experience.

Topics:

- Material Nonlinearity
- Contact
- Adaptive Meshing
- User Subroutines in Marc
- Heat Transfer and Thermal Stresses
- Global - Local (Structural Zooming) Analysis in Marc
- Restarts
- Performance
- Workshop Problems

MAR103 - Experimental Elastomer Analysis

The purpose of this course is to provide a fundamental understanding of how material testing and finite element analysis are combined to improve the design of rubber and elastomeric products.

Pre-requisites:

A basic knowledge of statics and strength of materials is highly recommended. Previous finite element analysis experience is recommended. And the knowledge of elastomeric materials.

Topics:

- Introduction
- Overview of Elastomer Testing and Analysis
- Uniaxial Tension/Compression Testing and Analysis
- Biaxial Tension/Compression Testing
- Pure Shear Testing
- Product Simulations with Specimen Data

MAR120 - Basic Nonlinear Analysis using Marc and Patran

MAR120 covers the use of Marc and Patran or AFEA (the interlocked combination of Patran and Marc) for the solution of complex engineering problems. Students who successfully complete this course will be able to: create finite element models representing nonlinear physical phenomena; select appropriate element types and mesh densities; understand the limitations of solving nonlinear FEA problems; select solution types for various nonlinear phenomena such as nonlinear dynamics, metal forming, elastomers, and contact problems; select error tolerance parameters and properly use automatic time-stepping techniques; and understand the basis of large deformation, rotation, and strain finite element analysis. Patran provides a Marc Preference which directly supports most Marc features and indirectly supports all Marc features. MSC customers that have been using Advanced FEA (which is replaced by AFEA) for meeting their analysis needs will find this new Marc Preference to be the ideal environment to continue their work. They are especially encouraged to attend this course. All the class practice (16 exercises) is made using Patran and Marc rather than Marc and Mentat. Engineers who have attended the MAR101 and MAR102 will also benefit from attending this class if they intend to use the Patran Marc Preference.

Pre-requisites:

A basic knowledge of statics and strength of materials is highly recommended and previous finite element analysis experience is recommended.

MAR121 - Advanced Nonlinear Analysis using Marc and Patran

The purpose of this course is to enhance the current Marc user's understanding of modeling nonlinear problems. Lectures are supported by hands-on modeling of nonlinear problems.

Pre-requisites:

MAR120 - Basic Nonlinear Analysis using Marc and Patran

Topics:

- Expand knowledge from MAR120 (Basic Nonlinear Analysis using Marc and Patran) course
- Practical aspects of rubber simulation
- Creep
- Superplastic forming
- Composite failure techniques
- Advanced contact techniques
- Adaptive meshing
- User subroutines
- Global/Local modeling
- Heat transfer and thermal stress
- Coupled Thermal/Structural analysis
- Restarts
- Performance
- Workshop Problems

Technical Support

If you encounter difficulties while using Marc, first please refer to the section(s) of the manual containing information on the commands you are trying to use or the type of problem you are trying to solve.

Visit SimCompanion

The product documentation is available in SimCompanion (http://simcompanion.mscsoftware.com). The SimCompanion gives you access to a wealth of resources for MSC Software products. You will find various information such as:

- Product documentations
- Knowledge base articles
- Product error lists (fixed and known issues for each release)
- SimAcademy webinars
- Product and support contact information

SimCompanion is a searchable database which allows you to find articles relevant to your inquiry. Valid MSC customer entitlement and login is required to access the database and documents. It is a single sign-on that gives you access to product documentation for complete list of products from MSC Software, allows you to manage your support cases, and participate in our discussion forums.

Help Us Help You

Clients frequently call up the support engineers at MSC Software with enquiry regarding models that do not run correctly. Our technical support staff can help you much more efficiently and effectively if you are working with a small model, since debugging a small model is much easier, and the turnaround time to rerun a (hopefully) corrected test model is minutes rather than hours.

- For information on the latest events, products and services for all products, refer to the MSC Software corporate site (www.mscsoftware.com).
- For technical support phone numbers and contact information, please visit: http://www.mscsoftware.com/Contents/Services/Technical-Support/Contact-Technical-Support.aspx.

1 Introduction

- Library Elements 9
- Incompressible and Nearly Incompressible Elements (Herrmann Formulation) 9
- Reduced Integration Elements 10
- Reduced Integration with Hourglass Control 11
- Assumed Strain Elements 11
- Constant Dilatational Elements 11
- Elements with Strain Smoothing 11
- Solution Procedures for Large Strain Analysis 11
- Distributed Loads 13
- Shell Layer Convention 16
- Transverse Shear for Thick Beams and Shells 17
- Beam Elements 17
- Element Characteristics 31
- Follow Force Stiffness Contribution 59
- Explicit Dynamics 59
- Local Adaptive Mesh Refinement 59

Library Elements

Marc contains an extensive element library. These elements provide coverage of plane stress and plane strain structures, axisymmetric structures (shell type or solid body, or any combination of the two), plate, beam and arbitrary shell structures, and full three-dimensional solid structures. A short description of each element and a summary of the data necessary for use of the elements is included in this section. Note that many elements serve the same purpose. Where possible, we indicate the preferred element that should be used. In general, we find that the lower-order quadrilateral elements give significantly better results than triangular elements in two dimensions. The higher-order isoparametric elements are more accurate, especially for problems involving thermal dependence. When using the contact capability, lower-order elements are advantageous. The CENTROID parameter should only be used for linear analyses. Plate analysis can be performed by degenerating one of the shell elements. The elements with midside nodes require a large bandwidth for the solution of the master stiffness matrix. As a general comment, you should be aware that the sophisticated elements in Marc enable problems to be solved with many fewer elements when compared to solutions with conventional constant stress elements. This requires you to exercise your analytical skills and judgement. In return, you are rewarded by a more accurate stress and displacement picture.

In this and subsequent sections, references are made to the first and second nodes, etc. of an element in order to define either the direction or sequence of the nodes. This order of the nodes is that which is defined by the connectivity matrix for the structure and is input by the CONNECTIVITY model definition option of Marc Volume C: Program Input .

Notes:	For all elements right-handed coordinate systems are used. For all two-dimensional elements, right-handed rotation is counterclockwise in the plane. In this chapter, nodes are numbered in the order that they appear in the connectivity matrix. These numbers are, of course, replaced by the appropriate node numbers for an actual structural model. For all shell elements, stress and stiffness states are calculated at eleven representative points through the thickness unless modified using the SHELL SECT parameter or through the COMPOSITE model definition option.
	If axisymmetric structures are modeled using axisymmetric elements, then the global x-axis is the axis of revolution, and the global y-axis indicates the radial direction. The y values of the coordinates of the element nodes cannot be negative.
	All shear strains are engineering values, not tensor values.

Incompressible and Nearly Incompressible Elements (Herrmann Formulation)

Certain elements in Marc (Elements 32-35, 58-61, 63, 66, 74, 80-84, 118-120, 128-130, 155-157 and 247) allow the study of incompressible and nearly incompressible materials in plane strain, axisymmetric, and three-dimensional cases through the use of a perturbed Lagrangian variational principle based on the Herrmann Formulation (see Marc Volume A: Theory and User Information). These elements can be used for large strain elasticity as well as plasticity. These elements are also used in rigid-plastic flow analysis problems (see Marc Volume A: Theory and User Information). The elements can also be used to advantage for compressible elastic materials, since their hybrid formulation usually gives more accurate stress prediction.

One of the important applications of the elements is large strain rubber elasticity with total Lagrange formulation. These elements can also be used for large strain rubber elasticity with Updated Lagrange procedure. However, for large strain rubber elasticity, it is often more efficient to use conventional displacement based elements (for example, elements 7, 10, and 11) with Updated Lagrange procedure.

The Herrmann elements can be used for large strain elastic-plasticity behavior when the multiplicative elastic-plastic model is used.

Reduced Integration Elements

For a number of isoparametric elements in Marc (Elements 22, 53-61, 69-71, 73 and 74, 114-123, 140), a reduced integration scheme is used to determine the stiffness matrix of the element. In such a reduced scheme, the integration is not exact, the contribution of the highest order terms in the deformation field is neglected. Reduced integration elements have specific advantages and disadvantages. The most obvious advantage is the reduced cost for element assembly. This is specifically significant for the three-dimensional elements (Elements 57, 61, 71, 117, 120, and 123). Another advantage is the improved accuracy which can be obtained with reduced integration elements for higher-order elements. The increase in accuracy is due to the fact that the higher-order deformation terms are coupled to the lower-order terms. The coupling is strong if the elements are distorted or the material compressibility becomes low. The higher-order terms cause strain gradients within the element which are not present in the exact solution. Hence, the stiffness is overestimated. Since the reduced integration scheme does not take the higher-order terms into account, this effect is not present in the reduced integration elements.

The same feature also forms the disadvantage of the element. Each of the reduced integration elements has some specific higher-order deformation mode(s) which do not give any contribution to the strain energy in the element. The planar elements have one such “breathing” or “hourglass” mode, shown in Figure 1-1. Whereas, the three-dimensional bricks have six breathing modes. Breathing modes can become dominant in meshes with a single array (8-node quads) or single stack (20-node bricks) of elements. In meshes using higher-order elements of this type, sufficient boundary conditions should be prescribed to suppress the breathing modes, or the exact integration element should be used. It can also be advantageous to combine reduced integration elements with an element with exact integration in the same mesh – this is always possible in Marc.

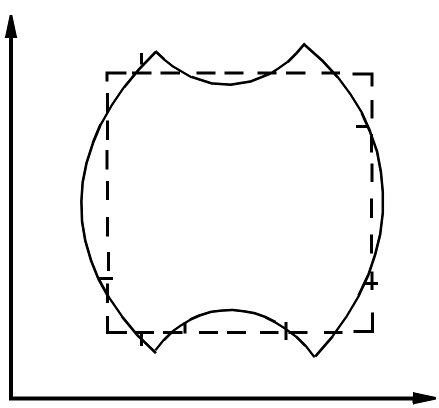


Figure 1-1 Breathing” or “Hourglass” Mode

Reduced Integration with Hourglass Control

The lower-order reduced integration elements (114 to 123, and 140) are formulated with an additional contribution to the stiffness matrix based upon a consistent Hu-Washizu variational principle to eliminate the hourglass modes normally associated with reduced integration elements. These elements use modified shape functions based upon the natural coordinates of the element, similar to the assumed strain elements. They are very accurate for elastic bending problems. You should note that as there is only one integration point for each element, elastic-plastic behavior is only evaluated at the centroid which can result in a loss in accuracy if only a single element is used through the thickness. Additionally, these elements do not lock when incompressible or nearly incompressible behavior is present. Unlike the assumed strain elements or the constant dilatation elements, no additional flags are required on the GEOMETRY option.

Assumed Strain Elements

For a number of linear elements in Marc (Elements 3, 7, 11, 160, 161, 163, and 185), a modified interpolation scheme can be used which improves the bending characteristics of the elements. This allows the ability to capture pure bending using a single element through the thickness. It is activated by use of the ASSUMED STRAIN parameter or by setting the third field of the GEOMETRY option to one. This can substantially improve the accuracy of the solution though the stiffness assembly computational costs increases.

Constant Dilatational Elements

For a number of linear elements in Marc (Elements 7, 10, 11, 19, 20, 136, 149, 151, 152 and 216), an optional integration scheme can be used which imposes a constant dilatational strain constraint on the element. This is often useful for inelastic analysis where incompressible or nearly incompressible behavior occurs. This option is automatically activated for low-order continuum elements by using the LARGE STRAIN or CONSTANT DILATATION parameter. It can also be activated by setting the second field of the GEOMETRY option to one.

Elements with Strain Smoothing

There are a few elements in Marc (Elements 239, 240 and 241) which use a strain smoothing procedure. Compared to the conventional linear triangular and tetrahedral elements (Elements 6, 2 and 134), their response for incompressible materials is better. Compared to the Herrmann type elements (Elements 155, 156 and 157), their response in bending is better and more material models are supported. Among other things, the smoothing procedure impacts the bandwidth of the global stiffness matrix, so the computational costs of these elements are relatively high.

Solution Procedures for Large Strain Analysis

There are two procedures, the total Lagrange and the Updated Lagrange, in Marc for the solution of large strain problems; each influences the choice of element technology and the resultant quantities. Large strain analysis is broadly divided between elastic analyses, using either the Mooney, Ogden, Gent, Arruda-Boyce, or Foam model, and plasticity analyses using the von Mises yield criteria. Any analysis can have a mixture of material types and element types which results in multiple solution procedures. For additional information, see Marc Volume C: Program Input , Chapter 3: Model Definition Options, Tables 3-12 and 3-13.

Large Strain Elasticity

The default procedure is the total Lagrange method, and is applicable for the Mooney, Ogden, and Foam model. When using the Mooney or Ogden material models for plane strain, generalized plane strain, axisymmetric, axisymmetric with twist, or three-dimensional solids in the total Lagrange frame work, Herrmann elements must be used to satisfy the incompressibility constraint. For plane stress, membrane, or shell analysis with these models, conventional displacement elements should be used. The elements will thin to satisfy the incompressibility constraint. When using the Foam material model, conventional displacement elements should be used. Using this procedure, the output includes the second Piola-Kirchhoff stress, the Cauchy (true) stress and the Green-Lagrange strain.

The updated Lagrange procedure can be used with either the Mooney, Ogden, Arruda-Boyce, or Gent material models. This procedure is not yet available for plane stress, membrane, or shell elements. A two-field variational principal is used to insure that incompressibility is satisfied. For more details, see Marc Volume A: Theory and User Information. Using this procedure, the output includes the Cauchy (true) stress and the logarithmic strain.

It is often more efficient to use conventional displacement based elements (elements 7, 10, and 11) with Updated Lagrange procedure for large strain elasticity.

Large Strain Plasticity

The updated Lagrange procedure should be used for large strain plasticity problems. There are two methods used for implementing the elastic-plastic kinematics: the additive decomposition and the multiplicative decomposition procedure. With the dominance of plasticity in the solution, the deformation becomes (approximately) incompressible. Both decomposition procedures work with conventional displacement elements for plane stress, plane strain, axisymmetric, or three-dimensional solids. When using the additive decomposition procedure with lower-order continuum elements (for example, elements 7, 10, 11, 19 and 20), the modified volume strain integration (constant dilatation) approach should be used.

When using the multiplicative decomposition procedure, a two-field variational procedure is used to satisfy the nearly incompressible condition for plane strain, axisymmetric, and three-dimensional solids. This can be used for all lower- and higher-order displacement elements as well as Herrmann elements. This is summarized in Tables 1-1 and 1-2.

Table 1-1 Large Strain Elasticity Element Selection

	Truss, Beam	Membrane	Shell	Plane Stress	Plane Strain Axisymmetric 3-D Solid	Strain Measure	Stress Measure
Total Lagrange	conv.*	conv.	conv.	conv.	Herrmann	Green-Lagrange	2nd Piola-Kirchhoff
Updated Lagrange	N/A	N/A	N/A	N/A	conv/Herrmann	Logarithmic	Cauchy
* conv. stands for conventional displacement formulation.							

Table 1-2 Large Strain Plasticity Element Selection

	Truss, Beam	Membrane	Shell	Plane Stress	Plane Strain Axisymmetric 3-D Solid	Strain Measure	Stress Measure
Updated Lagrange Additive Decomposition	conv.*	conv.	conv.	conv.	conv.	Logarithmic	Cauchy
Updated Lagrange Multiplicative Decomposition	N/A	N/A	N/A	conv.	conv/Herrmann	Logarithmic	Cauchy
* conv. stands for conventional displacement formulation.							

Output

When using the total Lagrange procedure or the updated Lagrange procedure, the strain and stress output is in the global x-y-z directions for continuum elements. For beam or shell elements, the output is in the local, element coordinate system based upon the original coordinate orientation for the total Lagrange procedure. For beams or shells, the output is in a co-rotational system attached to the deformed element if the updated Lagrange procedure is used.

Distributed Loads

There are two methods to define distributed loads on elements. The first method uses the nontable driven input. The second method uses the table driven input available in version Marc 2005 and later.

Nontable Driven Input

The first method requires the specification of the `IBODY` on the DIST LOAD, DIST FLUXES, etc. options. The `IBODY` codes defined in this manual for each element type provide information about the type of load and the edge or face they are applied to. The type of load includes normal pressures, shear loads (on selective element types), and volumetric loads. The `IBODY` also identifies whether the FORCEM, FLUX, FILM, etc. user subroutines are used. In general, unless noted elsewhere, a positive pressure in edges or faces results in a stress in the direction opposing the outward normal. The volumetric load types are not always defined in the remainder of this manual because they are consistent for all element types. They can be summarized in Table 1-3.

Table 1-3 Volumetric Load Types

IBODY	Load Type	Applicable Elements
100	Centrifugal load based on specifying ω^2 (ω is the angular velocity in radians/time) and $d\omega / dtime$ (which is the angular acceleration)	All mechanical
101	Heat generated by inelastic work	All thermal
102	Gravity; specify the acceleration due to gravity	All mechanical
103	Centrifugal and Coriolis load based on specifying ω^2 (ω is the angular velocity in radians/time) and $d\omega / dtime$ (which is the angular acceleration)	All mechanical
104	Centrifugal load based on specifying ω (ω is the angular velocity in cycles/time) and $d\omega / dtime$ (which is the angular acceleration)	All mechanical
105	Centrifugal and Coriolis load based on specifying ω (ω is the angular velocity in cycles/time) and $d\omega / dtime$ (which is the angular acceleration)	All mechanical
106	Uniform load per unit volume or uniform flux per unit volume	All mechanical, thermal, or electromagnetic
107	Nonuniform load per unit volume or nonuniform flux per unit volume	All mechanical, thermal, or electromagnetic
110	Uniform load per unit length	Beam and Truss
111	Nonuniform load per unit length	Beam and Truss
112	Uniform load per unit area	Shell and Membrane
113	Nonuniform load per unit area	Shell and Membrane

A set of `IBODY` codes are available to improve the compatibility with MSC Nastran CID load option. These load types allow the user to give the three components of the surface traction. The magnitude is the vector magnitude of components given. These loads are available for all continuum elements. They are defined in Table 1-4.

Table 1-4 CID Load Types (Not Table Driven Input)

IBODY	Specify Traction on Edge or Face	User Subroutine
-10	1	No
-11	1	Yes
-12	2	No
-13	3	Yes
-14	3	No
-15	3	Yes
-16	4	No

Table 1-4 CID Load Types (Not Table Driven Input)

IBODY	Specify Traction on Edge or Face	User Subroutine
-17	4	Yes
-18	5	No
-19	5	Yes
-20	6	No
-21	6	Yes

The element edge and face number is given in Marc Volume A: Theory and User Information , Chapter 9 Boundary Conditions in the Face ID for Distributed Loads, Fluxes, Charge, Current, Source, Films, and Foundations section.

Table Driven Input

The second method to define distributed loads is available when the table driven input format is used. In this case, three separate numbers are specified which give the type of load, identify the edge or face where load is applied and whether or not a user subroutine is applied. The load types are the same for all element types where they are applicable and are summarized in Table 1-5

Table 1-5 Distributed Load Types (Table Driven Input)

Load Type	Meaning
1	Normal pressure
2	Shear load in first tangential direction
3	Shear load in second tangential direction
11	Wave loading (beams only)
21	General traction (CID loads)

Additionally all of the volumetric load types mentioned in Table 1-3 may be used.

For mechanical analysis using shell elements as mentioned earlier, a positive pressure implies a load that is in the opposite direction to the normal. A discussion on the normals is given in the following section. An identical load may be considered to be positive on the “top” surface or negative on the “bottom” surface. For heat transfer shells, this is not the case, a thermal flux on the top and bottom surface are different. Hence, for thermal analyses, the distributed flux (load) types are given by:

Table 1-6 Thermal Load Types

Load Type	Meaning
1	Flux applied to bottom of shell or continuum element
10	Flux applied to top of shell or continuum element

Shell Layer Convention

The shell elements in Marc are numerically integrated through the thickness. The number of layers can be defined for homogeneous shells through the SHELL SECT parameter. The default is eleven layers. For problems involving homogeneous materials, Simpson's rule is used to perform the integration. For inhomogeneous materials, the number of layers is defined through the COMPOSITE model definition option. In such cases, the trapezoidal rule is used for numerical integration through the thickness.

The layer number convention is such that layer one lies on the side of the positive normal to the shell, and the last layer is on the side of the negative normal. The normal to the element is based upon both the coordinates of the nodal positions and upon the connectivity of the element. The definition of the normal direction can be defined for five different groups of elements.

Beams in Plane — Elements 16 and 45

For element 16, the s-direction is defined in the COORDINATES option; for element 45, the s-direction points from node 1 to node 3. The normal direction is obtained by a rotation of 90° from the direction of increasing s in the x-y plane.

Axisymmetric Shells — Elements 1, 15, 87, 88, 89, and 90

For element 1 or 88, the s-direction points from node 1 to node 2. For element 15, the s-direction is defined in the COORDINATES option. For elements 87, 89, and 90, the s-direction points from node 1 to node 3. The normal direction is obtained by a rotation of 90° from the direction of increasing s in the z-r plane.

Curvilinear Coordinate Shell — Elements 4, 8, and 24

The normal to the surface $\underset{\sim}{n}$ is $\underset{\sim}{a}_1 \times \underset{\sim}{a}_2$, where $\underset{\sim}{a}_1$ is a base vector tangent to the positive θ^1 line and $\underset{\sim}{a}_2$ is a base vector tangent to the positive θ^2 line.

Triangular Shell — Element 49 and 138

A set of basis vectors tangent to the surface is created first. The first V_1 is in the plane of the three nodes from node 1 to node 2. The second V_2 lies in the plane, perpendicular to V_1. The normal n is then formed as $V_1 \times V_2$.

Shell Elements 22, 72, 75, 85, 86, 139, and 140

A set of base vectors tangent to the surface is first created. The first $\underset{\sim}{t}_1$ is tangent to the first isoparametric coordinate direction. The second $\underset{\sim}{t}_2$ is tangent to the second isoparametric coordinate direction. In the simple case of a rectangular element, t_1 would be in the direction from node 1 to node 2 and t_2 would be in the direction from node 2 to node 3. In the nontrivial (nonrectangular) case, a new set of vectors $\underline{V}_1$ and $\underline{V}_2$ would be created which are an orthogonal projection of t_1 and t_2. The normal is then formed as $\underline{n} = \underline{V}_1 \times \underline{V}_2$. Note that the vector $\underline{m} = \underline{t}_1 \times \underline{t}_2$ would be in the same general direction as $\underline{n}$. That is, $\underline{n} \cdot \underline{m} > 0$.

Shell Elements 50, 85, 86, 87, and 88

For heat transfer shell elements 50, 85, 86, 87, and 88, multiple degrees of freedom are used to capture the temperature variation through the thickness directions. The first degree of freedom is a temperature on the top surface; that is, the positive normal side. The second degree of freedom is on the bottom surface; that is, the negative normal side. If a quadratic variation is selected, the third degree of freedom represents the temperature at the midsurface.

Transverse Shear for Thick Beams and Shells

Conventional finite element implementation of Mindlin shell theory results in the transverse shear distribution being constant through the thickness of the element. An extension has been made for the thick beam type 45 and the thick shells 22, 75, and 140 such that a "parabolic" distribution of the transverse shear stress is obtained. In subsequent versions, a more "parabolic" distribution of transverse shear can be used. It is based upon a strength of materials beam equilibrium approach. For beam 45, the distribution is exact and you no longer need to correct your Poisson's ratio for the shear factor. For thick shells 22, 75, and 140, the new formulation is approximate since it is derived by assuming that the stresses in two perpendicular directions are uncoupled. To activate the parabolic shear distribution and calculation of interlaminar shear, include the TSHEAR parameter.

Beam Elements

When using beam elements, it is necessary to define four attributes of each element. The four attributes are defined in the following sections and summarized in Tables 1-7 and 1-8.

Table 1-7 2-D Beams

Element Type	Length	Beam Orientation	Cross-section Definition	Cross-section Orientation
5	$\lvert x_2 - x_1 \rvert$	1 to 2	Rectangular via GEOMETRY	Global z
16	User-defined s	Increasing s	Rectangular via GEOMETRY	Global z
45	$\lvert x_3 - x_1 \rvert$	1 to 3	Rectangular via GEOMETRY	Global z

Table 1-8 3-D Beams

Element Type	Length	Beam Orientation	Cross-section Definition	Cross-section Orientation
13	User-defined s	Increasing s	Open section via BEAM SECT	via COORDINATES allows twist
14	$\lvert x_2 - x_1 \rvert$	1 to 2	Closed section circular default or BEAM SECT	GEOMETRY or COORDINATES
25	$\lvert x_2 - x_1 \rvert$	1 to 2	Closed section circular default or BEAM SECT	GEOMETRY or COORDINATES
31	Bending radius or $\lvert x_2 - x_1 \rvert$	1 to 2	Circular Pipe or BEAM SECT	GEOMETRY
52	$\lvert x_2 - x_1 \rvert$	1 to 2	A, I_{xx}, I_{yy} on GEOMETRY, BEAM SECT, or arbitrary solid section on BEAM SECT	GEOMETRY or COORDINATES
76	$\lvert x_3 - x_1 \rvert$	1 to 3	Closed section circular default or BEAM SECT	GEOMETRY or COORDINATES
77	$\lvert x_3 - x_1 \rvert$	1 to 3	Open section via BEAM SECT	GEOMETRY or COORDINATES
78	$\lvert x_2 - x_1 \rvert$	1 to 2	Closed section circular default or BEAM SECT	GEOMETRY or COORDINATES
79	$\lvert x_2 - x_1 \rvert$	1 to 2	Open section via BEAM SECT	GEOMETRY or COORDINATES
98	$\lvert x_2 - x_1 \rvert$	1 to 2	A, I_{xx}, I_{yy} on GEOMETRY or BEAM SECT, or arbitrary solid section on BEAM SECT	GEOMETRY or COORDINATES

Note: Element 31 is always an elastic element; elements 52 and 98 are elastic elements if they do not use numerical cross-section integration.

Length of Element

The length of a beam element is generally the distance between the last node and the first node of the element. The exceptions are for the curved beams 13, 16, and 31. For element types 13 and 16, you provide s as the length of the beam. If element 13 is used as a pipe bend, the length will depend on the bending radius.

Orientation of Beam Axis

The orientation of the beam (local z-axis) is generally from the first node to the last node. The exceptions are for the curved beams 13 and 16. For these elements, the axis is in the direction of increasing s (see Length of Element).

Cross-section Properties

The cross-section properties fall into five different groups.

1. For two-dimensional beams (elements 5, 16, and 95), only rectangular beams are possible. The height and thickness are given through the GEOMETRY option. The stress strain law is integrated through the height of the beam.
2. For beam elements 52 or 98 used in their elastic context, the area and moment of inertias can be specified directly through the GEOMETRY option. For the elastic pipe-bend element 31, the radius and thickness are defined in the GEOMETRY option. The BEAM SECT option can also be used to define the properties for these elements.
3. For closed-section elements (14, 25, 76, and 78), the default cross section is a circular cross section (see Figure 1-2). When using the default circular section, 16 points are used to integrate the material behavior through the cross section. General closed section elements can be defined using the BEAM SECT option.
4. For open-section beam elements (13, 77, and 79), the BEAM SECT must be used to define the cross section.

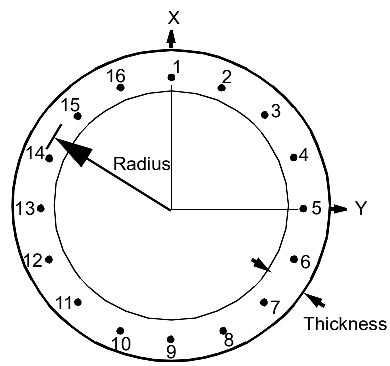


Figure 1-2 Default Cross Section

5. For arbitrary solid-section beam element (for example, 52 and 98 employing numerical cross-section integration), the BEAM SECT option must be used to define the cross section.

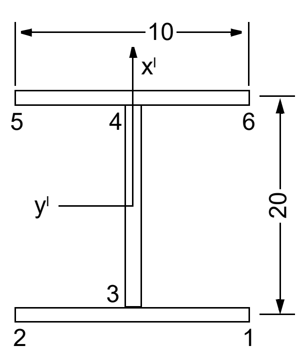
Definition of the Section

In any problem, any number of different beam sections can be included. Each section is defined by data blocks following the BEAM SECT parameter of the Marc input given in Marc Volume C: Program Input . The sections are numbered 1, 2, 3, etc. in the order they are input. Then, a particular section is obtained for an element by setting `EGEOM2` (GEOMETRY option, **2nd** data block, second field) to the floating point value of the section number; that is, 1, 2, or 3. For elements 14, 25, 76, or 78 if `EGEOM1` is nonzero, the default circular section is assumed. Other thin-walled closed section shapes can be defined in the BEAM SECT parameter. Thin-walled open section beams must always use a section defined in the BEAM SECT parameter. The thin-walled open section is defined using input data as shown in Figure 1-3. For elements 52 or 98, the section is assumed to be solid. Solid sections can be entered in two ways:

1. Enter the integrated section properties, like area and moments of inertia.
2. Define a standard section specifying its typical dimensions or defining an arbitrary solid section.

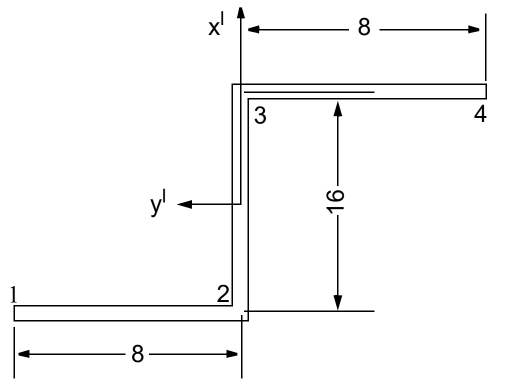
The first way can define a section without making a BEAM SECT definition when `EGEOM1` is nonzero. The second way can only define a section by making a BEAM SECT definition for it.

Section 1



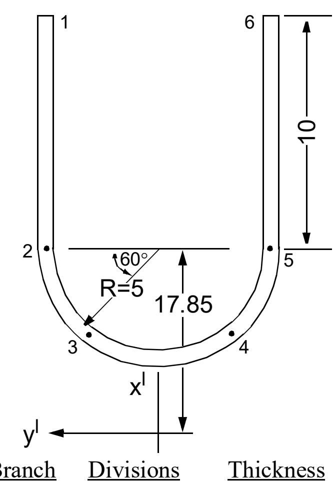
Branch	Divisions	Thickness
1-2	8	1.0
2-3	4	0.0
3-4	8	0.3
4-5	4	0.0
5-6	8	1.0

Section 2



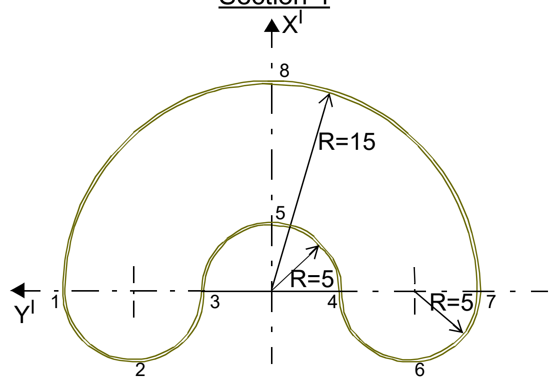
Branch	Divisions	Thickness
1-2	10	1.0
2-3	10	1.0
3-4	10	1.0

Section 3



Branch	Divisions	Thickness
1-2	8	0.5
2-3	4	0.5
3-4	8	0.5
4-5	4	0.5
5-6	8	0.5

Section 4



Branch	Divisions	Thickness
1-2	4	0.2
2-3	4	0.2
3-4	4	0.0
4-5	4	0.2
5-3	4	0.2
3-4	4	0.0
4-6	4	0.2
6-7	4	0.2
7-8	6	0.2
8-1	6	0.2

Figure 1-3 Section Definition Examples for Thin-Walled Sections

Standard (Default) Circular Section

For elements 14, 25, 76, and 78, the default cross section is that of a circular pipe. The positions of the 16 numerical integration points are shown in Figure 1-2. The contributions of the 16 points to the section quantities are obtained by numerical integration using Simpson's rule.

The rules and conventions for defining a thin-walled section are as follows:

1. The section is defined in an $x^1 - y^1$ coordinate system, with x^1 the first director at a point of the beam (see Figure 1-4).

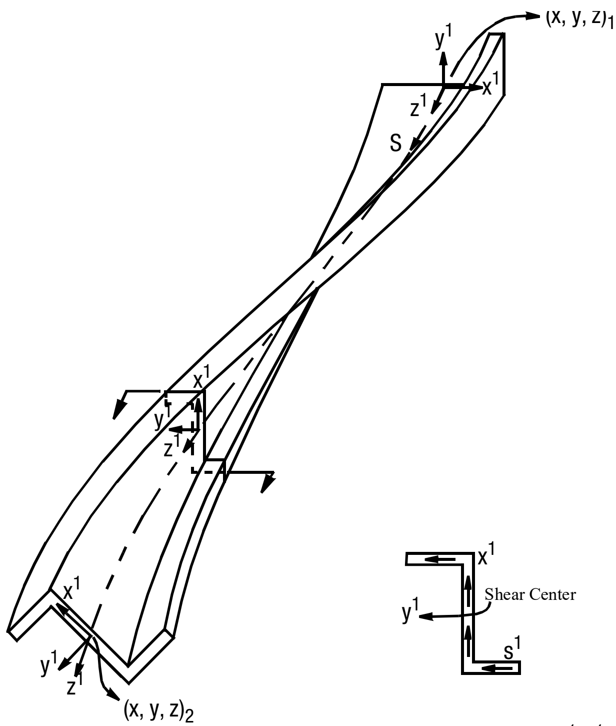


Figure 1-4 Beam Element Type 13 Including Twist, (x^1, y^1, z^1)

2. The section is input as a series of branches. Each branch can have a different geometry, but the branches must form a complete transverse of the section in the input sequence. Thus, the end point of a branch is always the start of the next branch. It is legitimate (in fact, it is often necessary) for the traverse of the section to double back on itself. This is achieved by specifying a branch with zero thickness.
3. Each branch is divided (by you) into segments. The stress points of the section, that is, the points used for numerical integration of section stiffness and also for output of stress, will be the segment division points. Then end points of any branch will always be stress points, and *there must always be an even number of divisions* (nonzero) in any branch. A maximum of 99stress points (98 divisions) can be used in a complete section, not counting branches of zero thickness.

4. Within a branch, the thickness varies linearly between the values given for thickness at the end points of the branch. The thickness can be discontinuous between branches. If the thickness at the end of the branch is given as an exact zero, the branch is assumed to be of constant thickness equal to the thickness given at the beginning of the branch.
5. The shape of any branch is interpolated as a cubic based on the values of x^1 and y^1 and their directions with respect to distance along the branch, which are input at the two ends of the branch. If both $\frac{dx^l}{ds}$ and $\frac{dy^l}{ds}$ are given as exact zeros at both ends of the branch, the branch is assumed to be straight as a default condition. Notice that x^1 and y^1 for the beginning of the branch are given only for the first branch of a section, since the beginning point of any other branch must be the same as the end point of the previous branch. Notice also that the section can have a discontinuous slope at the branch ends.
6. Any stress points separated by a distance less than t/10, where t is a thickness at one of these points, is merged into one point.

Example

As an example of defining beam sections, the I-section shown in Figure 1-3 can be defined by data shown in Table 1-9. The corresponding output for the I-Section is shown in Tables 1-10 and 1-11. Marc provides you with the location of each stress point on the section, the thickness at that point, the weighting associated with each point (for numerical integration of section stiffness) and the warping function (sectorial area) at each point. Notice the use of zero thickness branches in the traverse of the I-Section. Section 4 shows an example of multi-direction arcs; in which case, the zero thickness branches can be used to overcome the restriction on clockwise arcs.

Table 1-9 Beam Section Input Data (continued)for Section 1 in Figure 1-3

X_b	Y_b	dX_b/ds	dY_b/ds	X_e	Y_e	dX_e/ds	dY_e/ds
s	t_b	t_e					
beam sect							
I-section							
58	4 8	4 8					
-10.0	-5.0	0.0	1.0	-10.0	5.0	0.0	1.0
10.0	1.0	1.0					
		0.0	-1.0	-10.0	0.0	0.0	-1.0
5.0	0.0	0.0					
		1.0	0.0	10.0	0.0	1.0	0.0
20.0		0.5					
		0.0	1.0	10.0	5.0	0.0	1.0

X_b	Y_b	dX_b/ds	dY_b/ds	X_e	Y_e	dX_e/ds	dY_e/ds
s	t_b	t_e					
5.0	0.0	0.0					
		0.0	-1.0	10.0	-5.0	0.0	-1.0
10.0	1.0	1.0					
last							

Table 1-10 I-Section, Branch Definition Output Listing (continued)for Section 1 in Figure 1-3

I-Section											
Number of Branches 5			Intervals Per Branch 8 4 8 4 8								
Branch Definition											
Branch	x1	y1	x1p	y1p	x2	y2	x2p	y2p	p1	t1	t2
1	-10.000	-5.000	0.000	1.000	-10.000	5.000	0.000	1.000	10.000	1.000	1.000
2	-10.000	5.000	0.000	-1.000	-10.000	0.000	0.000	-1.000	5.000	0.000	0.000
3	-10.000	0.000	1.000	0.000	10.000	0.000	1.000	0.000	20.000	0.500	0.500
4	10.000	0.000	0.000	1.000	10.000	5.000	0.000	1.000	5.000	0.000	0.000
5	10.000	5.000	0.000	-1.000	10.000	-5.000	0.000	-1.000	10.000	1.000	1.000

Table 1-11 Beam Section Point Coordinates and Weights for Section 1 in Figure 1-3 Coordinates are with Respect to Shear Center

Section 1 (Open)					
Point Number	Coordinates in Section		Thickness	Warping Ftn.	Weight
1	-10.00000	-5.00000	1.00000	-50.00000	0.41667
2	-10.00000	-3.75000	1.00000	-37.50000	1.66667
3	-10.00000	-2.50000	1.00000	-25.00000	0.83333
4	-10.00000	-1.25000	1.00000	-12.50000	1.66667
5	-10.00000	0.00000	0.75000	0.00000	1.25000
6	-10.00000	1.25000	1.00000	12.50000	1.66667
7	-10.00000	2.50000	1.00000	25.00000	0.83333
8	-10.00000	3.75000	1.00000	37.50000	1.66667
9	-10.00000	5.00000	1.00000	50.00000	0.41667
10	-7.50000	0.00000	0.50000	0.00000	1.66667
11	-5.00000	0.00000	0.50000	0.00000	0.83333

Section 1 (Open)					
Point Number	Coordinates in Section		Thickness	Warping Ftn.	Weight
12	-2.50000	0.00000	0.50000	0.00000	1.66667
13	0.00000	0.00000	0.50000	0.00000	0.83333
14	2.50000	0.00000	0.50000	0.00000	1.66667
15	5.00000	0.00000	0.50000	0.00000	0.83333
16	7.50000	0.00000	0.50000	0.00000	1.66667
17	10.00000	0.00000	0.75000	0.00000	1.25000
18	10.00000	5.00000	1.00000	-50.00000	0.41667
19	10.00000	3.75000	1.00000	-37.50000	1.66667
20	10.00000	2.50000	1.00000	-25.00000	0.83333
21	10.00000	1.25000	1.00000	-12.50000	1.66667
22	10.00000	-1.25000	1.00000	12.50000	1.66667
23	10.00000	-2.50000	1.00000	25.00000	0.83333
24	10.00000	-3.75000	1.00000	37.50000	1.66667
25	10.00000	-5.00000	1.00000	50.00000	0.41667

Solid Sections

Solid sections can be defined by using one of the standard sections and specifying its typical dimensions or by using quadrilateral segments and specifying the coordinates of the four corner points of each quadrilateral segment in the section. The local axes of a standard section are always the symmetry axes, which also are the principal axes. The principal axes for each section are shown in Figure 1-5. For the more general sections, it is not required to enter the coordinates of the corner points of the quadrilateral segments with respect to the principal axes. The section is automatically realigned and you can make specifications how this realignment should be carried out. Furthermore you can make specifications about the orientation of the section in space. The details about these orientation methods are described in the sections Cross-section Orientation and Location of the Local Cross-section Axis of this volume.

Figure 1-5 shows the standard solid sections, the dimensions needed to specify them, and the orientation of their local axes. If the dimension **b** is omitted for an elliptical section, the section will be circular. If the dimension **b** is omitted for a rectangular section, the section will be square. If the dimension **c** is omitted for a trapezoidal or a hexagonal section, the section will degenerate to a triangle or a diamond. For the elliptical section, the center point is the first integration point. The second is located on the negative *y*-axis and the points are numbered radially outward and then counterclockwise. For the rectangular, trapezoidal and hexagonal sections the first integration point is nearest to the lower left corner and the last integration point is nearest to the upper right corner. They are numbered from left to right and then from bottom to top. The default integration scheme for all standard solid sections uses 25 integration points.

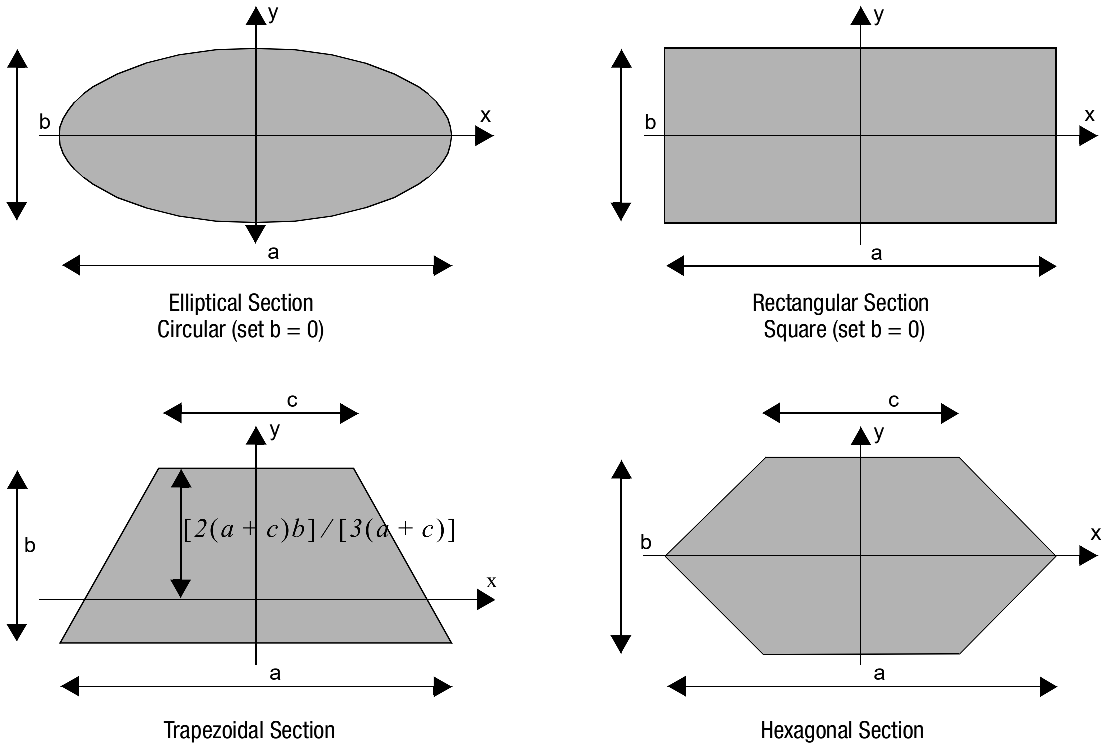


Figure 1-5 Standard Solid Section Types with Typical Dimensions

The rules and conventions for defining a general solid section are listed below:

1. Each quadrilateral segment is defined by entering the coordinates of the four corner points in the local xy-plane. The corner points must be entered in a counterclockwise sense.
2. The quadrilateral segments must describe a simply connected region; i.e., there must be no holes in the section, and the section cannot be composed of unconnected "islands" and should not be connected through a single point.
3. The quadrilateral segments do not have to match at the corners. Internally, the edges must, at least, partly match.
4. The segments must not overlap each other (i.e., the sum of the areas of each segment must equal the total area of the segment).
5. Each section can define its own integration scheme and this scheme applies to all segments in the section.
6. A section cannot have more than 100 integration points in total. This limits each section to, at most, 100 segments that, by necessity, will use single point integration. If less than 100 segments are present in a section, they may use higher order integration schemes, as long as the total number of points in the section does not exceed 100.
7. Integration points on matching edges are not merged, should they coincide.

This method allows you to define an arbitrary solid section in a versatile way, but you must make sure that the conditions of simply connectedness and nonoverlap are met. The assumptions underlying the solid sections are not accurate enough to model sections that are thin-walled in nature. It is not recommended to use them as an alternative to thin-walled sections.

Solid sections do not account for any warping of the section and, therefore, overestimate the torsion stiffness as it is predicted by the Saint Venant theory of torsion.

The first integration point in each quadrilateral segment is nearest to its first corner point and the last integration point is nearest to its third corner point. They are numbered going from first to second corner point and then from second to third corner point. The segments are numbered consecutively in the order in which they have been entered and the first integration point number in a new segment simply continues from the highest number in the previous segment. There is no special order requirement for the quadrilateral segments. They must only meet the previously outlined geometric requirements. For each section, the program provides the location of the integration points with respect to the principal axes and the integration weight factors of each point. Furthermore, it computes the coordinates of the center of gravity and the principal directions with respect to the input coordinate system. It also computes the area and the principal second moments of area of the cross section. For each solid section, all of this information is written to the output file.

All solid sections can be used in a pre-integrated fashion. In that case, the area A and the principal moments of area I_{xx} and I_{yy} are calculated by numerical integration. The torsional stiffness is always computed from the polar moment of inertia $J=I_{xx}+I_{yy}$. The shear areas are set equal to the section area A. The stiffness behavior may be altered by employing any of the stiffness factors. In a pre-integrated section, no layer information is computed and is, therefore, not available for output. Pre-integrated sections can only use elastic material behavior and cannot account for any inelasticity (e.g., plasticity). For pre-integrated sections there is no limit on the number of segments to define the section.

Figure 1-6 shows a general solid section built up from three quadrilateral segments using a 2 x 2 Gauss integration scheme. It also shows the numbering conventions adopted in a quadrilateral segment and its mapping onto the parametric space. For each segment in the example, the first corner point is the lower-left corner and the corner points have been entered in a counterclockwise sense. The segment numbers and the resulting numbering of the integration points are shown in the figure. Gauss integration schemes do not provide integration points in the corner points of the section. If this is desired, you can choose other integration schemes like Simpson or Newton-Cotes schemes. In general, a 2 x 2 Gauss or a 3 x 3 Simpson scheme in each segment suffices to guarantee exact section integration of linear elastic behavior.

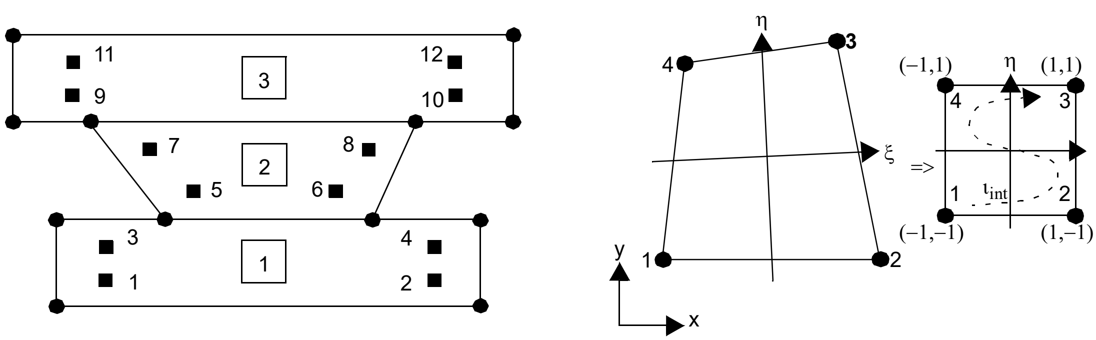


Figure 1-6 Beam Section Definition for Solid Section plus Numbering Conventions Adopted

Cross-section Orientation

The cross-section axis orientation is important both in defining the beam section and in interpreting the results. It is also used to define a local element coordinate system used for the PIN CODE option. The element moments given in the output and on the post file are with respect to the local axes defined here. The definition falls into the following three groups:

1. For two-dimensional beam elements (5, 16, and 45), there is no choice. The local x-axis is in the global z-direction.
2. For element type 13, the direction cosines of the local axis is given in the COORDINATES option. Different values can be prescribed at both nodes of the beam. This allows you to prescribe a twist to the element.
3. For all other beam elements, the local axis can be given in either of two ways:
   a. The coordinates of a point (point 3) is chosen to define the local x-z plane (see Figure 1-7).
   b. The direction cosines of a vector in the x-z plane is defined. The local x-axis is then the component of this vector which is perpendicular to the z-axis (see Figure 1-8).

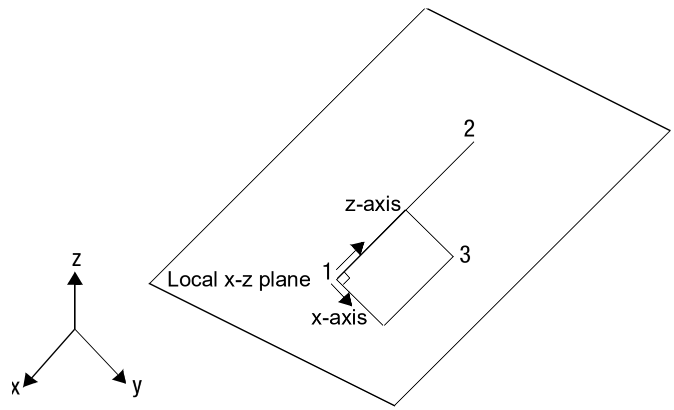


Figure 1-7 Local Axis Defined by Entering a Point

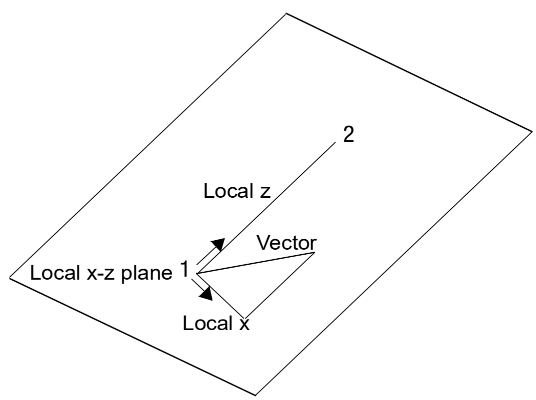


Figure 1-8 Local Axis Defined by Entering a Vector

Location of the Local Cross-section Axis

The origin of the local coordinate system is at the nodal point. The thin-walled beam section is defined with respect to this point. If the origin is the shear center of the cross section, the element behaves in the classical manner; a bending moment parallel to a principle axis of the cross section causes no twisting.

In the output, the location of the integration points is given with respect to the shear center (see Figure 1-9).

For the default circular cross section, the local origin is at the center, which is also the shear center.

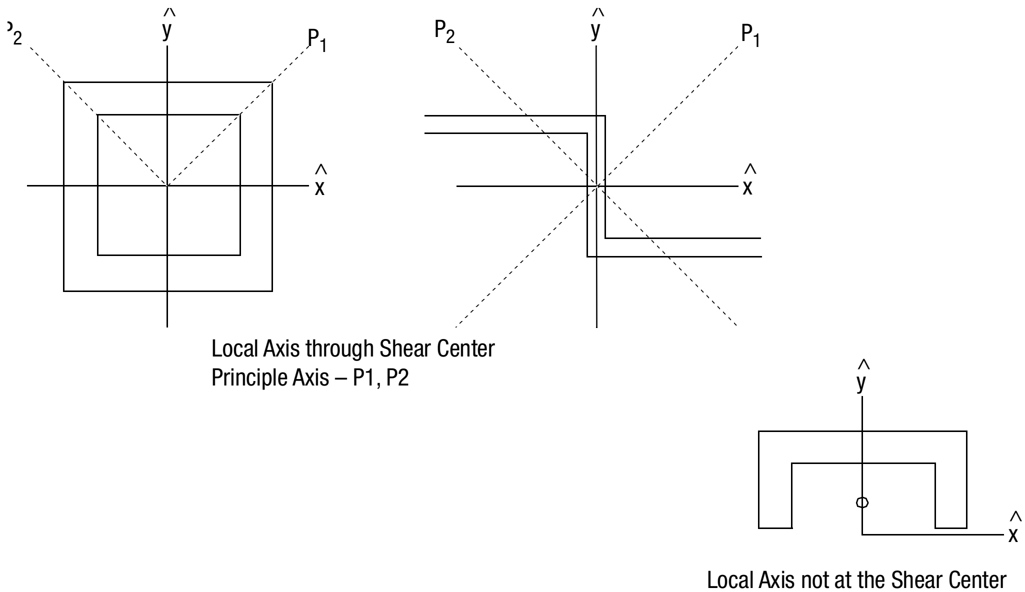


Figure 1-9 Location of Local Axis

The local axes of the solid beam section are always the principal axes and, thus the node is always at its center of gravity. The generalized force output is always with respect to these principal axes. For the sections that have their properties entered directly, the local x-direction is the one associated with I_{xx}, the first moment of area in the input. For standard solid section types using numerical integration the local x-directions are shown in Figure 1-5. For the general solid sections, built from quadrilateral segments, there are three ways to align a local direction in the plane of the section with the plane spanned by the beam axis and the global orientation vector discussed in the previous section, Cross-section Orientation . If the section has not been entered with respect to the principal axes, it is automatically realigned and the cross-section integration points are computed with respect to these principal axes. By default, the x-axis of the input coordinate system of the section (i.e., the system with respect to which the segment corner coordinates are specified) lies in the plane spanned by the beam axis and the global orientation vector. The formulation of the beam behavior however, is with respect to the principal axes. If the x-direction of the input system is not the same as the first principal direction, the global orientation vector is rotated about the beam axis such that after rotation it lies in the plane spanned by the beam axis and the first principal direction. The second way specifies that the first principal direction lies in the plane spanned by the beam axis and the global orientation vector. The third way specifies this condition for the second principal direction. Here the first and second principal directions are defined as the directions associated with the largest and smallest principal moment of area. A user-specified point in the plane of the section (by default a point to the immediate right of the center of gravity) is projected onto the desired principal axis and the direction from the center of gravity to this projection is taken as the positive direction along the axis. If the two principal moments of area are equal, the line from the center of gravity to the user specified or default point defines the positive principal direction.

As a first example, the section in Figure 1-10 is considered, which consists of two rectangular segments.

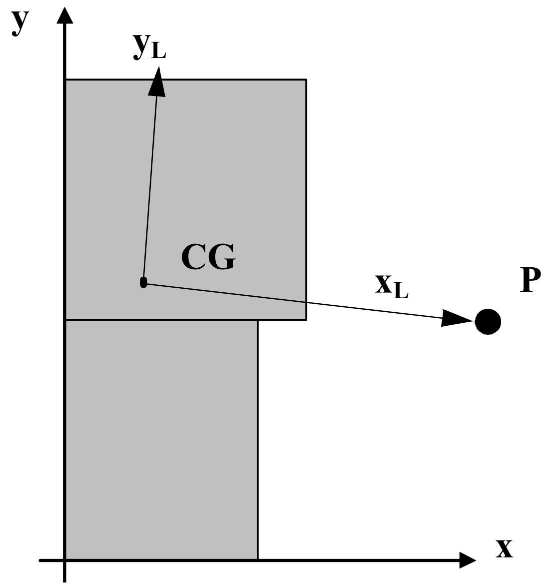


Figure 1-10 Solid Section Defined by Two Rectangular Segments

The input for the section is shown below. It defines one section with two quadrilateral segments using a 3x3 Simpson integration. The lower segment has size 0.8x1 and the upper segment has size 1x1. The first principal axis defines the local x-axis and the vector from the center of gravity (CG) to the projection of point P (2.0,1.0) onto this principal axis defines its positive direction. The coordinates of corner points of the segments and the coordinates of the point P have been entered with respect to the x,y-system in Figure 1-10.

```
geom1
-2,3,3
0.0,0.0,0.8,0.0,0.8,1.0,0.0,1.0
0.0,1.0,1.0,1.0,1.0,2.0,0.0,2.0,
,1,2.0,1.0
```

The output for this section is shown below. The center of gravity (CG) and the local directions (x_L,y_L) are also shown in Figure 1-10. In the output they are printed with respect to the x,y-system that was used to enter the section data. The coordinates of the integration points (layers) in the section are with respect to the local x_L,y_L-axes. The orientation of the section in space is such that the x_L-direction in Figure 1-10 lies in the plane spanned by the beam axis and the global orientation vector entered in the GEOMETRY model definition input.

```
cross-section      1, name: "geom1,"

element type 52 or 98, properties for quadrilateral solid cross-section         1
 area     0.18000E+01, normal  stiffness factor     0.10000E+01
 ixx      0.59866E+00, bending stiffness factor     0.10000E+01
 iyy      0.12623E+00, bending stiffness factor     0.10000E+01
 j        0.72489E+00, torsion stiffness factor     0.10000E+01
 ashx     0.18000E+01, shear   stiffness factor     0.10000E+01
 ashy     0.18000E+01, shear   stiffness factor     0.10000E+01

center of gravity (CG)  w.r.t. the input coordinate system: 0.45556E+00  0.10556E+01
1st principal direction w.r.t. the input coordinate system: 0.99553E+00 -0.94498E-01
2nd principal direction w.r.t. the input coordinate system: 0.94498E-01  0.99553E+00
local axes are the principal axes
number of segments in this section:     2

numerical integration scheme in 1st direction of a segment:  3-point Simpson
numerical integration scheme in 2nd direction of a segment:  3-point Simpson
numerical integration data for this section (coordinates are w.r.t. the principal axes)
 layer   x-coordinate   y-coordinate   weight factor
 =====   ============   ============   =============
     1   -0.35377E+00   -0.10939E+01    0.22222E-01
     2    0.44441E-01   -0.10561E+01    0.88889E-01
     3    0.44265E+00   -0.10183E+01    0.22222E-01
     4   -0.40102E+00   -0.59612E+00    0.88889E-01
```

As a second example the same section is considered, but the default method of alignment is used. The input for this section is shown below. The only difference with the previous input is that the second field in the last line has been left blank. Now the orientation of the section in space is such that the x-direction in Figure 1-10 lies in the plane spanned by the beam axis and the global orientation vector entered in the GEOMETRY model definition input.

```
geom1
-2,3,3
0.0,0.0,0.8,0.0,0.8,1.0,0.0,1.0
0.0,1.0,1.0,1.0,1.0,2.0,0.0,2.0,
,,2.0,1.0
```

The output for this section is shown below. Note that the output for the coordinates of the section integration points has not changed. The local direction in the geometry input is rotated over the reported angle to make sure the section retains the desired orientation in space while being formulated with respect to the principal directions.

```
cross-section      1, name: "geom1,"

element type 52 or 98, properties for quadrilateral solid cross-section         1
 area     0.18000E+01, normal  stiffness factor     0.10000E+01
 ixx      0.59866E+00, bending stiffness factor     0.10000E+01
 iyy      0.12623E+00, bending stiffness factor     0.10000E+01
 j        0.72489E+00, torsion stiffness factor     0.10000E+01
 ashx     0.18000E+01, shear   stiffness factor     0.10000E+01
 ashy     0.18000E+01, shear   stiffness factor     0.10000E+01

center of gravity (CG)  w.r.t. the input coordinate system: 0.45556E+00  0.10556E+01
1st principal direction w.r.t. the input coordinate system: 0.99553E+00 -0.94498E-01
2nd principal direction w.r.t. the input coordinate system: 0.94498E-01  0.99553E+00
local axes are the CG-shifted user input axes
alignment with principal axes is forced, section output is w.r.t. principal axes
local direction vector on geometry input is rotated over   -5.42 degrees about beam axis
number of segments in this section:     2

numerical integration scheme in 1st direction of a segment:  3-point Simpson
numerical integration scheme in 2nd direction of a segment:  3-point Simpson
numerical integration data for this section (coordinates are w.r.t. the principal axes)
 layer   x-coordinate   y-coordinate   weight factor
 =====   ============   ============   =============
     1   -0.35377E+00   -0.10939E+01    0.22222E-01
     2    0.44441E-01   -0.10561E+01    0.88889E-01
     3    0.44265E+00   -0.10183E+01    0.22222E-01
     4   -0.40102E+00   -0.59612E+00    0.88889E-01
```

Element Characteristics

In CHAPTER 3, the element description appears in the sequence in which they are developed in Marc. However, it is easier to recognize the applicability of each element by grouping it in its own structural class. This is shown in Table 1-12. Table 1-13 lists the element library in the sequence that the elements were developed.

Each element type has unique characteristics governing its behavior. This includes the number of nodes, the number of direct and shear stress components, the number of integration points used for stiffness calculations, the number of degrees of freedom, and the number of coordinates. This information is summarized in Table 1-14. These values are also very useful when writing user subroutines (see Marc Volume D: User Subroutines and Special Routines).

The availability of the updated Lagrange procedure for different element types is given in Table 1-14. If an element cannot support the updated Lagrange method, the total Lagrange method is used for that element.

A summary of which elements are available for heat transfer, Joule Heating, Conrad gap, channels, electrostatics and magnetostatics is presented in Table 1-15.

Table 1-12 Structural Classification of Elements

Element Structural Type		Function	Remarks
Three-dimensional truss	9	Linear	2-node straight
	12	Linear	4-node straight gap and friction
	51	Analytic	2-node cable element
	64	Quadratic	3-node curved
	97	Linear	4-node straight double gap and friction
Two-dimensional beam column	5	Linear/cubic	2-node straight
	16	Cubic	2-node curved
	45	Cubic	3-node curved Timoshenko theory
Cbush	194	Linear	2-node two-dimensional
	195	Linear	2-node three-dimensional
Three-dimensional beam column	13	Cubic	2-node curved open section
	14	Linear/cubic	2-node straight closed section
	25	Cubic	2-node straight closed section
	31	Analytic	2-node elastic
	52	Linear/cubic	2-node straight
	76	Linear/cubic	2 + 1-node straight closed section; use with Element 72
	77	Linear/cubic	2 + 1-node straight open section; use with Element 72
	78	Linear/cubic	2-node straight closed section; use with Element 75
	79	Linear/cubic	2-node straight open section
	98	Linear	2-node straight with transverse shear
Axisymmetric shell	1	Linear/cubic	2-node straight
	15	Cubic	2-node curved
	89	Quadratic	3-node curved thick shell theory
	90	Quadratic	3-node curved with arbitrary loading; thick shell theory

Table 1-12 Structural Classification of Elements (continued)

Element Structural Type		Function	Remarks
Plane stress	3	Linear	4-node quadrilateral
	26	Quadratic	8-node quadrilateral
	53	Quadratic	8-node reduced integration quadrilateral
	114	Linear/Assumed strain	4-node quadrilateral, reduced integration, with hourglass control
	124	Quadratic	6-node triangle
	160	Linear	4-node quadrilateral with piezoelectric capability
	201	Linear	3-node triangle
Plane strain	6	Linear	3-node triangle
	11	Linear	4-node quadrilateral
	27	Quadratic	8-node quadrilateral
	54	Quadratic	8-node reduced integration quadrilateral
	91	Linear/special	6-node semi-infinite
	93	Quadratic/special	9-node semi-infinite
	115	Linear/Assumed strain	4-node quadrilateral, reduced integration, with hourglass control
	125	Quadratic	6-node triangle
	161	Linear	4-node quadrilateral with piezoelectric capability
	239	Piecewise Linear	3 + 1-node triangle with strain smoothing
Generalized plane strain	19	Linear	4 + 2-node quadrilateral
	29	Quadratic	8 + 2-node quadrilateral
	56	Quadratic	8 + 2-node reduced integration quadrilateral

Table 1-12 Structural Classification of Elements (continued)

Element Structural Type		Function	Remarks
Axisymmetric solid	2	Linear	3-node triangle
	10	Linear	4-node quadrilateral
	20	Linear	4-node quadrilateral with twist
	28	Quadratic	8-node quadrilateral
	55	Quadratic	8-node reduced integration quadrilateral
	62	Quadratic	8-node quadrilateral with arbitrary loading
	67	Quadratic	8-node quadrilateral with twist
	73	Quadratic	8-node reduced integration quadrilateral and arbitrary loading
	92	Linear/special	6-node semi-infinite
	94	Quadratic/special	9-node semi-infinite
	95	Linear	4-node quadrilateral with bending
	96	Quadratic	8-node quadrilateral with bending
	116	Linear/Assumed strain	4-node quadrilateral, reduced integration, with hourglass control
	126	Quadratic	6-node triangle
	162	Linear	4-node quadrilateral with piezoelectric capability
	240	Piecewise Linear	3 + 1-node triangle with strain smoothing
Membrane three-dimensional	18	Linear	4-node quadrilateral
	30	Quadratic	8-node quadrilateral
	158	Linear	3-node triangle
	200	Quadratic	6-node triangle
Doubly-curved thin shell	4	Cubic	4-node curved quadrilateral
	8	Fractional cubic	3-node curved triangle
	24	Cubic patch	4 + 4-node curved quadrilateral
	49	Linear	3 + 3-node curved triangle discrete Kirchhoff
	72	Linear	4 + 4-node twisted quadrilateral discrete Kirchhoff
	138	Linear	3-node triangle discrete Kirchhoff
	139	Linear	4-node twisted quadrilateral discrete Kirchhoff

Table 1-12 Structural Classification of Elements (continued)

Element Structural Type		Function	Remarks
Doubly-curved thick shell	22	Quadratic	8-node curved quadrilateral with reduced integration
	75	Linear	4-node twisted quadrilateral
	140	Linear	4-node twisted quadrilateral, reduced integration with hourglass control
Three-dimensional solid	7	Linear	8-node hexahedron
	21	Quadratic	20-node hexahedron
	57	Quadratic	20-node reduced integration hexahedron
	107	Linear/special	12-node semi-infinite
	108	Quadratic	27-node semi-infinite special
	117	Linear/Assumed strain	8-node hexahedron, reduced integration with hourglass control
	127	Quadratic	10-node tetrahedron
	134	Linear	4-node tetrahedron
	136	Linear	6-node pentahedral
	163	Linear	8-node hexahedron with piezoelectric capability
	164	Linear	4-node tetrahedron with piezoelectric capability
	202	Quadratic	15-node pentahedral
	216	Linear	5-node pyramid
	218	Quadratic	13-node pyramid
	241	Piecewise Linear	4 + 1-node tetrahedron with strain smoothing
	242	Quadratic	20-node hexahedron with piezoelectric capability
	243	Quadratic	10-node tetrahedron with piezoelectric capability
Solid shell	185	Linear/Assumed	8-node hexahedron
Incompressible plane strain	32	Quadratic	8-node quadrilateral
	58	Quadratic	8-node reduced integration quadrilateral
	80	Linear	4 + 1-node quadrilateral
	118	Linear/Assumed strain	4 + 1-node quadrilateral, reduced integration with hourglass control
	128	Quadratic	6-node triangle
	155	Linear	3 + 1-node triangle

Table 1-12 Structural Classification of Elements (continued)

Element Structural Type		Function	Remarks
Incompressible generalized plane strain	34	Quadratic	8 + 2-node quadrilateral
	60	Quadratic	8 + 2-node with reduced integration quadrilateral
	81	Linear	4 + 3-node quadrilateral
Incompressible axisymmetric	33	Quadratic	8-node quadrilateral
	59	Quadratic	8-node with reduced integration quadrilateral
	63	Quadratic	8-node quadrilateral with arbitrary loading
	66	Quadratic	8-node quadrilateral with twist
	74	Quadratic	8-node reduced integration quadrilateral with arbitrary loading
	82	Linear	4 + 1-node quadrilateral
	83	Linear	4 + 1-node quadrilateral with twist
	119	Linear/Assumed strain	4 + 1-node quadrilateral reduced integration, with hourglass control
	129	Quadratic	6-node triangle
	156	Linear	3 + 1-node triangle
Incompressible three-dimensional solid	35	Quadratic	20-node hexahedron
	61	Quadratic	20-node with reduced integration hexahedron
	84	Linear	8 + 1-node hexahedron Node 9
	120	Linear/Assumed strain	8 + 1-node hexahedron, reduced integration with hourglass control
	130	Quadratic	10-node tetrahedron
	157	Linear	4 + 1-node tetrahedron
Pipe bend	17	Cubic	2-nodes in-section; 1-node out-of-section
	31	Special	2-node elastic

Table 1-12 Structural Classification of Elements (continued)

Element Structural Type		Function	Remarks
Rebar elements	23	Quadratic	20-node hexahedron
	46	Quadratic	8-node quadrilateral plane strain
	47	Quadratic	8 + 2-node quadrilateral generalized plane strain
	48	Quadratic	8-node quadrilateral axisymmetric
	142	Quadratic	8-node axisymmetric with twist
	143	Linear	4-node plane strain
	144	Linear	4-node axisymmetric
	145	Linear	4-node axisymmetric with twist
	146	Linear	8-node hexahedron
	147	Linear	4-node membrane
	148	Quadratic	8-node membrane
	165	Linear	2-node plane strain membrane
	166	Linear	2-node axisymmetric membrane
	167	Linear	2-node axisymmetric membrane with twist
	168	Quadratic	3-node plain strain membrane
	169	Quadratic	3-node axisymmetric membrane
	170	Quadratic	3-node axisymmetric membrane with twist
Cavity surface elements	171	Linear	2-node straight
	172	Linear	2-node straight, axisymmetric
	173	Linear	3-node triangle
	174	Linear	4-node quadrilateral
Continuum composite elements	149	Linear	8-node hexahedron
	150	Quadratic	20-node hexahedron
	151	Linear	4-node plane strain quadrilateral
	152	Linear	4-node axisymmetric quadrilateral
	153	Quadratic	8-node plane strain quadrilateral
	154	Quadratic	8-node axisymmetric quadrilateral

Table 1-12 Structural Classification of Elements (continued)

Element Structural Type		Function	Remarks
Structural interface elements	186	Linear	4-node planar
	187	Quadratic	8-node planar
	188	Linear	8-node three-dimensional
	189	Quadratic	20-node three-dimensional
	190	Linear	4-node axisymmetric
	191	Quadratic	8-node axisymmetric
	192	Linear	6-node three-dimensional
	193	Quadratic	15-node three-dimensional
Three-dimensional shear panel	68	Linear	4-node quadrilateral
Heat conduction three-dimensional link	36	Linear	2-node straight
	65	Quadratic	3-node curved
Heat conduction planar	37	Linear	3-node triangle
	39	Linear	4-node quadrilateral
	41	Quadratic	8-node quadrilateral
	69	Quadratic	8-node reduced integration quadrilateral
	101	Linear/special	6-node semi-infinite
	103	Linear/special	9-node semi-infinite
	121	Linear	4-node quadrilateral, reduced integration, with hourglass control
	131	Quadratic	6-node triangular
Heat conduction axisymmetric	38	Linear	3-node triangle
	40	Linear	4-node quadrilateral
	42	Quadratic	8-node quadrilateral
	70	Quadratic	8-node reduced integration quadrilateral
	102	Linear/special	6-node semi-infinite
	104	Quadratic/special	9-node semi-infinite
	122	Linear	4-node quadrilateral, reduced integration, with hourglass control
	132	Quadratic	6-node triangular

Table 1-12 Structural Classification of Elements (continued)

Element Structural Type		Function	Remarks
Heat conduction solids	43	Linear	8-node hexahedron
	44	Quadratic	20-node hexahedron
	71	Quadratic	20-node reduced integration hexahedron
	105	Linear/special	12-node semi-infinite
	106	Quadratic/special	27-node semi-infinite
	123	Linear	8-node hexahedron, reduced integration, with hourglass control
	133	Quadratic	10-node tetrahedron
	135	Linear	4-node tetrahedron
	137	Linear	6-node pentahedral
	203	Quadratic	15-node pentrahedral
	217	Linear	5-node pyramid
	219	Quadratic	13-node pyramid
Heat membrane	196	Linear	3-node triangle
	197	Quadratic	6-node triangle
	198	Linear	4-node quadrilateral
	199	Quadratic	8-node quadrilateral
Heat conduction shell	50	Linear	3-node triangle
	85	Linear	4-node quadrilateral
	86	Quadratic	8-node quadrilateral
Heat conduction axisymmetric shell	87	Quadratic	3-node curved
	88	Linear	2-node straight
Heat conduction continuum composite elements	175	Linear	8-node hexahedron
	176	Quadratic	20-node hexahedron
	177	Linear	4-node planar strain quadrilateral
	178	Linear	4-node axisymmetric quadrilateral
	179	Quadratic	8-node planar strain quadrilateral
	180	Quadratic	8-node axisymmetric quadrilateral

Table 1-12 Structural Classification of Elements (continued)

Element Structural Type		Function	Remarks
Heat transfer interface elements	220	Linear	4-node planar
	221	Quadratic	8-node planar
	222	Linear	8-node three-dimensional
	223	Quadratic	20-node three-dimensional
	224	Linear	4-node axisymmetric
	225	Quadratic	8-node axisymmetric
	226	Linear	6-node three-dimensional
	227	Quadratic	15-node three-dimensional
Magnetostatic three-dimensional solids	109	Linear	8-node hexahedron
	110	Linear/special	12-node semi-infinite
	181	Linear	4-node tetrahedron
	182	Quadratic	10-node tetrahedron
	183	Linear	2-node straight
	204	Linear	6-node pentahedral
	205	Quadratic	15-node pentrahedral
	206	Quadratic	20-node brick
Magnetodynamic Planar	111	Linear	4-node quadrilateral
	228	Linear	3-node quadrilateral
	231	Quadratic	6-node triangular
	234	Quadratic	8-node quadrilateral
Magnetodynamic Axisymmetric	112	Linear	4-node quadrilateral
	229	Linear	3-node triangular
	232	Quadratic	6-node triangular
	235	Quadratic	8-node quadrilateral
Magnetodynamic Solid	113	Linear	8-node hexahedron
	230	Linear	4-node tetrahedron
	233	Quadratic	10-node tetrahedron
	236	Quadratic	20-node hexahedron
	237	Linear	6-node pentahedral
	238	Quadratic	15-node pentahedral

Table 1-13 Element Library

Element	Description	Code
1	Two-node Axisymmetric Shell Element	(1)
2	Axisymmetric Triangular Ring Element	(2)
3	Two-dimensional (Plane Stress) Four-node, Isoparametric Quadrilateral	(3)
4	Curved Quadrilateral Thin-shell Element	(4)
5	Beam-column	(5)
6	Two-dimensional Plane Strain, Constant Stress Triangle	(6)
7	Eight-node Isoparametric Three-dimensional Hexahedron	(7)
8	Three-node, Triangular Arbitrary Shell	(8)
9	Three-dimensional Truss Element	(9)
10	Axisymmetric Quadrilateral Element (Isoparametric)	(10)
11	Plane Strain Quadrilateral Element (Isoparametric)	(11)
12	Friction and Gap Element	(12)
13	Open-section Beam	(13)
14	Closed-section Beam	(14)
15	Isoparametric, Two-node Axisymmetric Shell	(15)
16	Isoparametric, Two-node Curved Beam	(16)
17	Pipe-bend Element	(17)
18	Four-node, Isoparametric Membrane	(18)
19	Generalized Plane Strain Quadrilateral	(19)
20	Axisymmetric Torsional Quadrilateral	(20)
21	Three-dimensional, 20-node brick	(21)
22	Curved Quadrilateral Thick-shell Element	(22)
23	Three-dimensional, 20-node Rebar Element	(23)
24	Curved Quadrilateral Shell Element	(24)
25	Closed-section Beam in Three Dimensions	(25)
26	Plane Stress, Eight-node Distorted Quadrilateral	(26)
27	Plane Strain, Eight-node Distorted Quadrilateral	(27)
28	Axisymmetric, Eight-node Distorted Quadrilateral	(28)
29	Generalized Plane Strain, Distorted Quadrilateral	(29)
30	Membrane, Eight-node Distorted Quadrilateral	(30)
31	Elastic Curved Pipe (Elbow)/Straight Beam Element	(31)

Table 1-13 Element Library (continued)

Element	Description	Code
32	Plane Strain, Eight-node Distorted Quadrilateral, Herrmann or Mooney Material Formulation	(32)
33	Axisymmetric, Eight-node Distorted Quadrilateral, Herrmann or Mooney Material Formulation	(33)
34	Generalized Plane Strain, Eight-node Distorted Quadrilateral, Herrmann or Mooney Material Formulation	(34)
35	Three-dimensional, 20-node Brick, Herrmann or Mooney Material Formulation	(35)
36	Heat Transfer Element (three-dimensional link)	(36)
37	Heat Transfer Element (arbitrary planar triangle)	(37)
38	Heat Transfer Element (arbitrary axisymmetric triangle)	(38)
39	Heat Transfer Element (planar bilinear quadrilateral)	(39)
40	Heat Transfer Element (axisymmetric bilinear quadrilateral)	(40)
41	Heat Transfer Element (eight-node planar biquadratic quadrilateral)	(41)
42	Heat Transfer Element (eight-node axisymmetric biquadratic quadrilateral)	(42)
43	Heat Transfer Element (three-dimensional eight-node brick)	(43)
44	Heat Transfer Element (three-dimensional 20-node brick)	(44)
45	Curved Timoshenko Beam Element in a Plane	(45)
46	Plane Strain Rebar Element	(46)
47	Generalized Plane Strain Rebar Element	(47)
48	Axisymmetric Rebar Element	(48)
49	Triangular Shell Element	(49)
50	Triangular Heat Transfer Shell Element	(50)
51	Cable Element, Two-node	(51)
52	Elastic Beam	(52)
53	Plane Stress, Eight-node Quadrilateral with Reduced Integration	(53)
54	Plane Strain, Eight-node Distorted Quadrilateral with Reduced Integration	(54)
55	Axisymmetric, Eight-node Distorted Quadrilateral with Reduced Integration	(55)
56	Generalized Plane Strain, Ten-node Distorted Quadrilateral with Reduced Integration	(56)
57	Three-dimensional, 20-node Brick with Reduced Integration	(57)
58	Plane Strain, Eight-node Distorted Quadrilateral for Incompressible Behavior with Reduced Integration	(58)
59	Axisymmetric, Eight-node Distorted Quadrilateral for Incompressible Behavior with Reduced Integration	(59)

Table 1-13 Element Library (continued)

Element	Description	Code
60	Generalized Plane Strain, Ten-node Distorted Quadrilateral for Incompressible Behavior with Reduced Integration	(60)
61	Three-dimensional, 20-node Brick for Incompressible Behavior with Reduced Integration	(61)
62	Axisymmetric, EIght-node Quadrilateral for Arbitrary Loading	(62)
63	Axisymmetric, Eight-node Quadrilateral for Arbitrary Loading, Herrmann Formulation	(63)
64	Isoparametric, Three-node Truss Element	(64)
65	Heat Transfer Element, Three-node Link	(65)
66	Eight-node Axisymmetric with Twist, Herrmann Formulation	(66)
67	Eight-node Axisymmetric with Twist	(67)
68	Elastic, Four-node Shear Panel	(68)
69	Heat Transfer Element (Eight-node planar, biquadratic quadrilateral with Reduced Integration)	(69)
70	Heat Transfer Element (Eight-node axisymmetric, biquadratic quadrilateral with Reduced Integration)	(70)
71	Heat Transfer Element (three-dimensional 20-node brick with Reduced Integration)	(71)
72	Bilinear Constrained Shell	(72)
73	Axisymmetric, Eight-node Quadrilateral for Arbitrary Loading, with Reduced Integration	(73)
74	Axisymmetric, Eight-node Quadrilateral for Arbitrary Loading, Herrmann Formulation, with Reduced Integration	(74)
75	Bilinear Thick Shell	(75)
76	Thin-walled Beam in Three Dimensions Without Warping	(76)
77	Thin-walled Beam in Three Dimensions Including Warping	(77)
78	Thin-walled Beam in Three Dimensions Without Warping	(78)
79	Thin-walled Beam in Three Dimensions Including Warping	(79)
80	Incompressible Arbitrary Quadrilateral Plane Strain	(80)
81	Incompressible Generalized Plane Strain Quadrilateral	(81)
82	Incompressible Arbitrary Quadrilateral Axisymmetric Ring	(82)
83	Incompressible Axisymmetric Torsional Quadrilateral	(83)
84	Incompressible Three-dimensional Arbitrarily Distorted Cube	(84)
85	Bilinear Heat Transfer Shell	(85)
86	Curved Quadrilateral Heat Transfer Shell	(86)
87	Curved Axisymmetric Heat Transfer Shell	(87)
88	Linear Axisymmetric Heat Transfer Shell	(88)

Table 1-13 Element Library (continued)

Element	Description	Code
89	Thick Curved Axisymmetric Shell	(89)
90	Thick Curved Axisymmetric Shell for Arbitrary Loading	(90)
91	Linear Plane Strain Semi-infinite Element	(91)
92	Linear Axisymmetric Semi-infinite Element	(92)
93	Quadratic Plane Strain Semi-infinite Element	(93)
94	Quadratic Axisymmetric Semi-infinite Element	(94)
95	Axisymmetric Quadrilateral with Bending Element	(95)
96	Axisymmetric Eight-node Distorted Quadrilateral with Bending	(96)
97	Double Friction and Gap Element Used with 95 or 96	(97)
98	Elastic Beam with Transverse Shear	(98)
101	Six-node Planar Semi-infinite Heat Transfer	(101)
102	Six-node Axisymmetric Semi-infinite Heat Transfer	(102)
103	Nine-node Planar Semi-infinite Heat Transfer	(103)
104	Nine-node Axisymmetric Semi-infinite Heat Transfer	(104)
105	Twelve-node Three-dimensional Semi-infinite Heat Transfer	(105)
106	Twenty-seven-node Three-dimensional Semi-infinite Heat Transfer	(106)
107	Twelve-node Three-dimensional Semi-infinite	(107)
108	Twenty-node Three-dimensional Semi-infinite	(108)
109	Eight-node Three-dimensional Magnetostatics	(109)
110	Twelve-node Three-dimensional Semi-infinite Magnetostatics	(110)
111	Four-node Quadrilateral Planar Magnetodynamic	(111)
112	Four-node Quadrilateral Axisymmetric Magnetodynamic	(112)
113	Eight-node Three-Dimensional Brick, Magnetodynamic	(113)
114	Four-node Quadrilateral Plane Stress, Reduced Integration with Hourglass Control	(114)
115	Four-node Quadrilateral Plane Strain, Reduced Integration with Hourglass Control	(115)
116	Four-node Quadrilateral Axisymmetric, Reduced Integration with Hourglass Control	(116)
117	Eight-node Three-dimensional Brick, Reduced Integration with Hourglass Control	(117)
118	Incompressible 4+1-node, Quadrilateral, Plane Strain, Reduced Integration with Hourglass Control	(118)
119	Incompressible 4+1-node, Quadrilateral, Axisymmetric, Reduced Integration with Hourglass Control	(119)

Table 1-13 Element Library (continued)

Element	Description	Code
120	Incompressible 8+1-node, Three-Dimensional Brick, Reduced Integration with Hourglass Control	(120)
121	Four-node, Heat Transfer Planar, Reduced Integration with Hourglass Control	(121)
122	Four-node, Heat Transfer Axisymmetric, Reduced Integration with Hourglass Control	(122)
123	Eight-node, Heat Transfer Three-dimensional Brick, Reduced Integration with Hourglass Control	(123)
124	Six-node, Plane Stress Triangle	(124)
125	Six-node, Plane Strain Triangle	(125)
126	Six-node, Axisymmetric Triangle	(126)
127	Ten-node, Tetrahedron	(127)
128	Incompressible, Six-node Triangle	(128)
129	Incompressible, Six-node Triangle	(129)
130	Incompressible, Ten-node Tetrahedral	(130)
131	Six-node, Heat Transfer Planar	(131)
132	Six-node, Heat Transfer Axisymmetric	(132)
133	Ten-node, Heat Transfer Tetrahedral	(133)
134	Four-node, Tetrahedral	(134)
135	Four-node, Heat Transfer Tetrahedral	(135)
136	Six-node, Pentahedral Element	(136)
137	Six-node, Heat Transfer Pentahedral	(137)
138	Three-node, Thin Shell	(138)
139	Four-node, Thin Shell	(139)
140	Four-node, Thick Shell, Reduced Integration with Hourglass Control	(140)
141	Not Available	
142	Eight-node Axisymmetric Rebar with Twist	(142)
143	Four-node Plane Strain Rebar	(143)
144	Four-node Axisymmetric Rebar	(144)
145	Four-node Axisymmetric Rebar with Twist	(145)
146	Eight-node Brick Rebar	(146)
147	Four-node Membrane Rebar	(147)
148	Eight-node Membrane Rebar	(148)
149	Three-dimensional, Eight-node Composite Brick	(149)

Table 1-13 Element Library (continued)

Element	Description	Code
150	Three-dimensional, Twenty-node Composite Brick	(150)
151	Quadrilateral, Plane Strain, Four-node Composite	(151)
152	Quadrilateral, Axisymmetric, Four-node Composite	(152)
153	Quadrilateral, Plane Strain, Eight-node Composite	(153)
154	Quadrilateral, Axisymmetric, Eight-node Composite	(154)
155	3 + 1-node, Plane Strain, Low-order, Triangular, Herrmann Formulations	(155)
156	3 + 1-node, Axisymmetric, Low-order, Triangular, Herrmann Formulations	(156)
157	4 + 1-node, Three-dimensional, Low-order, Tetrahedron, Herrmann Formulations	(157)
158	Three-node membrane	(158)
159	Not Available	
160	Two-dimensional (plane stress), 4-node, Isoparametric Quadrilateral with Piezoelectric Capability	(160)
161	Plane Strain Quadrilateral Element (Isoparametric) with Piezoelectric Capability	(161)
162	Axisymmetric Quadrilateral Element (Isoparametric) with Piezoelectric Capability	(162)
163	8-node Isoparametric Three-dimensional Hexahedron with Piezoelectric Capability	(163)
164	4-node, Tetrahedral with Piezoelectric Capability	(164)
165	2-node Plane Strain Membrane	(165)
166	2-node Axisymmetric Rebar Membrane	(166)
167	2-node Axisymmetric Rebar Membrane with Twist	(167)
168	3-node Plane Strain Membrane	(168)
169	3-node Axisymmetric Rebar Membrane	(169)
170	3-node Axisymmetric Rebar Membrane with Twist	(170)
171	2-node Cavity Surface Planar	(171)
172	2-node Cavity Surface Axisymmetric	(172)
173	3-node Cavity Surface Triangle	(173)
174	4-node Cavity Surface Quadrilateral	(174)
175	Three-dimensional, Heat Transfer, Eight-node Composite Brick	(175)
176	Three-dimensional, Heat Transfer, Twenty-node Composite Brick	(176)
177	Quadrilateral, Planar Heat Transfer, Four-node Composite	(177)
178	Quadrilateral, Axisymmetric Heat Transfer, Four-node Composite	(178)
179	Quadrilateral, Planar Heat Transfer, Eight-node Composite	(179)
180	Quadrilateral, Axisymmetric Heat Transfer, Eight-node Composite	(180)

Table 1-13 Element Library (continued)

Element	Description	Code
181	Four-node Three-dimensional Magnetostatics	(181)
182	Ten-node Three-dimensional Magnetostatics	(182)
183	Two-node Three-dimensional Magnetostatics	(183)
184	Three-dimensional Ten-node Tetrahedron	(184)
185	Solid Shell	(185)
186	Four-node Planar Interface Element	(186)
187	Eight-node Planar Interface Element	(187)
188	Eight-node Three-dimensional Interface Element	(188)
189	Twenty-node Three-dimensional Interface Element	(189)
190	Four-node Axisymmetric Interface Element	(190)
191	Eight-node Axisymmetric Interface Element	(191)
192	Six-node Three-dimensional Interface Element	(192)
193	Fifteen-node Three-dimensional Interface Element	(193)
194	Two-dimensional Cbush Element	(194)
195	Three-dimensional Cbush Element	(195)
196	Three-node Bilinear Heat Transfer Membrane	(196)
197	Six-node Biquadratic Heat Transfer Membrane	(197)
198	Four-node Isoparametric Heat Transfer Element	(198)
199	Eight-node Biquadratic Heat Transfer Membrane	(199)
200	Six-node Biquadratic Isoparametric Membrane	(200)
201	Two-dimensional Plane Stress Triangle	(201)
202	Fifteen-node Structural Pentahedral	(202)
203	Fifteen-node Heat Transfer Pentahedral	(203)
204	Six-node Magnetostatic Pentahedral	(204)
205	Fifteen-node Magnetostatic Pentahedral	(205)
206	Twenty-node Magnetostatic Brick	(206)
216	Five-node Structural Pyramid	(216)
217	Five-node Heat Transfer Pyramid	(217)
218	Thirteen-node Structural Pyramid	(218)
219	Thirteen-node Heat Transfer Pyramid	(219)
220	Four-node Planar Heat Transfer Interface Element	(220)
221	Eight-node Planar Heat Transfer Interface Element	(221)

Table 1-13 Element Library (continued)

Element	Description	Code
222	Eight-node Three-dimensional Heat Transfer Interface Element	(222)
223	Twenty-node Three-dimensional Heat Transfer Interface Element	(223)
224	Four-node Axisymmetric Heat Transfer Interface Element	(224)
225	Eight-node Axisymmetric Heat Transfer Interface Element	(225)
226	Six-node Three-dimensional Heat Transfer Interface Element	(226)
227	Fifteen-node Three-dimensional Heat Transfer Interface Element	(227)
228	Three-node Planar	(228)
229	Three-node Axisymmetric	(229)
230	Four-node Tetrahedral	(230)
231	Six-node Planar	(231)
232	Six-node Axisymmetric	(232)
233	Ten-node Tetrahedral	(233)
234	Eight-node Quadrilateral	(234)
235	Eight-node Axisymmetric Quadrilateral	(235)
236	20-node Brick	(236)
237	Six-node Pentahedral	(237)
238	Fifteen-node Pentahedral	(238)
239	3 + 1-node Plane Strain with Strain Smoothing	(239)
240	3 + 1-node Axisymmetric with Strain Smoothing	(240)
241	4 + 1-node Tetrahedron with Strain Smoothing	(241)
242	20-node Isoparametric Three-dimensional Hexahedron with Piezoelectric Capability	(242)
243	10-node Tetrahedral with Piezoelectric Capability	(243)

Table 1-14 Summary of Element Properties

Element	Number of Nodes	Number of Direct Stress	Number of Shear Stress	Number of Integration Points	Number of Degrees of Freedom	Number of Coordinates	Updated Lagrange Available
1	2	2	1	1	3	2	Yes
2	3	3	1	1	2	2	Yes
3	4	2	1	4	2	2	Yes
4	4	2	1	9	12	14	Yes
5	2	1	1	3	3	2	No
6	3	3	1	1	2	2	Yes
7	8	3	3	8	3	3	Yes
8	3	2	1	7	9	11	Yes
9	2	1	0	1	2/3	2/3	Yes
10	4	3	1	4	2	2	Yes
11	4	3	1	4	2	2	Yes
12	4	1	2	1	2/3	2/3	No
13	2	1	0	3	8	13	No
14	2	1	1	3	6	6	Yes
15	2	2	0	3	4	5	Yes
16	2	1	0	3	4	5	Yes
17	3	2	0	3	6	5	No
18	4	2	1	4	3	3	No
19	6	3	1	4	2	2	Yes
20	4	3	3	4	3	2	Yes
21	20	3	3	27	3	3	Yes
22	8	2	3	4	6	3	Yes
23	20	1	0	20	3	3	No
24	8	2	1	28	9	11	Yes
25	2	1	1	3	7	6	Yes
26	8	2	1	9	2	2	Yes
27	8	3	1	9	2	2	Yes
28	8	3	1	9	2	2	Yes
29	10	3	1	9	2	2	Yes
30	8	2	1	9	3	3	No

Table 1-14 Summary of Element Properties (continued)

Element	Number of Nodes	Number of Direct Stress	Number of Shear Stress	Number of Integration Points	Number of Degrees of Freedom	Number of Coordinates	Updated Lagrange Available
31	2	2	1	2	6	3	No
32	8	3	1	9	3	2	Yes
33	8	3	1	9	3	2	Yes
34	10	3	1	9	3	2	Yes
35	20	3	3	27	4	3	Yes
36	2	1	0	1	1	3	No
37	3	2	0	1	1	2	No
38	3	2	0	1	1	2	No
39	4	2	0	4	1	2	No
40	4	2	0	4	1	2	No
41	8	2	0	9	1	2	No
42	8	2	0	9	1	2	No
43	8	3	0	8	1	3	No
44	20	3	0	27	1	3	No
45	3	1	1	2	3	2	Yes
46	8	1	0	10	2	2	No
47	10	1	0	10	2	2	No
48	8	1	0	10	2	2	No
49	6	2	1	1	3	3	Yes
50	3	3	0	1	2	3	No
51	2	1	0	1	3	3	No
52	2	1	0	3	6	6	Yes
53	8	2	1	4	2	2	Yes
54	8	3	1	4	2	2	Yes
55	8	3	1	4	2	2	Yes
56	10	3	1	4	2	2	Yes
57	20	3	3	8	3	3	Yes
58	8	3	1	4	3	2	Yes
59	8	3	1	4	3	2	Yes
60	10	3	1	4	3	2	Yes

Table 1-14 Summary of Element Properties (continued)

Element	Number of Nodes	Number of Direct Stress	Number of Shear Stress	Number of Integration Points	Number of Degrees of Freedom	Number of Coordinates	Updated Lagrange Available
61	20	3	3	8	4	3	Yes
62	8	3	3	9	3	2	No
63	8	3	3	9	4	2	No
64	3	1	0	3	3	3	Yes
65	3	1	0	3	1	3	No
66	8	3	3	9	4	2	No
67	8	3	3	9	3	2	No
68	4	0	1	1	3	3	No
69	8	2	0	4	1	2	No
70	8	2	0	4	1	2	No
71	20	3	0	8	1	3	No
72	8	2	1	4	3	3	Yes
73	8	3	3	4	3	2	No
74	8	3	3	4	4	2	No
75	4	2	3	4	6	3	Yes
76	3	1	1	2	6	6	Yes
77	3	1	0	2	7	6	Yes
78	2	1	1	2	6	6	Yes
79	2	1	0	2	7	6	Yes
80	5	3	1	4	2	2	Yes
81	7	3	1	4	2	2	Yes
82	5	3	1	4	2	2	Yes
83	5	3	3	4	3	2	Yes
84	9	3	3	8	3	3	Yes
85	4	3	0	4	2/3	3	No
86	8	3	0	9	2/3	3	No
87	3	2	0	3	2/3	2	No
88	2	2	0	3	2/3	2	No
89	3	2	1	2	3	2	Yes
90	3	2	3	2	5	2	No

Table 1-14 Summary of Element Properties (continued)

Element	Number of Nodes	Number of Direct Stress	Number of Shear Stress	Number of Integration Points	Number of Degrees of Freedom	Number of Coordinates	Updated Lagrange Available
91	6	3	1	6	2	2	No
92	6	3	1	6	2	2	No
93	9	3	1	9	2	2	No
94	9	3	1	9	2	2	No
95	4	3	3	4	5	2	No
96	8	3	3	9	5	2	No
97	4	2	2	1	4	2	No
98	2	1	2	1	6	6	Yes
101	6	2	0	6	1	2	No
102	6	2	0	6	1	2	No
103	9	2	0	9	1	2	No
104	9	2	0	9	1	2	No
105	12	3	0	12	1	3	No
106	27	3	0	27	1	3	No
107	12	3	3	12	3	3	No
108	27	3	3	27	3	3	No
109	8	3	0	8	3	3	No
110	12	3	0	12	3	3	No
111	4	2	0	4	4	2	No
112	4	2	0	4	4	2	No
113	8	3	0	8	4	3	No
114	4	2	1	1	2	2	Yes
115	4	3	1	1	2	2	Yes
116	4	3	1	1	2	2	Yes
117	8	3	3	1	3	3	Yes
118	5	3	1	1	2	2	No
119	5	3	1	1	2	2	No
120	9	3	3	1	3	3	No
121	4	2	0	1	1	2	No
122	4	2	0	1	1	2	No

Table 1-14 Summary of Element Properties (continued)

Element	Number of Nodes	Number of Direct Stress	Number of Shear Stress	Number of Integration Points	Number of Degrees of Freedom	Number of Coordinates	Updated Lagrange Available
123	8	3	0	1	1	3	No
124	6	2	1	3	2	2	Yes
125	6	3	1	3	2	2	Yes
126	6	3	1	3	2	2	Yes
127	10	3	3	4	3	3	Yes
128	6	3	1	3	3	2	No
129	6	3	1	3	3	2	No
130	10	3	3	4	4	3	No
131	6	2	0	3	1	2	No
132	6	2	0	3	1	2	No
133	10	3	0	4	1	3	No
134	4	3	3	1	3	3	Yes
135	4	3	0	1	1	3	No
136	6	3	3	6	3	3	Yes
137	6	3	0	6	1	3	No
138	3	2	1	1	6	3	Yes
139	4	2	1	4	6	3	Yes
140	4	2	3	1	6	3	Yes
141	Not Available						
142	8	1	0	10	3	2	No
143	4	1	0	10	2	2	No
144	4	1	0	10	2	2	No
145	4	1	0	10	3	2	No
146	8	1	0	20	3	3	No
147	4	1	0	20	3	3	No
148	8	1	0	20	3	3	No
149	8	3	3	up to 2040	3	3	Yes
150	20	3	3	up to 2040	3	3	Yes
151	4	3	1	up to 2040	2	2	Yes
152	4	3	1	up to 2040	2	2	Yes

Table 1-14 Summary of Element Properties (continued)

Element	Number of Nodes	Number of Direct Stress	Number of Shear Stress	Number of Integration Points	Number of Degrees of Freedom	Number of Coordinates	Updated Lagrange Available
153	8	3	1	up to 2040	2	2	Yes
154	8	3	1	10	2	2	Yes
155	3 + 1	3	1	3	3	2	Yes
156	3 + 1	3	1	3	3	2	Yes
157	4 + 1	3	3	4	4	3	Yes
158	3	2	1	1	3	3	No
159	Not Available						
160	4	2	1	4	3	2	No
161	4	3	1	4	3	2	No
162	4	3	1	4	3	2	No
163	8	3	3	8	4	3	No
164	4	3	3	1	4	3	No
165	2	1	0	10	2	2	No
166	2	1	0	10	2	2	No
167	2	1	0	10	3	2	No
168	3	1	0	10	2	2	No
169	3	1	0	10	2	2	No
170	3	1	0	10	3	2	No
171	2	0	0	0	2	2	No
172	2	0	0	0	2	2	No
173	3	0	0	0	3	3	No
174	4	0	0	0	3	3	No
175	8	3	0	up to 2040	1	3	No
176	20	3	0	up to 2040	1	3	No
177	4	2	0	up to 2040	1	2	No
178	4	2	0	up to 2040	1	2	No
179	8	2	0	up to 2040	1	2	No
180	8	2	0	up to 2040	1	2	No
181	4	3	0	1	3	3	No
182	10	3	0	4	3	3	No

Table 1-14 Summary of Element Properties (continued)

Element	Number of Nodes	Number of Direct Stress	Number of Shear Stress	Number of Integration Points	Number of Degrees of Freedom	Number of Coordinates	Updated Lagrange Available
183	2	0	0	0	3	3	No
184	10	3	3	4	3	3	Yes
185	8	3	3	Number of Layers	3	3	Yes
186	4	1	1	2	2	2	Yes
187	8	1	1	3	2	2	Yes
188	8	1	2	4	3	3	Yes
189	20	1	2	8 or 9	3	3	Yes
190	4	1	1	2	2	2	Yes
191	8	1	1	3	2	2	Yes
192	6	1	2	3	3	3	Yes
193	6	1	2	6 or 7	3	3	Yes
194	2	2	1	1	2	2	Yes*
195	2	3	3	1	3	3	Yes*
196	3	2	0	1	1	3	No
197	6	2	0	7	1	3	No
198	4	2	0	4	1	3	No
199	8	2	0	9	1	3	No
200	6	2	1	7	3	3	No
201	3	2	1	1	2	2	Yes
202	15	3	3	21	3	3	Yes
203	15	3	0	21	1	3	No
204	6	3	0	6	3	3	No
205	15	3	0	21	3	3	No
206	20	3	0	27	3	3	No
216	5	3	3	5	3	3	Yes
217	5	3	0	5	1	3	No
218	13	3	3	8	3	3	Yes
219	13	3	0	8	1	3	No
220	4	1	0	2	1	2	Yes
221	8	1	0	3	1	2	Yes

Table 1-14 Summary of Element Properties (continued)

Element	Number of Nodes	Number of Direct Stress	Number of Shear Stress	Number of Integration Points	Number of Degrees of Freedom	Number of Coordinates	Updated Lagrange Available
222	8	1	0	4	1	3	Yes
223	20	1	0	8 or 9	1	3	Yes
224	4	1	0	2	1	2	Yes
225	8	1	0	3	1	2	Yes
226	6	1	0	3	1	3	Yes
227	6	1	0	6 or 7	1	3	Yes
228	3	2	0	1	4	2	No
229	3	2	0	1	4	2	No
230	4	3	0	1	4	3	No
231	6	2	0	3	4	2	No
232	6	2	0	3	3	2	No
233	10	3	0	4	4	3	No
234	8	2	0	9	4	2	No
235	8	2	0	9	4	2	No
236	20	3	0	27	4	3	No
237	6	3	0	6	4	3	No
238	15	3	0	18	4	3	No
239	3 + 1	3	1	3	2	2	Yes
240	3 + 1	3	1	3	2	2	Yes
241	4 + 1	3	3	4	3	3	Yes
242	20	3	3	27	4	3	No
243	10	3	3	4	4	3	No

*Cbush elements only use an updated system if the element coordinate system is used.

Table 1-15 Overview of Marc Heat Transfer Element Types

Element Number	Heat Transfer	Joule Heating	Gap	Channel	Electrostatic	Magnetostatic
36	yes*	yes	no**	no	no	no
37	yes	yes	no	no	yes	yes
38	yes	yes	no	no	yes	yes
39	yes	yes	yes	yes	yes	yes
40	yes	yes	yes	yes	yes	yes
41	yes	yes	yes	yes	yes	yes
42	yes	yes	yes	yes	yes	yes
43	yes	yes	yes	yes	yes	no
44	yes	yes	yes	yes	yes	no
50	yes	no	no	no	yes	no
65	yes	yes	no	no	no	no
69	yes	yes	yes	yes	yes	yes
70	yes	yes	yes	yes	yes	yes
71	yes	yes	yes	yes	yes	no
85	yes	no	no	no	yes	no
86	yes	no	no	no	yes	no
87	yes	no	no	no	yes	no
88	yes	no	no	no	yes	no
101	yes	no	no	no	yes	yes
102	yes	no	no	no	yes	yes
103	yes	no	no	no	yes	yes
104	yes	no	no	no	yes	yes
105	yes	no	no	no	yes	no
106	yes	no	no	no	yes	no
109	no	no	no	no	no	yes
110	no	no	no	no	no	yes
121	yes	yes	yes	yes	yes	yes
122	yes	yes	yes	yes	yes	yes
123	yes	yes	yes	yes	yes	no

*yes – capabilities are available.**no – capabilities are not available

Table 1-15 Overview of Marc Heat Transfer Element Types (continued)

Element Number	Heat Transfer	Joule Heating	Gap	Channel	Electrostatic	Magnetostatic
131	yes	yes	no	no	yes	yes
132	yes	yes	no	no	yes	yes
133	yes	yes	no	no	yes	no
135	yes	yes	no	no	yes	no
136	yes	yes	no	no	yes	no
175	yes	yes	no	no	yes	no
176	yes	yes	no	no	yes	no
177	yes	yes	no	no	yes	yes
178	yes	yes	no	no	yes	yes
179	yes	yes	no	no	yes	yes
180	yes	yes	no	no	yes	yes
181	no	no	no	no	no	yes
182	no	no	no	no	no	yes
183	no	no	no	no	no	yes
196	yes	yes	no	no	yes	no
197	yes	yes	no	no	yes	no
198	yes	yes	no	no	yes	no
199	yes	yes	no	no	yes	no
203	yes	yes	no	no	yes	no
217	yes	yes	no	no	yes	no
219	yes	yes	no	no	yes	no
220	yes	no	no	no	no	no
221	yes	no	no	no	no	no
222	yes	no	no	no	no	no
223	yes	no	no	no	no	no
224	yes	no	no	no	no	no
225	yes	no	no	no	no	no
226	yes	no	no	no	no	no
227	yes	no	no	no	no	no
228						

*yes – capabilities are available.**no – capabilities are not available

Follow Force Stiffness Contribution

When activating the FOLLOW FOR parameter, the distributed loads are calculated based upon the current deformed configuration. It is possible to activate an additional contribution which goes into the stiffness matrix. This improves the convergence. This capability is available for element types 3, 7, 10, 11, 18, 72, 75, 80, 82, 84, 114, 115, 116, 117, 118, 119, 120, 139, 140, 149, 151, 152, 160, 161, 162, 163 and 185.

Explicit Dynamics

The explicit dynamics formulation IDYN=5 model is restricted to the following elements:

2, 3, 5, 6, 7, 9, 11, 18, 19, 20, 52, 64, 75, 98, 114, 115, 116, 117, 118, 119, 120, 138, 139, and 140

When using this formulation, the mass matrix is defined semi-analytically; that is., no numerical integration is performed. In addition, a quick method is used to calculate the stability limit associated with each element. For these reasons, this capability has been limited to the elements mentioned above.

Local Adaptive Mesh Refinement

Marc has a capability to perform local adaptive mesh refinement to improve the accuracy of the solution. This capability is invoked by using the ADAPTIVE parameter and model definition option. The adaptive meshing is available for the following 2-D and 3-D elements:

2, 3, 6, 7, 10, 11, 18, 19, 20, 37, 38, 39, 40, 43, 75, 80, 81, 82, 83, 84, 111, 112, 113, 114, 115, 116, 117, 118, 119, 120, 121, 122, 123, 139, 140, 196, 198, 201, 239, 240, and 241.

The adaptive procedure works by subdividing the existing element, based upon the error criteria specified. In two-dimensional analyses, triangles are divided into four triangles and quadrilaterals are divided into four quadrilaterals, while in three-dimensional analyses, tetrahedrals are subdivided into eight tetrahedrals and hexahedrals are subdivided into eight hexahedrals.

2 Marc Element Classifications

- Class 1 Elements 1 and 5 63
- Class 2 Elements 9 and 36 64
- Class 3 Elements 2, 6, 37, 38, 50, 196, 201, 228, and 229 65
- Class 4 Elements 3, 10, 11, 18, 19, 20, 39, 40, 80, 81, 82, 83, 111, 112, 160, 161, 162, 198, 230, and 231 67
- Class 5 Elements 7, 43, 84, 113, and 163 69
- Class 6 Elements 64 and 65 71
- Class 7 Elements 26, 27, 28, 29, 30, 32, 33, 34, 41, 42, 62, 63, 66, 67, 199, 234, and 235 72
- Class 8 Elements 53, 54, 55, 56, 58, 59, 60, 69, 70, 73, and 74 74
- Class 9 Elements 21, 35, 44, 236 and 242 76
- Class 10 Elements 57, 61, and 71 78
- Class 11 Elements 15, 16, and 17
- Class 12 Element 45 81
- Class 13 Elements 13, 14, 25, and 52 82
- Class 14 Elements 124, 125, 126, 128, 129, 131, 132, 197, 200, 231, and 232 83

- Class 15 Elements 127, 130, 133, 233 and 243 85
- Class 16 Elements 114, 115, 116, 118, 119, 121, and 122 87
- Class 17 Elements 117, 120, and 123 88
- Class 18 Elements 134, 135, 164 and 247 89
- Class 19 Elements 136, 137, 204, and 237 90
- Class 20 Elements 202, 203, 205, and 238 91
- Class 21 Elements 216 and 217 92
- Class 22 Elements 218 and 219 93
- Special Elements Elements 4, 8, 12, 22, 23, 24, 31, 45, 46, 47, 48, 49, 68, 72, 75, 76, 77, 78, 79, 85, 86, 87, 88, 89, 90, 91, 92, 93, 94, 95, 96, 97, 98, 99, 100, 101, 102, 103, 104, 105, 106, 107, 108, 109, 110, 138, 139, 140, 142, 143, 144, 145, 146, 147, 148, 149, 150, 151, 152, 153, 154, 155, 156, 157, 165, 166, 167, 168, 169, 170, 171, 172, 173, 174, 175, 176, 177, 178, 179, 180, 181, 182, 183, 185, 186, 187, 188, 189, 190, 191, 192, 193, 194, 195, 220, 221, 222, 223, 224, 225, 226, 227, 239, 240, 241, 244, 245, 246 and 247 94

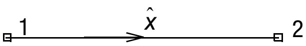
Marc contains a large number of element types. Functionally, these are used to model trusses, beams, plates, shells, plane stress, plane strain and general three-dimensional continua. Where appropriate, there are corresponding heat transfer elements which are found in Table 1-13. In CHAPTER 2, these elements are separated into classes based on their respective node number, interpolation functions, and integration schemes. In CHAPTER 3, these elements are detailed in numerical order.

Class 1

Elements 1 and 5

This class consists of two-noded elements with linear interpolation along the length and cubic interpolation normal to the length.

There is a local coordinate system $\hat{x}$, which is in the direction from node 1 to node 2 as shown below, associated with the element.



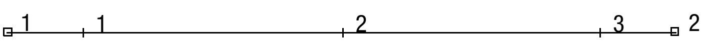
The displacement assumptions are as follows:

$$u^T = a_1 + a\hat{x}$$

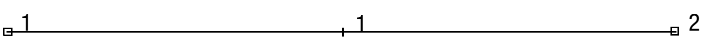
$$u^N = a_3 + a_4\hat{x} + a_5\hat{x}^2 + a_6\hat{x}^3$$

where u^T and u^N are displacements tangential and normal to $\hat{x}$ respectively.

There are three degrees of freedom associated with each of the nodes, two global displacements, and one rotation. The integration along the length of the element is performed with one or three Gaussian integration points, whose spacing is shown below.



Stiffness Integration Element Type 5



Stiffness Integration Element Type 1

This element uses two-point integration points along the length to form equivalent nodal loads and the mass matrix.

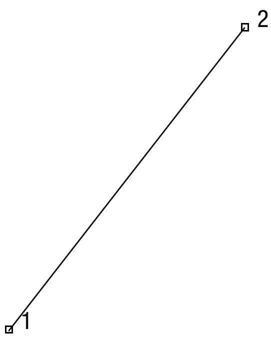
The stress-strain relation is integrated through the thickness by Simpson's rule, using the number of points defined in the SHELL SECT parameter (11 by default). The first layer is on the positive normal side which is obtained by rotating the $\hat{x}$ axis 90° counterclockwise.

The elements in class 11 (elements 15 and 16) are preferable elements to use.

Class 2

Elements 9 and 36

This class consists of two-noded elements with linear interpolation along the length as shown below.

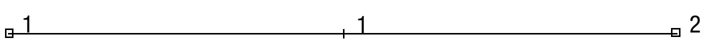


There is only stiffness associated along the length of the element. The interpolation function can be expressed as follows:

$$\psi = \psi_1 \xi + \psi_2 (1 - \xi)$$

where ψ_1, ψ_2 are the values of the function at the end nodes and ζ is the normalized coordinate $(0 \le \zeta \le 1)$.

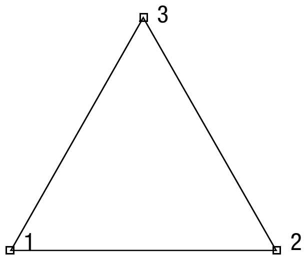
Element 9 has three degrees of freedom per node, and element 36 has one degree of freedom per node. There is a single integration point at the centroid the element as shown below. The mass matrix uses two-point integration.



Class 3

Elements 2, 6, 37, 38, 50, 196, 201, 228, and 229

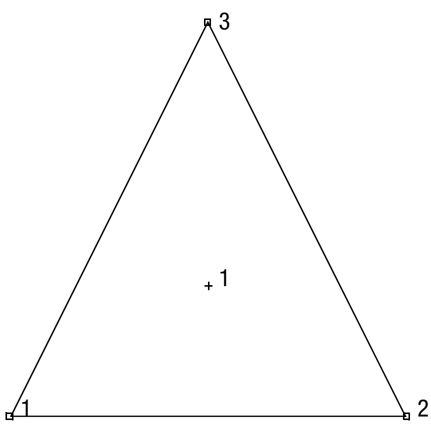
This class consists of three-noded triangular elements with linear interpolation functions as shown below.



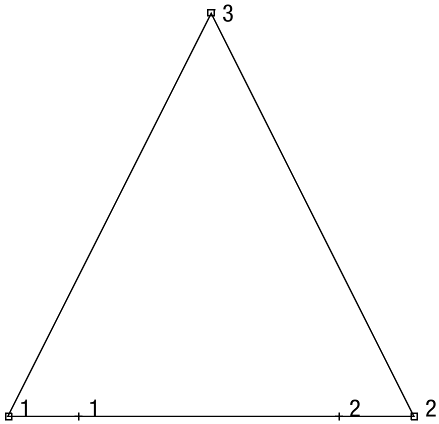
The node numbering must be counterclockwise in the plane of the element. The function is assumed to be expressed in the form

$$\psi = a_0 + a_1 x + a_2 y$$

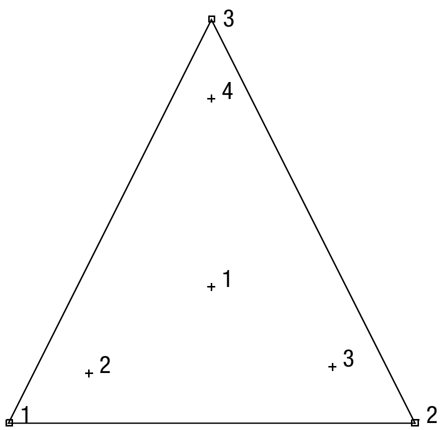
For elements 2 and 6, there are two degrees of freedom. For elements 37 and 38, there is one degree of freedom. For heat transfer shell element 50, there are either two or three degrees of freedom. There is a single integration point at the centroid of the element as shown below.



For distributed surface pressures (flux), two integration points, as shown below, are used.



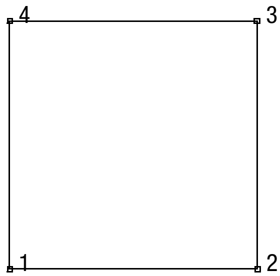
For volumetric forces (flux) or mass matrix, four integration points, as shown below, are used.



Class 4

Elements 3, 10, 11, 18, 19, 20, 39, 40, 80, 81, 82, 83, 111, 112, 160, 161, 162, 198, 230, and 231

This class consists of four-noded isoparametric elements with bilinear interpolation. The element node numbering must be given in counterclockwise direction following the right-hand rule as shown below.



The element is formed by mapping from the x-y (z-r) plane to the ξ, η plane.

Both the mapping and the function assumption take the form:

$$x = a_0 + a_1\xi + a_2\eta + a_3\xi\eta$$
$$\psi = b_0 + b_1\xi + b_2\eta + b_3\xi\eta$$

Either the coordinate or function can be expressed in terms of the nodal quantities by the interpolation functions.

$$x = \sum_{i=1}^{4} x_i\phi_i \quad \text{where}$$

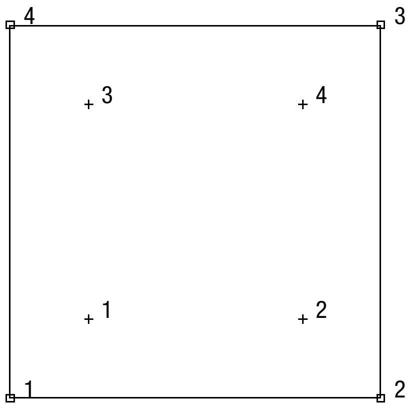
$$\phi_1 = \frac{1}{4}(1-\xi)(1-\eta)$$

$$\phi_2 = \frac{1}{4}(1+\xi)(1-\eta)$$

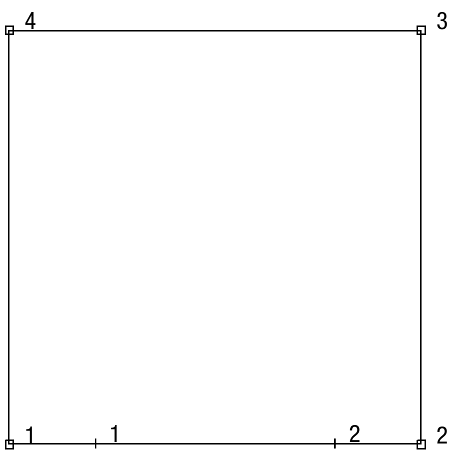
$$\phi_3 = \frac{1}{4}(1+\xi)(1+\eta)$$

$$\phi_4 = \frac{1}{4}(1-\xi)(1+\eta)$$

There are three degrees of freedom associated with each node for elements 18 and 20. There are two degrees of freedom per node for elements 3, 10, 11, and 19. There is one degree of freedom for elements 39 and 40. These elements use four-point Gaussian integration as shown below.



For distributed surface pressures, two-point Gaussian integration, as shown below, is used.



For elements 10, 11, 19, and 20, an optional integration scheme can be used which imposes a constant dilatational strain on the element. This is equivalent to a selective integration where the four Gaussian points are used for the deviatoric contribution of strain and the centroid for the dilatation contribution. This is flagged using the second parameter of the GEOMETRY option.

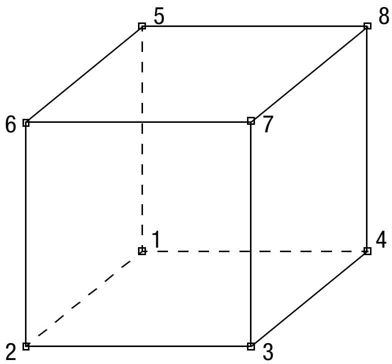
For elements 3 and 11, an optional assumed strain interpolation formulation is available. This significantly improves the behavior of this element in bending. This is flagged using the third parameter of the GEOMETRY option.

For elements 80, 81, 82, and 83, there is one extra node with a single degree of freedom (pressure). These elements use a mixed formulation for incompressible analysis.

Class 5

Elements 7, 43, 84, 113, and 163

This class consists of eight-noded, isoparametric, three-dimensional brick elements with trilinear interpolation. The node numbering must follow the rules as shown and given below.



Nodes 1, 2, 3, and 4 are corners of one face, given in counterclockwise order when viewed from inside the element. Node 5 is on the same edge as node 1, node 6 as node 2, node 7 as node 3, and node 8 as node 4. The element is based on the following type of displacement assumption and mapping from the x-y-z space into a cube in the ξ, η, ζ space.

$$x = a_0 + a_1 \xi + a_2 \eta + a_3 \xi + a_4 \xi \eta + a_5 \zeta \xi + a_6 \xi \zeta + a_7 \xi \eta \zeta$$

$$\psi = b_0 + b_1 \xi + b_2 \eta + b_3 \xi + b_4 \xi \eta + b_5 \eta \zeta + b_6 \xi \zeta + b_7 \xi \eta \zeta$$

Either the coordinate or function can be expressed in terms of the nodal quantities by the integration functions.

$$x = \sum_{i=1}^{8} x_i \phi_i \qquad \text{where}$$

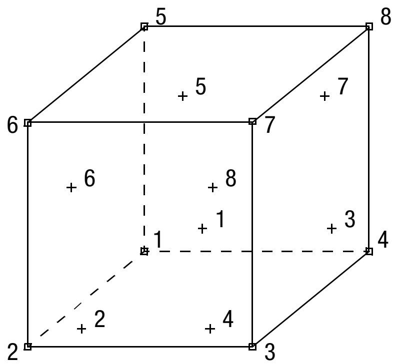
$$\phi_1 = \frac{1}{8}(1-\xi)(1-\eta)(1-\zeta) \qquad \phi_5 = \frac{1}{8}(1-\xi)(1-\eta)(1+\zeta)$$

$$\phi_2 = \frac{1}{8}(1+\xi)(1-\eta)(1-\zeta) \qquad \phi_6 = \frac{1}{8}(1+\xi)(1-\eta)(1+\zeta)$$

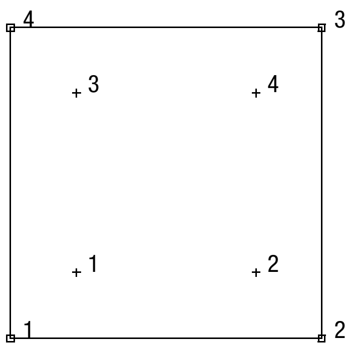
$$\phi_3 = \frac{1}{8}(1+\xi)(1+\eta)(1-\zeta) \qquad \phi_7 = \frac{1}{8}(1+\xi)(1+\eta)(1+\zeta)$$

$$\varphi_4 = \frac{1}{8}(1-\xi)(1+\eta)(1-\zeta) \qquad \phi_8 = \frac{1}{8}(1-\xi)(1+\eta)(1+\zeta)$$

There are three degrees of freedom associated with each node for element 7 and one degree of freedom for each node for element 43. These elements use eight-point Gaussian integration as shown below.



For distributed pressure (flux) loads, four-point Gaussian integration, as shown below, is used.



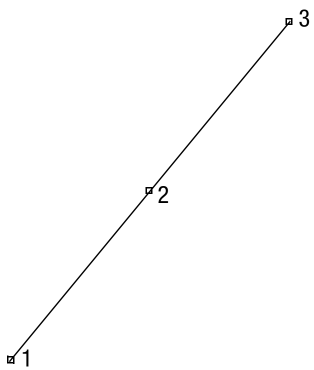
An optional integration scheme can be used for element 7 which imposes a constant dilatational strain on the element. This is equivalent to a selective integration where the eight Gaussian points are used for the deviatoric contribution of strain and the centroid for the dilatation contribution. This is flagged using the second parameter of the GEOMETRY option. For element 7, an optional assumed strain interpolation formulation is available. This significantly improves the behavior of this element in bending. This is flagged using the third parameter of the GEOMETRY option.

For element 84, there is one extra node with a single degree of freedom (pressure). This element uses a mixed formulation for incompressible analysis.

Class 6

Elements 64 and 65

This class consists of three-noded isoparametric links with quadratic interpolation along the length as shown below.



The node numbering is as shown. The interpolation function is such that element is parabolic. The element is based on the following displacement assumption, and mapping from the x-y-z space into a straight line.

$$x = a_0 + a_1 \zeta + a_2 \zeta^2$$

Either the coordinates or function can be expressed in terms of the nodal quantities by the interpolation functions

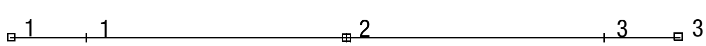
$$x = \sum_{i=1}^{3} x_i \phi_i \qquad \text{where}$$

$$\phi_1 = \frac{1}{2} \xi (\xi - 1)$$

$$\phi_2 = \frac{1}{2} \xi (1 + \xi)$$

$$\phi_3 = (1 - \xi)(1 + \xi)$$

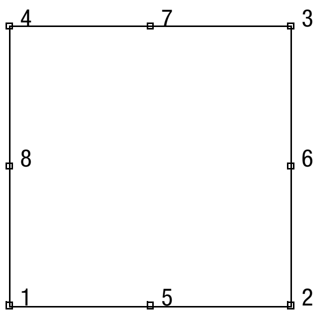
There are three degrees of freedom associated with each node for element 64. There is one degree of freedom per node for element 65. These elements use three-point Gaussian integration as shown below.



Class 7

Elements 26, 27, 28, 29, 30, 32, 33, 34, 41, 42, 62, 63, 66, 67, 199, 234, and 235

This class consists of eight-noded isoparametric distorted quadrilateral elements with biquadratic interpolation and full integration. The node numbering must be counterclockwise in the plane as shown below.



The four corner nodes come first and then the midside nodes with node 5 on the 1-2 edge, etc. The interpolation function is such that each edge has parabolic variation along itself. The element is based on the following displacement assumption, and mapping from the x-y (z-r) space to a square in the ζ, η space.

$$x = a_0 + a_1 \xi + a_2 \eta + a_3 \xi^2 + a_4 \xi \eta + a_5 \eta^2 + a_6 \xi^2 \eta + a_7 \xi \eta^2$$

Either the coordinates or function can be expressed in terms of the nodal quantities by the interpolation functions.

$$x = \sum_{i=1}^{8} x_i \phi_i \qquad \text{where}$$

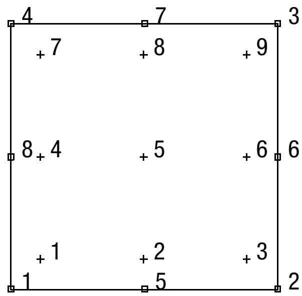
$$\phi_1 = \frac{1}{4}(1-\xi)(1-\eta)(-1-\xi-\eta) \qquad \phi_5 = \frac{1}{2}(1-\xi^2)(1-\eta)$$

$$\phi_2 = \frac{1}{4}(1+\xi)(1-\eta)(-1+\xi-\eta) \qquad \phi_6 = \frac{1}{2}(1+\xi)(1-\eta^2)$$

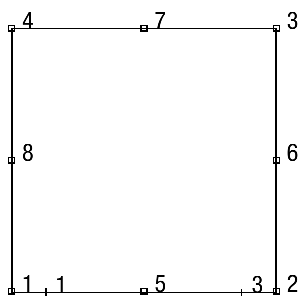
$$\phi_3 = \frac{1}{4}(1+\xi)(1+\eta)(-1+\xi+\eta) \qquad \phi_7 = \frac{1}{2}(1-\xi^2)(1+\eta)$$

$$\phi_4 = \frac{1}{4}(1-\xi)(1+\eta)(-1-\xi+\eta) \qquad \phi_8 = \frac{1}{2}(1-\xi)(1-\eta^2)$$

These elements use nine-point Gaussian integration as shown below for the calculation of the stiffness and mass matrix and evaluation of equivalent volumetric loads.



For distributed surface pressures (flux), three-point integration is used as shown below.



Elements 29 and 34 have two additional nodes which are used to formulate a generalized plane strain condition. These additional nodes are shared between all the elements in the mesh.

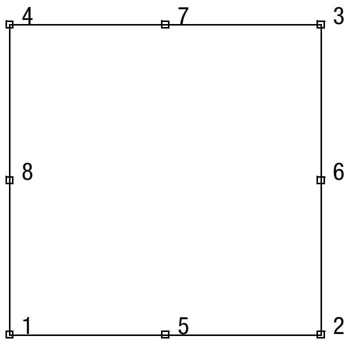
Elements 62 and 63 have been modified to be used in conjunction with the Fourier option. This allows nonaxisymmetric loads to be applied on an axisymmetric object. These elements can only be used for linear elastic static behavior.

Elements 32, 33, 34, and 63 have been modified so that they can be used for problems concerning incompressible or nearly incompressible materials. These elements use an extension of the Herrmann variational principle. They have an additional degree of freedom at the corner nodes, which represents the hydrostatic pressure. This function is interpolated linearly through the element.

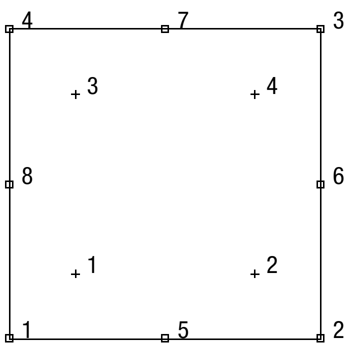
Class 8

Elements 53, 54, 55, 56, 58, 59, 60, 69, 70, 73, and 74

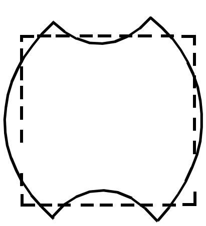
This class consists of eight-noded isoparametric distorted quadrilateral elements with biquadratic interpolation and reduced integration (see illustration below).



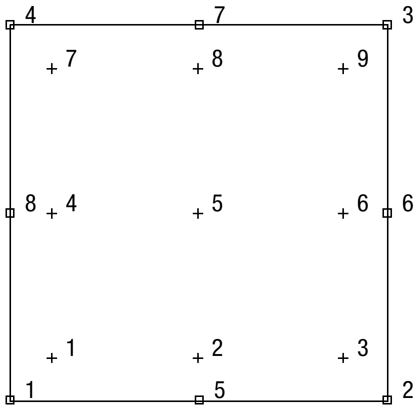
These elements have the same node numbering and interpolation functions as the elements in Class 7. The only difference is that these elements use a reduced integration technique in calculation of the stiffness matrix. The stiffness matrix is calculated using four-point Gaussian integration as shown below.



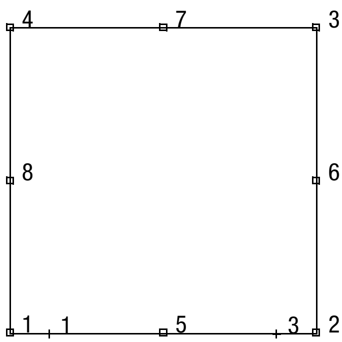
Using this method, quadratic functions as used in the interpolation functions are not integrated exactly. The contributions of the higher order terms in the deformation field are neglected. This results in singular modes; that is, deformations which do not contribute to the strain energy in the element. These elements have one such mode, as shown below.



The integration of the mass matrix and the consistent volumetric loads is done using nine-point Gaussian integration as shown below.



The surface distributed loads is integrated using three-point Gaussian integration as shown below.



Elements 56 and 60 have two additional nodes which are used to formulate a generalized plane strain condition. These additional nodes are shared between all the elements in the mesh.

Elements 73 and 74 have been modified to be used in conjunction with the Fourier option. This allows nonaxisymmetric loads to be applied on an axisymmetric object. These elements can only be used for linear elastic static behavior.

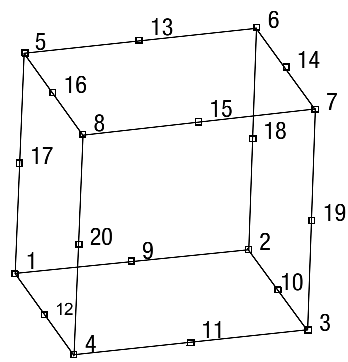
Elements 58, 59, 60, and 74 have been modified so that they can be used for problems concerning incompressible or nearly incompressible materials. These elements use an extension of the Herrmann principle. They have an additional degree of freedom at the corner nodes which represents the hydrostatic pressure. This function is interpolated linearly through the element.

Class 9

Elements 21, 35, 44, 236 and 242

This class consists of 20-node isoparametric distorted three-dimensional brick elements with full integration. The convention for the node numbering is as follows:

> Nodes 1, 2, 3, 4 are the corners of one face, given in counterclockwise order when viewed from inside the element. Nodes 5, 6, 7, 8 are corners of the opposite face; that is node 5 shares an edge with 1, node 6 with 2, etc. Nodes 9, 10, 11, 12 are the midside nodes of the 1, 2, 3, 4 face; node 9 between 1 and 2, node 10 between 2 and 3, etc. Similarly, nodes 13, 14, 15, 16 are midside nodes on the 5, 6, 7, 8 face; node 13 between 5 and 6, etc. Finally, node 17 is the midpoint of the 1-5 edge, node 18 of the 2-6 edge, etc. This is shown below. Each edge forms a parabola.

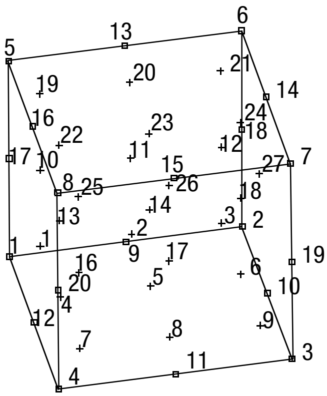


The element is based on the following displacement assumption, and mapping from the x-y-z space to a cube in ξ, η, ζ space.

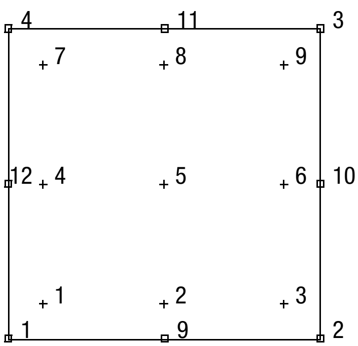
$$x = a_0 + a_1 \xi + a_2 \eta + a_3 \zeta + a_4 \xi^2 + a_5 \xi \eta + a_6 \eta^2 + a_7 \eta \zeta +$$
$$a_8 \zeta^2 + a_9 \xi \zeta + a_{10} \zeta^2 \eta + a_{11} \zeta \eta^2 + a_{12} \eta^2 \zeta + a_{13} \eta \zeta^2 +$$
$$a_{14} \xi^2 \zeta + a_{15} \xi \zeta^2 + a_{16} \xi \eta \zeta + a_{17} \xi^2 \eta \zeta + a_{18} \xi \eta^2 \zeta +$$
$$a_{19} \xi \eta \zeta^2$$

The resulting interpolation function is triquadratic.

These elements use 27-point Gaussian integration as shown below.



These integration points can be considered to be made up of three planes perpendicular to the ζ axis (i.e., parallel to the 1, 2, 3, 4, 9, 10, 11, and 12 face such that integration points 1-9 are closest to the 1, 2, 3, 4 face; integration points 19-27 are closest to the 5, 6, 7, 8 face; and integration points 10-18 are in between). Integration point 14 is in the centroid of the element. The calculation of the mass matrix of equivalent volumetric force uses the same integration scheme. Surface pressures are integrated using a nine-point Gaussian integration as shown below.

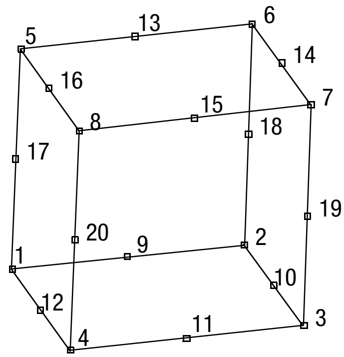


Element 35 has been modified so that it can be used for problems concerning incompressible or nearly incompressible materials. This element uses an extension of the Herrmann variational principle. It has an additional degree of freedom at the corner nodes which represents the hydrostatic pressure. This function is interpolated linearly throughout the element.

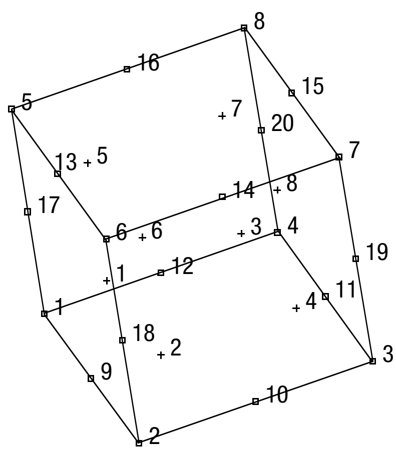
Class 10

Elements 57, 61, and 71

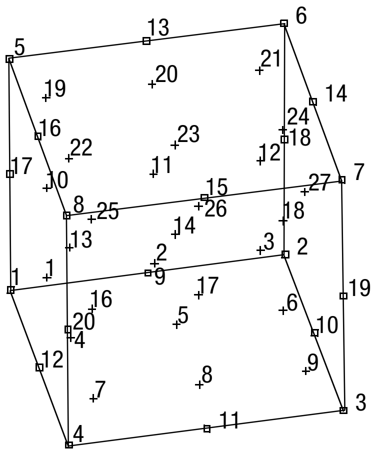
This class consists of 20-noded isoparametric, distorted, three-dimensional brick elements with reduced integration using triquadratic interpolation functions.These elements have the same node numbering and interpolation functions as the elements in Class 9. The element nodes are shown below.



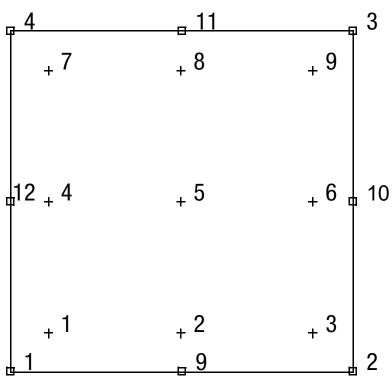
The only difference is that these elements use a reduced integration technique in calculation of the stiffness matrix. The stiffness matrix is calculated using eight-point Gaussian integration as shown below.



Using reduced integration, quadratic functions as used in the interpolating functions are not integrated exactly. The contribution of the higher order terms in the deformation field are neglected. This results in singular modes; that is, deformations which do not contribute to the strain energy in the element. These elements have six such modes. The integration of the mass matrix and the consistent volumetric loads is done using 27-point Gaussian integration as shown below.



The surface distributed loads are integrated using nine-point Gaussian integration as shown below.

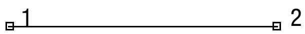


Element 61 has been modified so that it can be used for problems concerning incompressible or nearly incompressible materials. This element uses an extension of the Herrmann variational principle. It has an additional degree of freedom at the corner nodes which represents the hydrostatic pressure. This function is interpolated linearly throughout the element.

Class 11

Elements 15, 16, and 17

This class consists of two-noded axisymmetric curved shell, curved beam and pipe bend with cubic interpolation functions. These elements are shown below.



They use a Hermite cubic interpolation function to express the nodal displacements. In this case,

$$u(\xi) = H_{01}(\xi)\,u_1 + H_{02}(\xi)\,u_2 + H_{11}(\xi)u'_1 + H_{12}(\xi)u'_2$$

where u is the value of the function at the nodes and u' is its first derivative. The Hermite polynomials are as follows:

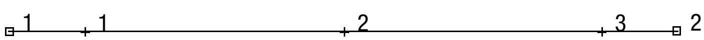
$$H_{01}(\xi) = (2+\xi)(1-\xi)^2/4$$

$$H_{02}(\xi) = (2-\xi)(1+\xi)^2/4$$

$$H_{11}(\xi) = (1+\xi)(1-\xi)^2/4$$

$$H_{12}(\xi) = -(1-\xi)(1+\xi)^2/4$$

These elements use three-point Gaussian integration as shown below.

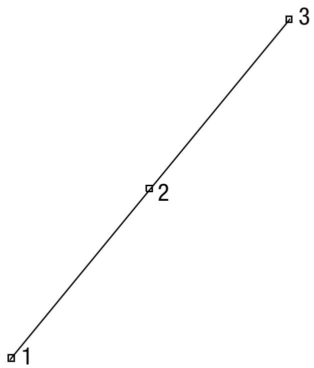


The stress-strain law is integrated through the thickness using Simpson's rule. The elements have two global displacement and two derivative degrees of freedom at each node.

Class 12

Element 45

This class contains a three-noded curved Timoshenko beam in a plane which allows transverse shear as well as axial straining as shown below.



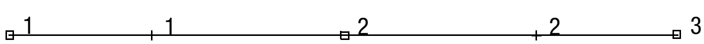
It uses quadratic interpolation for the beam axis, together with quadratic polynomial interpolation for the cross-section rotation. The center node (node 2) has been added to allow this quadratic representation of the cross-section rotation. Hence:

$$u(\xi) = \frac{1}{2}\xi(\xi - 1)u_1 + (1 - \xi)(1 + \xi)u_2 + \frac{1}{2}\xi(1 + \xi)u_3$$

and

$$\phi(\xi) = \frac{1}{2}\xi(\xi - 1)\phi_1 + (1 - \xi)(1 + \xi)\phi_2 + \frac{1}{2}\xi(1 + \xi)\phi_3$$

The interpolation of u, v, and ϕ are uncoupled. The calculation of the generalized strain terms leads to coupling in the shear calculation. This element is integrated using two-point Gaussian integration along the beam axis as shown below.



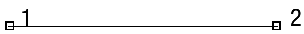
The mass matrix and volumetric loads are integrated based upon three-point Gaussian integration.

The stress-strain law is integrated through the thickness using Simpson's rule.

Class 13

Elements 13, 14, 25, and 52

This class consists of two-noded beam elements written in the global x-y-z space as shown below.



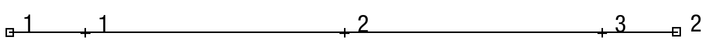
Each one of these elements has a different function summarized as follows:

Element 13 Open section thin-walled beam
Element 14 Closed section thin-walled beam
Element 25 Closed section thin-walled beam
Element 52 Beam

The interpolation functions can be summarized as follows:

	Along the Axis	Normal to Axis	Twist
13	Cubic	Cubic	Cubic
14	Linear	Cubic	Linear
25	Cubic	Cubic	Linear
52	Linear	Cubic	Linear

These elements are integrated using three-point Gaussian integration along the beam axis as shown below.

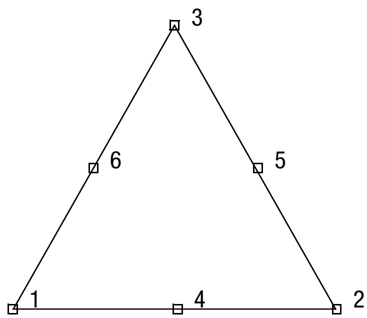


The stress-strain law is integrated using Simpson's rule through the cross section of the element. Elements 14 and 25 have an annular default cross section. The default values of the annular pipe can be set on the GEOMETRY option.
General cross-section geometry for beam elements can be defined using the BEAM SECT parameter.

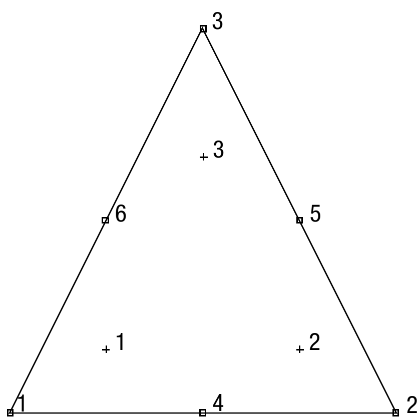
Class 14

Elements 124, 125, 126, 128, 129, 131, 132, 197, 200, 231, and 232

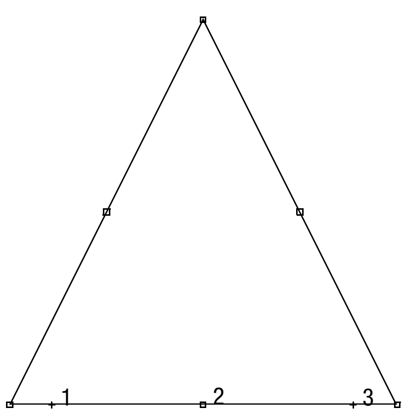
This class consists of six-noded triangular elements with quadratic interpolation functions as shown below.



The node numbering must be counterclockwise in the plane of the element. The functions are expressed with respect to area coordinate systems. The stiffness and mass matrix is integrated using three integration points as shown below.



The distributed loads are integrated using three point integration along the edge as shown below.

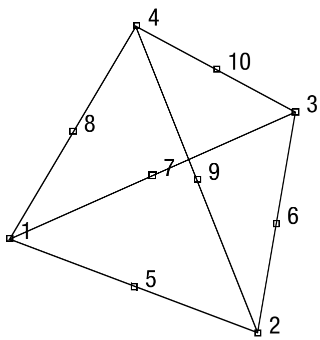


When using the Herrmann formulation, elements 128 or 129, the corner points contain an additional degree of freedom.

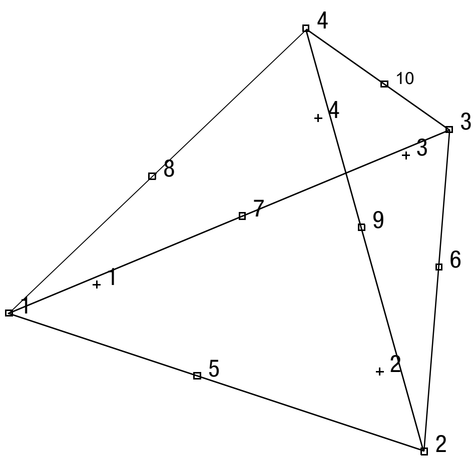
Class 15

Elements 127, 130, 133, 233 and 243

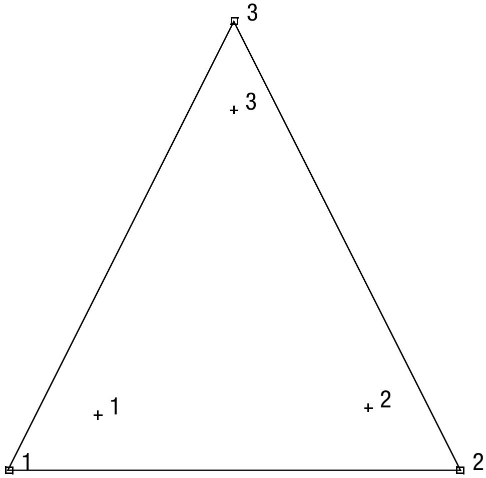
This class consists of ten-noded tetrahedral elements with quadratic interpolation functions as shown below.



The interpolation functions are expressed with respect to area coordinate systems. The stiffness matrix is integrated using four integration points as shown below. The mass matrix is integrated with sixteen integration points.



The distributed loads on a face are integrated using three integration points as shown below.

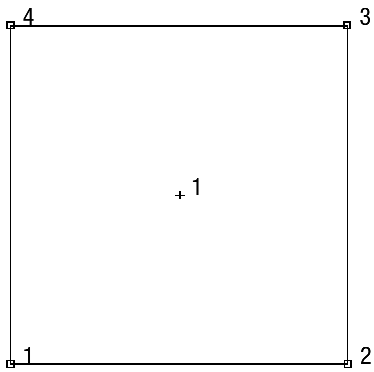


When using the Herrmann formulation, element 130, the corner points contain an additional degree of freedom.

Class 16

Elements 114, 115, 116, 118, 119, 121, and 122

This class consists of four-noded isoparametric planar elements written with respect to the natural coordinate system of the element. They are reduced integration elements, but an additional contribution has been made to the stiffness matrix to eliminate the problems associated with hourglass modes. The node numbering must be counterclockwise in the plane, as shown below, with a single integration point for the stiffness matrix.



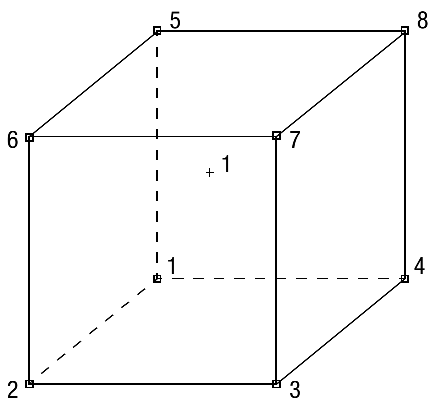
The element uses conventional 2 x 2 integration for the mass matrix and two-point integration along the edge for distributed loads.

For elements 118 and 119, there is one extra node with a single degree of freedom (pressure). These elements use a mixed formulation for incompressible analysis.

Class 17

Elements 117, 120, and 123

This class consists of eight-noded isoparametric elements written with respect to the natural coordinate system of the element. They are reduced integration elements, but an additional contribution has been made to the stiffness matrix to eliminate the problems with hourglass modes. Nodes 1, 2, 3, and 4 are corner nodes of one face given in counterclockwise order when viewed from inside the element. As shown below, node 5 is on the same edge as node 1; node 6 as node 2; node 7 as node 3; and node 8 as node 4.



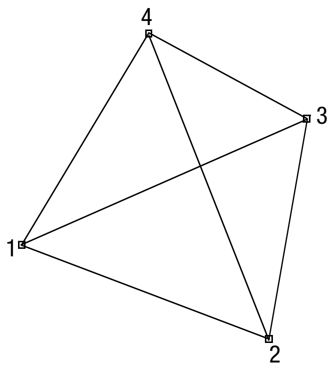
A single integration point is used for the stiffness matrix. The element uses conventional 2 x 2 x 2 integration for the mass matrix and 2 x 2 integration on a face for distributed loads.

For element 120, there is an additional node with a single degree of freedom (pressure). This element uses a mixed formulation for incompressible analysis.

Class 18

Elements 134, 135, 164 and 247

This class consists of four-noded tetrahedral elements with linear interpolation functions as shown below:



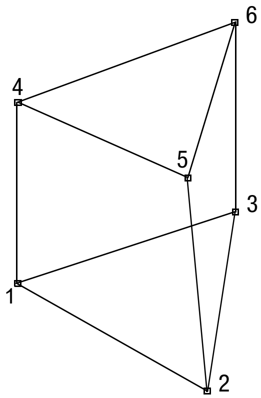
The interpolation functions are expressed with respect to the area coordinate system. The stiffness matrix is integrated using a single integration point at the centroid. The distributed load on a face is integrated using a single integration point at the centroid of the face.

Note that this element gives very poor behavior when used for incompressible or nearly incompressible behavior, such as plasticity.

Class 19

Elements 136, 137, 204, and 237

This class consists of six-node pentahedral elements with linear interpolation functions as shown below:

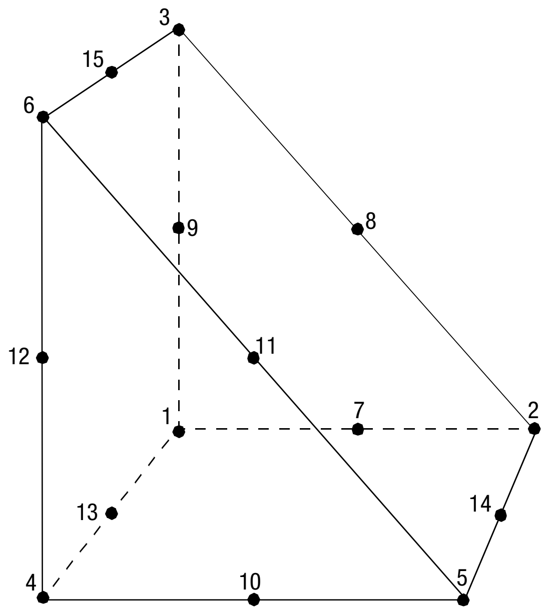


For volumetric integration, a six-point Gaussian scheme is used. For distributed loads on the faces, a 2 x 2 integration procedure is used on the quadrilateral faces, and a three -point scheme is used on the triangular faces.

Class 20

Elements 202, 203, 205, and 238

This class consists of fifteen-node pentahedral elements with quadratic interpolation functions as shown below:

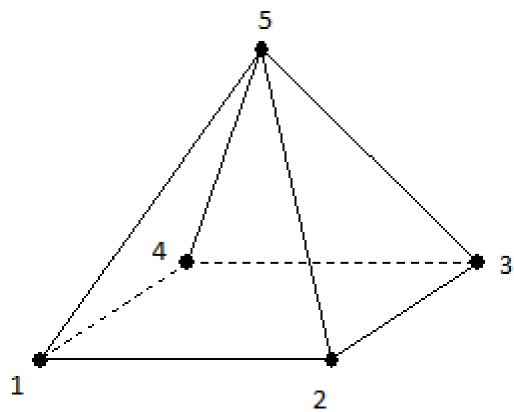


For volumetric integration, a twenty-one point Gaussian scheme is used. For distributed loads on the faces, a 3 x 3 integration procedure is used on the quadrilateral faces, and a seven -point scheme is used on the triangular faces.

Class 21

Elements 216 and 217

This class consists of five-node pyramid elements with linear interpolation functions as shown below:

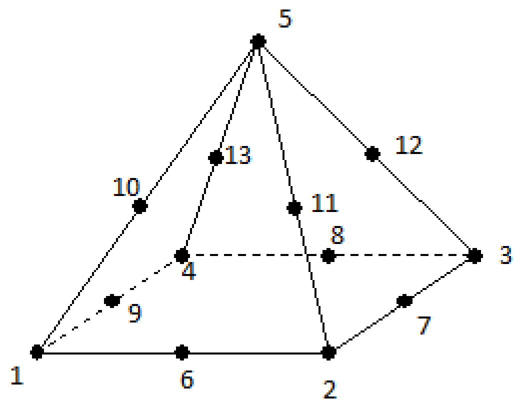


For volumetric integration, a five-point Gaussian scheme is used. For distributed loads on the faces, a 2 x 2 integration procedure is used on the quadrilateral faces, and a three -point scheme is used on the triangular faces.

Class 22

Elements 218 and 219

This class consists of thirteen-node pyramid elements with quadratic interpolation functions as shown below:



For volumetric integration, an eight point Gaussian scheme is used. For distributed loads on the faces, a 3 x 3 integration procedure is used on the quadrilateral faces, and a seven-point scheme is used on the triangular faces.

Special Elements

Elements 4, 8, 12, 22, 23, 24, 31, 45, 46, 47, 48, 49, 68, 72, 75, 76, 77, 78, 79, 85, 86, 87, 88, 89, 90, 91, 92, 93, 94, 95, 96, 97, 98, 99, 100, 101, 102, 103, 104, 105, 106, 107, 108, 109, 110, 138, 139, 140, 142, 143, 144, 145, 146, 147, 148, 149, 150, 151, 152, 153, 154, 155, 156, 157, 165, 166, 167, 168, 169, 170, 171, 172, 173, 174, 175, 176, 177, 178, 179, 180, 181, 182, 183, 185, 186, 187, 188, 189, 190, 191, 192, 193, 194, 195, 220, 221, 222, 223, 224, 225, 226, 227, 239, 240, 241, 244, 245, 246 and 247

These special elements do not belong to any of the 22 element classes described in this chapter. These elements include Beam and Shell Elements (4, 8, 22, 24, 31, 49, 72, 75-79, 85-90, 98, and 138-140), Friction and Gap Elements (12 and 97), Rebar Elements (23, 46-48, 142-148, and 165 -170), Cavity Surface Elements (171-174), Elastic Shear Panel (68), Semi-infinite Elements (91-94 and 101-108), Axisymmetric Elements with Bending (95 and 96), Composite Elements (149-154 and 175-180), Lower order Triangular and Tetrahedral Elements with a cubic bubble function (155-157), Triangular and Tetrahedral Elements with Strain Smoothing (239, 240, and 241), Magnetostatic Elements (109, 110, 181, 182and 183), Interface Elements (186, 187, 188, 189, 190, 191, 192, and 193), Generalized Spring-Damper Elements (194, 195), Heat Transfer Interface Elements (220, 221, 222, 223, 224, 225, 226, and 227) and Spring/ Dashpot Elements (244, 245, and 246). See Chapter 3: Element Library for detailed information on these elements.

3 Element Library

- Element 1 Straight Axisymmetric Shell 110
- Element 2 Axisymmetric Triangular Ring 113
- Element 3 Plane Stress Quadrilateral 116
- Element 4 Curved Quadrilateral, Thin-shell Element 120
- Element 5 Beam Column 125
- Element 6 Two-dimensional Plane Strain Triangle 129
- Element 7 Three-dimensional Arbitrarily Distorted Brick 132
- Element 8 Curved Triangular Shell Element 138
- Element 9 Three-dimensional Truss 144
- Element 10 Arbitrary Quadrilateral Axisymmetric Ring 147
- Element 11 Arbitrary Quadrilateral Plane-strain 151
- Element 12 Friction And Gap Link Element 155
- Element 13 Open Section Thin-Walled Beam 160
- Element 14 Thin-walled Beam in Three Dimensions without Warping 165
- Element 15 Axisymmetric Shell, Isoparametric Formulation 171

- Element 16 Curved Beam in Two-dimensions, Isoparametric Formulation 175
- Element 17 Constant Bending, Three-node Elbow Element 179
- Element 18 Four-node, Isoparametric Membrane 187
- Element 19 Generalized Plane Strain Quadrilateral 191
- Element 20 Axisymmetric Torsional Quadrilateral 196
- Element 21 Three-dimensional 20-node Brick 200
- Element 22 Quadratic Thick Shell Element 206
- Element 23 Three-dimensional 20-node Rebar Element 211
- Element 24 Curved Quadrilateral Shell Element 213
- Element 25 Thin-walled Beam in Three-dimensions 219
- Element 26 Plane Stress, Eight-node Distorted Quadrilateral 224
- Element 27 Plane Strain, Eight-node Distorted Quadrilateral 228
- Element 28 Axisymmetric, Eight-node Distorted Quadrilateral 232
- Element 29 Generalized Plane Strain, Distorted Quadrilateral 236
- Element 30 Membrane, Eight-node Distorted Quadrilateral 241
- Element 31 Elastic Curved Pipe (Elbow)/Straight Beam 245
- Element 32 Plane Strain Eight-node Distorted Quadrilateral, Herrmann Formulation 249
- Element 33 Axisymmetric, Eight-node Distorted Quadrilateral, Herrmann Formulation 254
- Element 34 Generalized Plane Strain Distorted Quadrilateral, Herrmann Formulation 259
- Element 35 Three-dimensional 20-node Brick, Herrmann Formulation 264

- Element 36 Three-dimensional Link (Heat Transfer Element) 270
- Element 37 Arbitrary Planar Triangle (Heat Transfer Element) 272
- Element 38 Arbitrary Axisymmetric Triangle (Heat Transfer Element) 274
- Element 39 Bilinear Planar Quadrilateral (Heat Transfer Element) 276
- Element 40 Axisymmetric Bilinear Quadrilateral Element (Heat Transfer Element) 279
- Element 41 Eight-node Planar Biquadratic Quadrilateral (Heat Transfer Element) 282
- Element 42 Eight-node Axisymmetric Biquadratic Quadrilateral (Heat Transfer Element) 285
- Element 43 Three-dimensional Eight-node Brick (Heat Transfer Element) 288
- Element 44 Three-dimensional 20-node Brick (Heat Transfer Element) 292
- Element 45 Curved Timoshenko Beam in a Plane 296
- Element 46 Eight-node Plane Strain Rebar Element 300
- Element 47 Generalized Plane Strain Rebar Element 302
- Element 48 Eight-node Axisymmetric Rebar Element 304
- Element 49 Finite Rotation Linear Thin Shell Element 307
- Element 50 Three-node Linear Heat Transfer Shell Element 312
- Element 51 Cable Element 316
- Element 52 Elastic or Inelastic Beam 319
- Element 53 Plane Stress, Eight-node Distorted Quadrilateral with Reduced Integration 324
- Element 54 Plane Strain, Eight-node Distorted Quadrilateral with Reduced Integration 329

- Element 55 Axisymmetric, Eight-node Distorted Quadrilateral with Reduced Integration 333
- Element 56 Generalized Plane Strain, Distorted Quadrilateral with Reduced Integration 338
- Element 57 Three-dimensional 20-node Brick with Reduced Integration 343
- Element 58 Plane Strain Eight-node Distorted Quadrilateral with Reduced Integration Herrmann Formulation 350
- Element 59 Axisymmetric, Eight-node Distorted Quadrilateral with Reduced Integration, Herrmann Formulation 355
- Element 60 Generalized Plane Strain Distorted Quadrilateral with Reduced Integration, Herrmann Formulation 360
- Element 61 Three-dimensional, 20-node Brick with Reduced Integration, Herrmann Formulation 366
- Element 62 Axisymmetric, Eight-node Quadrilateral for Arbitrary Loading (Fourier) 372
- Element 63 Axisymmetric, Eight-node Distorted Quadrilateral for Arbitrary Loading, Herrmann Formulation (Fourier) 376
- Element 64 Isoparametric, Three-node Truss 380
- Element 65 Heat Transfer Element, Three-node Link 383
- Element 66 Eight-node Axisymmetric Herrmann Quadrilateral with Twist 385
- Element 67 Eight-node Axisymmetric Quadrilateral with Twist 390
- Element 68 Elastic, Four-node Shear Panel 394
- Element 69 Eight-node Planar Biquadratic Quadrilateral with Reduced Integration (Heat Transfer Element) 396

- Element 70 Eight-node Axisymmetric Biquadrilateral with Reduced Integration (Heat Transfer Element) 399
- Element 71 Three-dimensional 20-node Brick with Reduced Integration (Heat Transfer Element) 402
- Element 72 Bilinear Constrained Shell Element 406
- Element 73 Axisymmetric, Eight-node Quadrilateral for Arbitrary Loading with Reduced Integration (Fourier) 412
- Element 74 Axisymmetric, Eight-node Distorted Quadrilateral for Arbitrary Loading, Herrmann Formulation, with Reduced Integration (Fourier) 416
- Element 75 Bilinear Thick-shell Element 420
- Element 76 Thin-walled Beam in Three Dimensions without Warping 425
- Element 77 Thin-walled Beam in Three Dimensions including Warping 431
- Element 78 Thin-walled Beam in Three Dimensions without Warping 436
- Element 79 Thin-walled Beam in Three Dimensions including Warping 441
- Element 80 Arbitrary Quadrilateral Plane Strain, Herrmann Formulation 446
- Element 81 Generalized Plane Strain Quadrilateral, Herrmann Formulation 450
- Element 82 Arbitrary Quadrilateral Axisymmetric Ring, Herrmann Formulation 455
- Element 83 Axisymmetric Torsional Quadrilateral, Herrmann Formulation 459
- Element 84 Three-dimensional Arbitrarily Distorted Brick, Herrmann Formulation 463
- Element 85 Four-node Bilinear Shell (Heat Transfer Element) 469

- Element 86 Eight-node Curved Shell (Heat Transfer Element) 474
- Element 87 Three-node Axisymmetric Shell (Heat Transfer Element) 479
- Element 88 Two-node Axisymmetric Shell (Heat Transfer Element) 483
- Element 89 Thick Curved Axisymmetric Shell 487
- Element 90 Thick Curved Axisymmetric Shell – for Arbitrary Loading (Fourier) 491
- Element 91 Linear Plane Strain Semi-infinite Element 495
- Element 92 Linear Axisymmetric Semi-infinite Element 498
- Element 93 Quadratic Plane Strain Semi-infinite Element 501
- Element 94 Quadratic Axisymmetric Semi-infinite Element 504
- Element 95 Axisymmetric Quadrilateral with Bending 507
- Element 96 Axisymmetric, Eight-node Distorted Quadrilateral with Bending 512
- Element 97 Special Gap and Friction Link for Bending 517
- Element 98 Elastic or Inelastic Beam with Transverse Shear 519
- Element 99 Three-dimensional Link (Heat Transfer Element) 525
- Element 100 Three-dimensional Link (Heat Transfer Element) 527
- Element 101 Six-node Plane Semi-infinite Heat Transfer Element 529
- Element 102 Six-node Axisymmetric Semi-infinite Heat Transfer Element 532

- Element 103 Nine-node Planar Semi-infinite Heat Transfer Element 535
- Element 104 Nine-node Axisymmetric Semi-infinite Heat Transfer Element 538
- Element 105 Twelve-node 3-D Semi-infinite Heat Transfer Element 541
- Element 106 Twenty-seven-node 3-D Semi-infinite Heat Transfer Element 544
- Element 107 Twelve-node 3-D Semi-infinite Stress Element 547
- Element 108 Twenty-seven-node 3-D Semi-infinite Stress Element 550
- Element 109 Eight-node 3-D Magnetostatic Element 553
- Element 110 Twelve-node 3-D Semi-infinite Magnetostatic Element 557
- Element 111 Arbitrary Quadrilateral Planar Magnetodynamic 560
- Element 112 Arbitrary Quadrilateral Axisymmetric Magnetodynamic Ring 564
- Element 113 Three-dimensional Magnetodynamic Arbitrarily Distorted Brick 568
- Element 114 Plane Stress Quadrilateral, Reduced Integration 572
- Element 115 Arbitrary Quadrilateral Plane Strain, Reduced Integration 576
- Element 116 Arbitrary Quadrilateral Axisymmetric Ring, Reduced Integration 580
- Element 117 Three-dimensional Arbitrarily Distorted Brick, Reduced Integration 584
- Element 118 Arbitrary Quadrilateral Plane Strain, Incompressible Formulation with Reduced Integration 589
- Element 119 Arbitrary Quadrilateral Axisymmetric Ring, Incompressible Formulation with Reduced Integration 593

- Element 120 Three-dimensional Arbitrarily Distorted Brick, Incompressible Reduced Integration 597
- Element 121 Planar Bilinear Quadrilateral, Reduced Integration (Heat Transfer Element) 602
- Element 122 Axisymmetric Bilinear Quadrilateral, Reduced Integration (Heat Transfer Element) 605
- Element 123 Three-dimensional Eight-node Brick, Reduced Integration (Heat Transfer Element) 608
- Element 124 Plane Stress, Six-node Distorted Triangle 611
- Element 125 Plane Strain, Six-node Distorted Triangle 615
- Element 126 Axisymmetric, Six-node Distorted Triangle 619
- Element 127 Three-dimensional Ten-node Tetrahedron 623
- Element 128 Plane Strain, Six-node Distorted Triangle, Herrmann Formulation 627
- Element 129 Axisymmetric, Six-node Distorted Triangle, Herrmann Formulation 631
- Element 130 Three-dimensional Ten-node Tetrahedron, Herrmann Formulation 635
- Element 131 Planar, Six-node Distorted Triangle (Heat Transfer Element) 640
- Element 132 Axisymmetric, Six-node Distorted Triangle (Heat Transfer Element) 643
- Element 133 Three-dimensional Ten-node Tetrahedron (Heat Transfer Element) 646
- Element 134 Three-dimensional Four-node Tetrahedron 649
- Element 135 Three-dimensional Four-node Tetrahedron (Heat Transfer Element) 653
- Element 136 Three-dimensional Arbitrarily Distorted Pentahedral 656
- Element 137 Three-dimensional Six-node Pentahedral (Heat Transfer Element) 660

- Element 138 Bilinear Thin-triangular Shell Element 663
- Element 139 Bilinear Thin-shell Element 668
- Element 140 Bilinear Thick-shell Element with One-point Quadrature 673
- Element 141 Heat Transfer Shell 679
- Element 142 Eight-node Axisymmetric Rebar Element with Twist 680
- Element 143 Four-node Plane Strain Rebar Element 683
- Element 144 Four-node Axisymmetric Rebar Element 685
- Element 145 Four-node Axisymmetric Rebar Element with Twist 688
- Element 146 Three-dimensional 8-node Rebar Element 691
- Element 147 Four-node Rebar Membrane 693
- Element 148 Eight-node Rebar Membrane 695
- Element 149 Three-dimensional, Eight-node Composite Brick Element 697
- Element 150 Three-dimensional, Twenty-node Composite Brick Element 703
- Element 151 Quadrilateral, Plane Strain, Four-node Composite Element 709
- Element 152 Quadrilateral, Axisymmetric, Four-node Composite Element 713
- Element 153 Quadrilateral, Plane Strain, Eight-node Composite Element 717
- Element 154 Quadrilateral, Axisymmetric, Eight-node Composite Element 722
- Element 155 Plane Strain, Low-order, Triangular Element, Herrmann Formulations 727
- Element 156 Axisymmetric, Low-order, Triangular Element, Herrmann Formulations 731

- Element 157 Three-dimensional, Low-order, Tetrahedron, Herrmann Formulations 735
- Element 158 Three-node Membrane Element 739
- Element 159 Four-node, Thick Shell Element 742
- Element 160 Arbitrary Plane Stress Piezoelectric Quadrilateral 743
- Element 161 Arbitrary Plane Strain Piezoelectric Quadrilateral 748
- Element 162 Arbitrary Quadrilateral Piezoelectric Axisymmetric Ring 752
- Element 163 Three-dimensional Piezoelectric Arbitrary Distorted Brick 756
- Element 164 Three-dimensional Four-node Piezo-Electric Tetrahedron 762
- Element 165 Two-node Plane Strain Rebar Membrane Element 766
- Element 166 Two-node Axisymmetric Rebar Membrane Element 768
- Element 167 Two-node Axisymmetric Rebar Membrane Element with Twist 770
- Element 168 Three-node Plane Strain Rebar Membrane Element 772
- Element 169 Three-node Axisymmetric Rebar Membrane Element 774
- Element 170 Three-node Axisymmetric Rebar Membrane Element with Twist 776
- Element 171 Two-node 2-D Cavity Surface Element 778
- Element 172 Two-node Axisymmetric Cavity Surface Element 780
- Element 173 Three-node 3-D Cavity Surface Element 782
- Element 174 Four-node 3-D Cavity Surface Element 784
- Element 175 Three-dimensional, Eight-node Composite Brick Element (Heat Transfer Element) 786

- Element 176 Three-dimensional, Twenty-node Composite Brick Element (Heat Transfer Element) 790
- Element 177 Quadrilateral Planar Four-node Composite Element (Heat Transfer Element) 793
- Element 178 Quadrilateral, Axisymmetric, Four-node Composite Element (Heat Transfer Element) 796
- Element 179 Quadrilateral, Planar Eight-node Composite Element (Heat Transfer Element) 799
- Element 180 Quadrilateral, Axisymmetric, Eight-node Composite Element (Heat Transfer Element) 802
- Element 181 Three-dimensional Four-node Magnetostatic Tetrahedron 805
- Element 182 Three-dimensional Ten-node Magnetostatic Tetrahedron 808
- Element 183 Three-dimensional Magnetostatic Current Carrying Wire 811
- Element 184 Three-dimensional Ten-node Tetrahedron 813
- Element 185 Three-dimensional Eight-node Solid Shell, Selective Reduced Integration 817
- Element 186 Four-node Planar Interface Element 823
- Element 187 Eight-node Planar Interface Element 826
- Element 188 Eight-node Three-dimensional Interface Element 829
- Element 189 Twenty-node Three-dimensional Interface Element
- Element 190 Four-node Axisymmetric Interface Element 837
- Element 191 Eight-node Axisymmetric Interface Element
- Element 192 Six-node Three-dimensional Interface Element 843
- Element 193 Fifteen-node Three-dimensional Interface Element 847
- Element 194 2-D Generalized Spring - CBUSH/CFAST Element 851
- Element 195 3-D Generalized Spring - CBUSH/CFAST Element 854

- Element 196 Three-node, Bilinear Heat Transfer Membrane 858
- Element 197 Six-node, Biquadratic Heat Transfer Membrane 861
- Element 198 Four-node, Isoparametric Heat Transfer Element 864
- Element 199 Eight-node, Biquadratic Heat Transfer Membrane 868
- Element 200 Six-node, Biquadratic Isoparametric Membrane 872
- Element 201 Two-dimensional Plane Stress Triangle 875
- Element 202 Three-dimensional Fifteen-node Pentahedral 878
- Element 203 Three-dimensional Fifteen-node Pentahedral (Heat Transfer Element) 882
- Element 204 Three-dimensional Magnetostatic Pentahedral 885
- Element 205 Three-dimensional Fifteen-node Magnetostatic Pentahedral 888
- Element 206 Twenty-node 3-D Magnetostatic Element 892
- Element 207 Not Available 895
- Element 208 Not Available 896
- Element 209 Not Available 897
- Element 210 Not Available 898
- Element 211 Not Available 899
- Element 212 Not Available 900
- Element 213 Not Available 901
- Element 214 Not Available 902
- Element 215 Not Available 903
- Element 216 Three-dimensional Five-node Pyramid 904
- Element 217 Three-dimensional Five-node Pyramid (Heat Transfer Element) 908
- Element 218 Three-dimensional Thirteen-node Pyramid 911

- Element 219 Three-dimensional Thirteen-node Pyramid (Heat Transfer Element) 915
- Element 220 Four-node Planar Heat Transfer Interface Element 918
- Element 221 Eight-node Planar Heat Transfer Interface Element 921
- Element 222 Eight-node Three-dimensional Heat Transfer Interface Element 924
- Element 223 Twenty-node Three-dimensional Heat Transfer Interface Element 927
- Element 224 Four-node Axisymmetric Heat Transfer Interface Element 930
- Element 225 Eight-node Axisymmetric Heat Transfer Interface Element 933
- Element 226 Six-node Three-dimensional Heat Transfer Interface Element 936
- Element 227 Fifteen-node Three-dimensional Heat Transfer Interface Element 939
- Element 228 Arbitrary Triangle Planar Magnetodynamic 942
- Element 229 Arbitrary Triangle Axisymmetric Magnetodynamic Ring 945
- Element 230 Three-dimensional Magnetodynamic Arbitrarily Distorted Tetrahedral 949
- Element 231 Arbitrary Triangle Planar Magnetodynamic 952
- Element 232 Arbitrary Triangle Axisymmetric Magnetodynamic Ring 956
- Element 233 Three-dimensional Magnetodynamic Arbitrarily Distorted Tetrahedral 959
- Element 234 Arbitrary Quadrilateral Planar Magnetodynamic 963
- Element 235 Arbitrary Quadrilateral Axisymmetric Magnetodynamic Ring 967
- Element 236 Three-dimensional Magnetodynamic Arbitrarily Distorted Brick 971

- Element 237 Three-dimensional Magnetodynamic Arbitrarily Distorted Pentahedral 976
- Element 238 Three-dimensional Magnetodynamic Arbitrarily Distorted Pentahedral 980
- Element 239 Two-dimensional Plane Strain Triangle with Strain Smoothing 984
- Element 240 Axisymmetric Triangular Ring with Strain Smoothing 988
- Element 241 Three-dimensional Four-node Tetrahedron with Strain Smoothing 992
- Element 242 Three-dimensional, 20 node Piezoelectric Arbitrary Distorted Brick 996
- Element 243 Three-dimensional, 10 node Piezoelectric Tetrahedron 1002
- Element 244 2-D Planar Spring/Dashpot Element 1007
- Element 245 Axisymmetric Spring/Dashpot Element 1009
- Element 246 3-D Spring/Dashpot Element 1011
- Element 247 Three-dimensional, Low-order, Tetrahedron, Herrmann Formulation 1013

The remainder of this volume describes each element type. The geometric information that you are required to input is described here. The output associated with each element type is also discussed.

Element 1

Straight Axisymmetric Shell

This is a two-node, axisymmetric, thick-shell element with a linear displacement assumption based on the global displacements and rotation. The strain-displacement relationships used are suitable for large displacements and large membrane strains. One-point Gaussian integration is used for the stiffness and two-point Gaussian integration is used for the mass and pressure determination. All constitutive models can be used with this element.

Element types 15 and 89 are more accurate elements.

Quick Reference

Type 1

Axisymmetric, straight, thick-shell element.

Connectivity

Two nodes per element (see Figure 3-1).

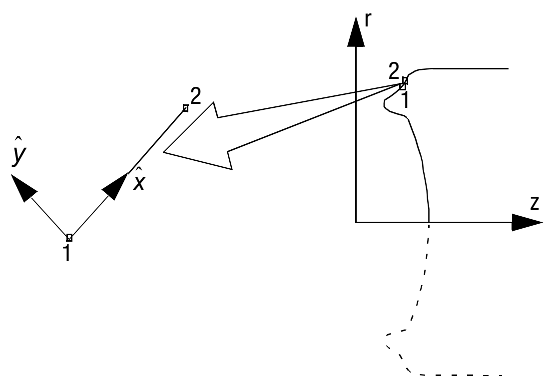


Figure 3-1 Two-node, Axisymmetric Shell Element

Geometry

Constant thickness along length of the element. Thickness of the element is stored in the first data field (EGEOM1).

If the element needs to be offset from the user-specified position, the eighth data field (EGEOM8) is set to -2.0. In this case, an extra line containing the offset information is provided (see the GEOMETRY option in Marc Volume C: Program Input). The offset magnitudes at the first and second nodes are taken to be along the element normal and are provided in the first and second data fields of the extra line respectively. A uniform offset for the element can be set by providing the offset magnitude in the first data field and then setting the constant offset flag (sixth data field of the extra line) to 1.

Coordinates

1 = z
2 = r

Degrees of Freedom

1 = u = axial (parallel to symmetry axis)
2 = v = radial (normal to symmetry axis)
3 = ϕ = right hand rotation

Tractions

Distributed loads. Selected with IBODY as follows:

Load Type	Description
0	Uniform pressure.
1	Nonuniform pressure.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in the z, r-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Pressure assumed positive in the direction opposite to the normal.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the center line of the element is as follows:

1 = meridional membrane
2 = circumferential membrane
3 = transverse shear
4 = meridional curvature
5 = circumferential curvature

Output of Stresses

The stresses are given at the center line of the shell element and at the integration points through the thickness as follows:

1 = meridional stress
2 = circumferential stress
3 = transverse shear

The integration point numbering sequence progresses in the positive local $\hat{y}$ directions.

Transformation

The displacement degrees of freedom can be transformed to local directions.

Tying

Standard type 23 with elements 2 and 10.

Output Points

Centroid of the element.

Section Stress Integration

Simpson's rule is used to integrate through the thickness. Use the SHELL SECT parameter to specify the number of layers.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – output of stress and strain in meridional and circumferential direction. Thickness is updated.

Note:	Shell theory only applies if strain variation through the thickness is small.

Note, however, that since the curvature calculation is linearized, you have to select your load steps such that the rotations remain small within a load step.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 88. See Element 88 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered a design variable for this element type.

Element 2

Axisymmetric Triangular Ring

Element type 2 is a three-node, isoparametric, triangular element. It is written for axisymmetric applications and uses bilinear interpolation functions. The strains are constant throughout the element and this results in a poor representation of shear behavior.

In general, one needs more of these lower-order elements than the higher-order elements such as type 124. Hence, use a fine mesh.

The stiffness of this element is formed using one-point integration at the centroid. The mass matrix of this element is formed using four-point Gaussian integration.

This element should not be used in cases where incompressible or nearly incompressible behavior occurs because it will lock. This includes rubber elasticity, large strain plasticity and creep. For such problems, use element type 156 instead. Or, the cross-triangle approach can be used.

Quick Reference

Type 2

Axisymmetric triangular ring element.

Connectivity

Three nodes per element (see Figure 3-2). Node numbering must counterclockwise.

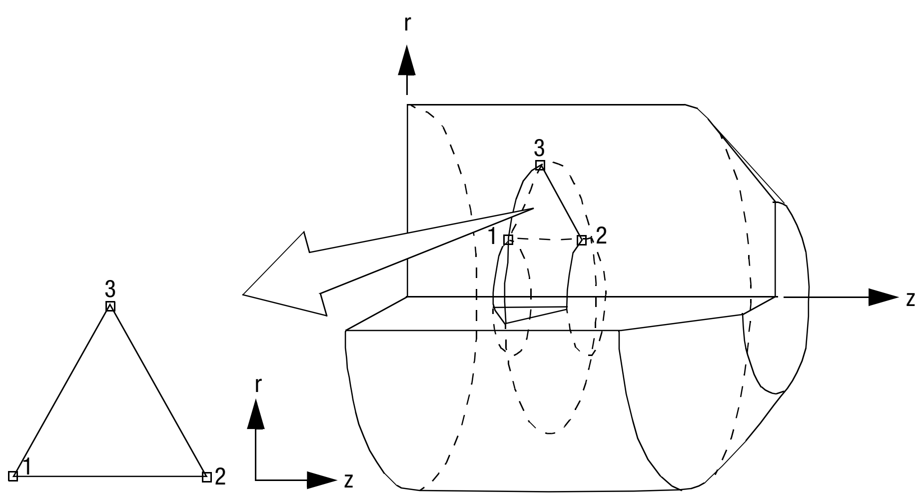


Figure 3-2 Linear Displacement Triangular Ring Element

Geometry

Not required for this element.

Coordinates

1 = z
2 = r

Degrees of Freedom

1 = u = axial (parallel to symmetry axis)
2 = v = radial (normal to symmetry axis)

Tractions

Load types for distributed loads are as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force per unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element; magnitude supplied through the FORCEM user subroutine.
4	Nonuniform body force per unit volume in first coordinate direction; magnitude supplied through the FORCEM user subroutine.
5	Nonuniform body force per unit volume in second coordinate direction; magnitude supplied through the FORCEM user subroutine.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element; magnitude supplied through the FORCEM user subroutine.
8	Uniform pressure on 3-1 face of the element.
9	Nonuniform pressure on 3-1 face of the element; magnitude supplied through the FORCEM user subroutine.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in the x-, y-, and z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Load Type	Description
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

All pressures are positive when directed into the element. In addition, point loads can be applied at the nodes.

Concentrated loads applied at the nodes must be the value of the load integrated around the circumference.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid of the element is as follows:

$1 = \varepsilon_{zz}$

$2 = \varepsilon_{rr}$

$3 = \varepsilon_{\theta\theta}$

$4 = \gamma_{zr}$

Output of Stresses

Output of stresses is also at the centroid of the element and follows the same scheme as Output of Strains .

Transformation

Two global degrees of freedom can be transformed to local coordinates. In this case, the corresponding applied nodal loads should also be in the local direction.

Tying

Can be tied to axisymmetric shell type 1 by typing type 23.

Output Points

Output is only available at the centroid. Element mesh must generally be fine for accuracy.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available. Output of stress and strain in global coordinate directions. “Crossed triangle” approach recommended.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 38. See Element 38 for a description of the conventions used for entering the flux and film data for this element.

Element 3

Plane Stress Quadrilateral

Element 3 is a four-node, isoparametric, arbitrary quadrilateral written for plane stress applications. As this element uses bilinear interpolation functions, the strains tend to be constant throughout the element. This results in a poor representation of shear behavior. The shear (or bending) characteristics can be improved by using alternative interpolation functions. This assumed strain procedure is flagged through the GEOMETRY option.

In general, one needs more of these lower-order elements than the higher-order elements such as 26 or 53. Hence, use a fine mesh.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using four-point Gaussian integration.

All constitutive models can be used with this element.

Note:	To improve the bending characteristics of the element, the interpolation functions are modified for the assumed strain formulation.

Quick Reference

Type 3

Plane stress quadrilateral.

Connectivity

Node numbering must be counterclockwise (see Figure 3-3).

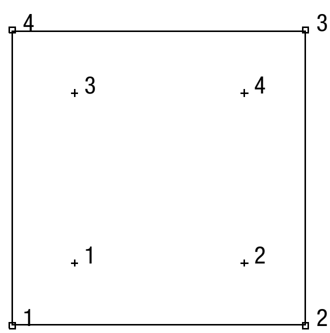


Figure 3-3 Plane Stress Quadrilateral

Geometry

The thickness is stored in the first data field (`EGEOM1`). Default thickness is one.

The second field is not used.

In the third field, a one activates the assumed strain formulation.

Coordinates

Two global coordinates x and y.

Degrees of Freedom

1 = u (displacement in the global x direction)
2 = v (displacement in the global y direction)

Distributed Loads

Load types for distributed loads are defined as follows:

Load Type	Description
* 0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force per unit volume in second coordinate direction.
* 3	Nonuniform pressure on 1-2 face of the element; magnitude supplied through the FORCEM user subroutine.
4	Nonuniform body force per unit volume in first coordinate direction; magnitude supplied through the FORCEM user subroutine.
5	Nonuniform body force per unit volume in second coordinate direction; magnitude supplied through the FORCEM user subroutine.
* 6	Uniform pressure on 2-3 face of the element.
* 7	Nonuniform pressure on 2-3 face of the element; magnitude supplied through the FORCEM user subroutine.
* 8	Uniform pressure on 3-4 face of the element.
* 9	Nonuniform pressure on 3-4 face of the element; magnitude supplied through the FORCEM user subroutine.
* 10	Uniform pressure on 4-1 face of the element.
* 11	Nonuniform pressure on 4-1 face of the element; magnitude supplied through the FORCEM user subroutine.
* 20	Uniform shear force on side 1-2 (positive from 1 to 2).
* 21	Nonuniform shear force on side 1-2; magnitude supplied through user the FORCEM user subroutine.
* 22	Uniform shear force on side 2-3 (positive from 2 to 3).
* 23	Nonuniform shear force on side 2-3; magnitude supplied through the FORCEM user subroutine.
* 24	Uniform shear force on side 3-4 (positive from 3 to 4).
* 25	Nonuniform shear force on side 3-4; magnitude supplied through the FORCEM user subroutine.
* 26	Uniform shear force on side 4-1 (positive from 4 to 1).
* 27	Nonuniform shear force on side 4-1; magnitude supplied through the FORCEM user subroutine.

Load Type	Description
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter two magnitudes: the first value is gravity acceleration in x-direction; the second is gravity acceleration in the y-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

All pressures are positive when directed into the element. Load types shown with an asterisk (*) require the magnitude of the load to be entered as force per unit area. To prescribe these loads in force per unit length, add 50 to the load type. This is often useful in design optimization where the thickness changes, but it is desired that the applied force remain the same. In addition, point loads can be applied at the nodes.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid of the element is as follows:

$1 = \varepsilon_{xx}$

$2 = \varepsilon_{yy}$

$3 = \gamma_{xy}$

Note:	Although $\varepsilon_{zz} = \frac{-\nu}{E}(\sigma_{xx} + \sigma_{yy})$, it is not printed and is posted as 0 for isotropic materials. For Mooney or Ogden (TL formulation) Marc post code 49 provides the thickness strain for plane stress elements. See Marc Volume A: Theory and User Information , Chapter 12 Output Results, Element Information for von Mises intensity calculation for strain.

Output of Stresses

Output of Stresses is the same as for the Output of Strains .

Transformation

Two global degrees of freedom can be transformed to local coordinates.

Tying

Use UFORMSN user subroutine.

Output Points

Output is available at the centroid or at the four numerical integration points shown in Figure 3-3.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available. Output of true stress and logarithmic strain in global coordinate directions. Thickness is updated if LARGE STRAIN is specified.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 39. See Element 39 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux due to dissipation of plastic work specified with type 101.

Assumed Strain

The assumed strain formulation is available to improve the in-plane bending behavior. This increases the stiffness assembly costs per element and improves the accuracy.

Design Variables

The thickness can be considered a design variable for this element.

Element 4

Curved Quadrilateral, Thin-shell Element

This is an isoparametric, doubly-curved thin-shell element using bicubic interpolation functions. The element is based in Koiter-Sanders shell theory, fulfilling continuity requirements, and represents rigid body modes exactly when used as a rectangle in the mapped surface coordinate plane. The element contains no patching functions, so that it is restricted to quadrilateral meshes with a maximum of four elements sharing one common node. However, the element is rapidly convergent in most problems which allow such a mesh. Note that any suitable surface coordinate systems can be chosen, so that the mesh need not be rectangular on the actual surface. This element cannot be used with CONTACT.

Geometry

The element is isoparametric, so that the actual surface is interpolated from nodal coordinates. The mesh is defined in the $\theta^1 - \theta^2$ plane of surface coordinates. Then, the actual surface is approximated with a surface defined by cubic interpolation on the interior of each element based on the following set of 14 nodal coordinates:

$$\theta^1,\ \theta^2,\ x,\ \frac{\partial x}{\partial\theta^1},\ \frac{\partial x}{\partial\theta^2},\ y,\ \frac{\partial y}{\partial\theta^1},\ \frac{\partial y}{\partial\theta^2},\ z,\ \frac{\partial z}{\partial\theta^1},\ \frac{\partial z}{\partial\theta^2},\ \frac{\partial^2 x}{\partial\theta^1\partial\theta^2},\ \frac{\partial^2 y}{\partial\theta^1\partial\theta^2},\ \frac{\partial^2 z}{\partial\theta^1\partial\theta^2}$$

In most practical cases, the surface is definable as follows:

$$\begin{aligned} x &= x(\theta^1, \theta^2) \\ y &= y(\theta^1, \theta^2) \\ z &= z(\theta^1, \theta^2) \end{aligned}$$

Then, the usual procedure is to define the mesh in the $\theta^1 - \theta^2$ plane (as a rectangular mesh) by supplying the first two coordinates (θ^1, θ^2) at each node through the COORDINATES input option. Then, the remaining 12 coordinates are defined at each node through the use of the FXORD option (see Marc Volume A: Theory and User Information) or the UFXORD user subroutine (see Marc Volume D: User Subroutines and Special Routines).

The element can have variable thickness since Marc allows linear variation of the thickness with respect to θ^1 and θ^2. Note, however, you should ensure that the thickness is continuous from one element to the next; otherwise, the tying option must be used to uncouple the membrane strain. In a continuous mesh, continuity of all membrane strain components is assumed.

Displacement

There are the following 12 degrees of freedom at each node:

$$u,\ \frac{\partial u}{\partial\theta^1},\ \frac{\partial u}{\partial\theta^2},\ v,\ \frac{\partial v}{\partial\theta^1},\ \frac{\partial v}{\partial\theta^2},\ w,\ \frac{\partial w}{\partial\theta^1},\ \frac{\partial w}{\partial\theta^2},\ \frac{\partial^2 u}{\partial\theta^1\partial\theta^2},\ \frac{\partial^2 v}{\partial\theta^1\partial\theta^2},\ \frac{\partial^2 w}{\partial\theta^1\partial\theta^2}$$

where u, v, w are the Cartesian components of displacement.

Displacement is interpolated by complete bicubic functions on the interior of an element, so that equality of the above nodal degrees of freedom at the coincident nodes of abutting elements ensures the necessary continuity required for thin shell theory.

Note that fixed displacement boundary conditions should never be associated with all 12 degrees of freedom at each node, since three degrees of freedom must always determine middle surface (membrane) strains at the node. Care must be exercised in the specification of kinematic boundary conditions. They must be fully specified, but not over specified. The application of moments and torsional springs must consider the dimensions of the generalized coordinates so that the forces and conjugate displacements multiply together to yield mechanical work.

Connectivity Specification and Numerical Integration

The nodal point numbers of the element must be given in counterclockwise order on the θ^1 - θ^2 plane, starting with the point i (min θ^1 and θ^2). Thus, in Figure 3-4, the connectivity must be given as i, j, k, l.

The element is integrated numerically using nine points (Gaussian quadrature). The first integration point is always closest to the first node of the element; then, the integration points are numbered as shown in Figure 3-4.

Point 5 (centroid of the element in the θ^1 and θ^2 plane) is used for stress output if the CENTROID parameter is not flagged. The CENTROID parameter should be used for any nonlinear analysis with this element.

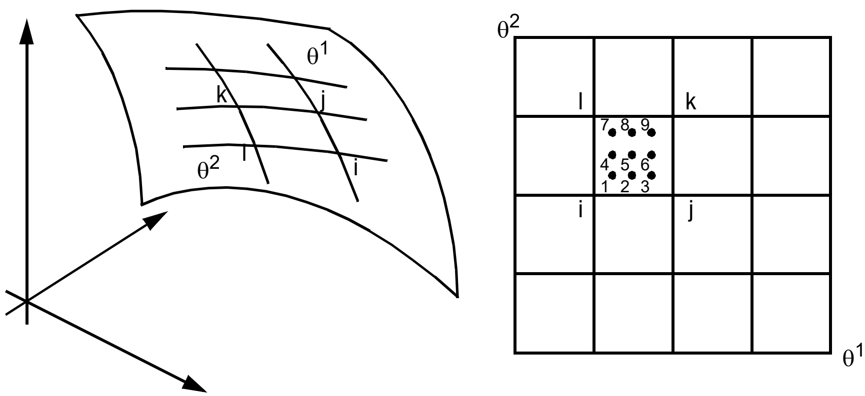


Figure 3-4 Form of Element 4

Quick Reference

Type 4

Doubly curved quadrilateral thin-shell element.

Connectivity

Four nodes per element. The first node given in the connectivity list must have minimum θ^1 and θ^2.

Geometry

Thickness at the element centroid is input in the first data field (EGEOM1).

Rate of change of thickness with respect to $\theta^1 \left(\frac{\partial t}{\partial \theta^1}\right)$ is in the second data field (EGEOM2)

Rate of change of thickness with respect to $\theta^2 \left(\frac{\partial t}{\partial \theta^2}\right)$ is in the third data field (EGEOM3).

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

Coordinates

The geometry is defined by 14 nodal coordinates. The surface must be rectangular in the θ_1 and θ_2 plane. The required 14 nodal coordinates are as follows:

$$\theta_1,\ \theta_2,\ x,\ \frac{\partial x}{\partial \theta_1},\ \frac{\partial x}{\partial \theta_2},\ y,\ \frac{\partial y}{\partial \theta_1},\ \frac{\partial y}{\partial \theta_2},\ z,\ \frac{\partial z}{\partial \theta_1},\ \frac{\partial z}{\partial \theta_2},\ \frac{\partial^2 x}{\partial \theta_1 \partial \theta_2},\ \frac{\partial^2 y}{\partial \theta_1 \partial \theta_2},\ \frac{\partial^2 z}{\partial \theta_1 \partial \theta_2}$$

Usually, the FXORD option or the UFXORD user subroutine can be used to minimize the coordinate input.

Degrees of Freedom

There are 12 degrees of freedom per node as follows:

$$u,\ \frac{\partial u}{\partial \theta^1},\ \frac{\partial u}{\partial \theta^2},\ v,\ \frac{\partial v}{\partial \theta^1},\ \frac{\partial v}{\partial \theta^2},\ w,\ \frac{\partial w}{\partial \theta^1},\ \frac{\partial w}{\partial \theta^2},\ \frac{\partial^2 u}{\partial \theta^1 \partial \theta^2},\ \frac{\partial^2 v}{\partial \theta^1 \partial \theta^2},\ \frac{\partial^2 w}{\partial \theta^1 \partial \theta^2}$$

Tractions

Distributed loadings are as follows:

Load Type	Description
1	Uniform self-weight per unit surface area in -z-direction.
2	Uniform pressure.
3	Nonuniform pressure; magnitude supplied by the FORCEM user subroutine.
4	Nonuniform load per unit volume in arbitrary direction; magnitude and direction supplied in the FORCEM user subroutine.
11	Uniform load per unit length on the 1-2 edge in the x-direction.
12	Uniform load per unit length on the 1-2 edge in the y-direction.
13	Uniform load per unit length on the 1-2 edge in the z-direction.
21	Uniform load per unit length on the 2-3 edge in the x-direction.
22	Uniform load per unit length on the 2-3 edge in the y-direction.
23	Uniform load per unit length on the 2-3 edge in the z-direction.
31	Uniform load per unit length on the 3-4 edge in the x-direction.
32	Uniform load per unit length on the 3-4 edge in the y-direction.

Load Type	Description
33	Uniform load per unit length on the 3-4 edge in the z-direction.
41	Uniform load per unit length on the 4-1 edge in the x-direction.
42	Uniform load per unit length on the 4-1 edge in the y-direction.
43	Uniform load per unit length on the 4-1 edge in the z-direction.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in the x-, y-, and z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Components of stretch and curvature in surface coordinate directions.

Output of Stresses

Stress output as physical components of σ^{11}, σ^{22}, σ^{12} in surface coordinate system at points equally spaced through the thickness with the first point on the surface in the direction of the positive normal.

Transformation

Cartesian displacement components and their derivatives can be transformed to a local system. The surface coordinate system is not affected by this transformation.

Special Transformation

The shell transformation option type 4 can be used to permit easier application of point loads, moments and/or boundary conditions on a node. For a description of this transformation type, see Volume A: Theory and User Information. Note that if the FOLLOW FOR parameter is invoked, the transformations are based on the updated configuration of the element.

Tying

Tying type 18 can be used for intersecting shells. Tying type 19 can be used for beam stiffened shells using element 13 as a stiffener on this element.

Output Points

Centroid or nine Gaussian integration points, if the CENTROID or ALL POINTS parameter is used as shown in Figure 3-4.

Note:	The element is sensitive to boundary conditions. Ensure that every required boundary condition is properly applied.

Section Stress Integration

Integration through the thickness is performed numerically using Simpson's rule. Use the SHELL SECT parameter to define the number of integration points.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available. Output of stress and strain as for total Lagrangian approach. Thickness is updated.

Note: Shell theory only applies if strain variation through the thickness is small.

Large Deformation Analysis

The large deformation analysis allows either the Lagrangian or updated Lagrangian description used in Marc. In the present version, however, only large deflection terms corresponding to the stretching strains have been introduced. This approximation is usually acceptable even for nonlinear buckling analysis.

Coupled Analysis

Not available for this element. Use either element types 22, 72, 75, or 139.

Element 5

Beam Column

This element is a straight, two-node, rectangular-section, beam-column element using linear interpolation parallel to its length, and cubic interpolation in the normal direction. (Element 16 is a full cubic beam that is more accurate for many applications.)

The degrees of freedom are the u and v displacements, and the right-handed rotation at the two end points of the element. The strain-stress transformation is formed by a Simpson's rule integration using the number of points defined in the SHELL SECT parameter through the thickness of the element. The stiffness is formed by Gaussian integration along the length of the element using three points. All constitutive relations can be used with this element type.

Quick Reference

Type 5

Two-dimensional, rectangular-section beam-column.

Connectivity

Two nodes per element (see Figure 3-5).

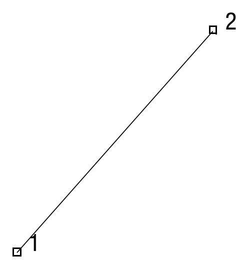


Figure 3-5 Beam-Column Element

Geometry

A rectangular section is assumed. The height is input in the first data field (EGEOM1). The cross-sectional area is input in the second data field (EGEOM2). The third data field is not used.

Beam Offsets Pin Codes and Local Beam Orientation

If an offset is applied to the beam geometry or pin codes are applied to the degrees of freedom of the end point of the beams or the beam axis is defined with respect to a local coordinate system, then a code is placed in the 8th field of the 3rd data block of the GEOMETRY option (EGEOM8). This code is formed by the negative sum of ioffset, iorien, and ipin where:

ioffset	=	0	-	no offset
		1	-	beam offset
iorien	=	0	-	conventional definition of local beam orientation; beam axis given in 4th through 6th field with respect to global system

		10	-	the local beam orientation entered in the 4th through 6th field is given with respect to the coordinate system of the first beam node. This vector is transformed to the global coordinate system before any further processing.
ipin	=	0	-	no pin codes are used
		10 0	-	pin codes are used.

If ioffset = 1, an additional line is read in the GEOMETRY option (4th data block).

If ipin = 100, an additional line is read in the GEOMETRY option (5th data block)

The offset vector at the first corner node is provided in the first three fields and the offset vector at the second corner node is provided in the next three fields of the extra line. The coordinate system used for the offset vectors at the two nodes is indicated by the corresponding flags in the eighth and ninth fields of the extra line. A flag setting of 0 indicates global coordinate system, 1 indicates local element coordinate system and 3 indicates local nodal coordinate system. When the flag setting is 1, the local beam coordinate system is constructed by taking the local x axis along the beam, local z axis as [0,0,1] and local y axis as perpendicular to both.

Another way to input a pin code is by using the PIN CODE option in the model definition. This option is supported by Mentat while the input using the GEOMETRY option is not.

Coordinates

Two coordinates in the global x and y directions.

Degrees of Freedom

1 = u displacement
2 = v displacement
3 = right-handed rotation

Tractions

Distributed loads according to the value of IBODY are as follows:

Load Type	Description
0	Uniform pressure force/unit length.
1	Nonuniform pressure force/unit length.
11	Fluid drag/buoyancy loading – fluid behavior specified in the FLUID DRAG option.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter the magnitudes of gravity acceleration in the x- and y-direction.

Load Type	Description
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

In the nonuniform case, the load magnitude must be specified by the FORCEM user subroutine.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Two generalized strains, membrane stretching and bending.

Output of Stresses

Stresses at each integration point or the centroid. Each of these points has points equally spaced through its thickness. The stress is output at each representative point.

The first point is on the positive normal (positive local y) face. The last point is on the negative face.

Transformation

Standard transformation transforms first two global degrees of freedom to local degrees of freedom.

Tying

No standard tying available. Use the UFORMSN user subroutine.

Output Points

Centroid or three Gaussian integration points.

Section Stress Integration

Integration through the thickness is performed numerically using Simpson's rule. Use the SHELL SECT parameter to define the number of integration points.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available. Use element type 16 instead.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 36. See Element 36 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The height and/or the cross-sectional area can be considered as design variables.

Element 6

Two-dimensional Plane Strain Triangle

Element 6 is a three-node, isoparametric, triangular element written for plane strain applications. This element uses bilinear interpolation functions. The strains are constant throughout the element. This results in a poor representation of shear behavior.

In general, you need more of these lower-order elements than the higher-order elements such as element type 125. Hence, use a fine mesh.

The stiffness of this element is formed using one point integration at the centroid. The mass matrix of this element is formed using four-point Gaussian integration.

This element should not be used in cases where incompressible or nearly incompressible behavior occurs because it will lock. This includes rubber elasticity, large strain plasticity and creep. For such problems, use element type 155 instead. Or, the cross-triangle approach can be used.

Quick Reference

Type 6

Two-dimensional, plane strain, three-node triangle.

Connectivity

Node numbering must be counterclockwise (see Figure 3-6).

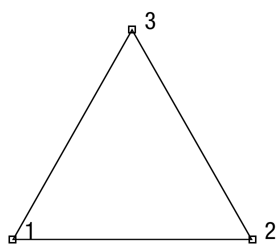


Figure 3-6 Plane-Strain Triangle

Geometry

Thickness stored in first data field (`EGOM1`). Default thickness is unity. Other fields are not used.

Coordinates

Two coordinates in the global x and y directions.

Degrees of Freedom

1 = u displacement
2 = v displacement

Tractions

Load types for distributed loads are as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force per unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element; magnitude supplied through the FORCEM user subroutine.
4	Nonuniform body force per unit volume in first coordinate direction; magnitude supplied through the FORCEM user subroutine.
5	Nonuniform body force per unit volume in second coordinate direction; magnitude supplied through the FORCEM user subroutine.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element; magnitude supplied through the FORCEM user subroutine.
8	Uniform pressure on 3-1 face of the element.
9	Nonuniform pressure on 3-1 face of the element; magnitude supplied through the FORCEM user subroutine.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in the x-, y-, and z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid of the element is as follows:

$1 = \varepsilon_{xx}$

$2 = \varepsilon_{yy}$

$3 = \varepsilon_{zz} = 0$

$4 = \gamma_{xy}$

Output of Stresses

Same as Output of Strains .

Transformation

Nodal degrees of freedom can be transformed to local degrees of freedom. Corresponding nodal loads must be applied in local direction.

Tying

Use the UFORMSN user subroutine.

Output Points

Only available at the centroid. Element must be small for accuracy.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available. Output of stress and strain in global coordinate directions. “Crossed triangle” approach recommended.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 37. See Element 37 for a description of the conventions used for entering the flux and film data for this element.

Element 7

Three-dimensional Arbitrarily Distorted Brick

Element type 7 is an eight-node, isoparametric, arbitrary hexahedral. As this element uses trilinear interpolation functions, the strains tend to be constant throughout the element. This results in a poor representation of shear behavior. The shear (or bending) characteristics can be improved by using alternative interpolation functions. This assumed strain procedure is flagged through the GEOMETRY option.

In general, you need more of these lower-order elements than the higher-order elements such as types 21 or 57. Hence, use a fine mesh.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using eight-point Gaussian integration.

For nearly incompressible behavior, including plasticity or creep, it is advantageous to use an alternative integration procedure. This constant dilatation method which eliminates potential element locking is flagged through the GEOMETRY option.

This element can be used for all constitutive relations. When using incompressible rubber materials (for example, Mooney and Ogden), the element must be used within the Updated Lagrange framework.

For rubber materials with total Lagrange procedure, element type 84 can be used. This is slightly more expensive because of the extra pressure degrees of freedom associated with element type 84.

Notes:	For the assumed strain formulation, the interpolation functions are modified to improve the bending characteristics of the element.
	As in all three-dimensional analyses, a large nodal bandwidth results in long computing times. Optimize the nodal bandwidth.

Quick Reference

Type 7

Three-dimensional, eight-node, first-order, isoparametric element (arbitrarily distorted brick).

Connectivity

Eight nodes per element. Node numbering must follow the scheme below (see Figure 3-7):

> Nodes 1, 2, 3, and 4 are corners of one face, given in counterclockwise order when viewed from inside the element. Node 5 has the same edge as node 1. Node 6 has the same edge as node 2. Node 7 has the same edge as node 3. Node 8 has the same edge as node 4.

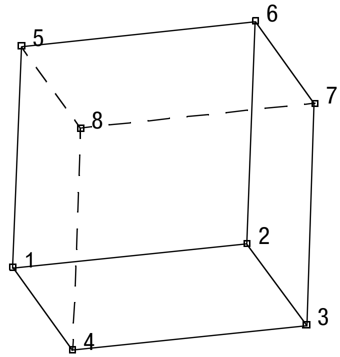


Figure 3-7 Arbitrarily Distorted Cube

Geometry

If the automatic brick to shell constraints are to be used, the first field must contain the transition thickness (see Figure 3-8). Note that in a coupled analysis, there are no constraints for the temperature degrees of freedom.

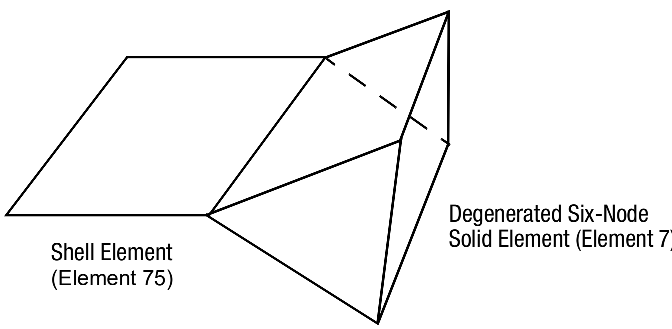


Figure 3-8 Shell-to-Solid Automatic Constraint

If a nonzero value is entered in the second data field (`EGEOM2`), the volumetric strain is constant throughout the element. This is particularly useful for analysis of approximately incompressible materials, and for analysis of structures in the fully plastic range. It is also recommended for creep problems in which it is attempted to obtain the steady state solution.

If a one is placed in the third field, the assumed strain formulation is activated.

Coordinates

Three coordinates in the global x-, y-, and z-directions.

Degrees of Freedom

Three global degrees of freedom u, v, and w per node.

Distributed Loads

Distributed loads chosen by value of IBODY are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3-4 face.
1	Nonuniform pressure on 1-2-3-4 face; magnitude supplied through the FORCEM user subroutine.
2	Uniform body force per unit volume in -z-direction.
3	Nonuniform body force per unit volume (e.g., centrifugal force); magnitude and direction supplied through the FORCEM user subroutine.
4	Uniform pressure on 6-5-8-7 face.
5	Nonuniform pressure on 6-5-8-7 face (FORCEM user subroutine).
6	Uniform pressure on 2-1-5-6 face.
7	Nonuniform pressure on 2-1-5-6 face (FORCEM user subroutine).
8	Uniform pressure on 3-2-6-7 face.
9	Nonuniform pressure on 3-2-6-7 face (FORCEM user subroutine).
10	Uniform pressure on 4-3-7-8 face.
11	Nonuniform pressure on 4-3-7-8 face (FORCEM user subroutine).
12	Uniform pressure on 1-4-8-5 face.
13	Nonuniform pressure on 1-4-8-5 face (FORCEM user subroutine).
20	Uniform pressure on 1-2-3-4 face.
21	Nonuniform load on 1-2-3-4 face; magnitude and direction supplied in the FORCEM user subroutine.
22	Uniform body force per unit volume in -z-direction.
23	Nonuniform body force per unit volume (e.g., centrifugal force); magnitude and direction supplied through the FORCEM user subroutine.
24	Uniform pressure on 6-5-8-7 face.
25	Nonuniform load on 6-5-8-7 face; magnitude and direction supplied in the FORCEM user subroutine.
26	Uniform pressure on 2-1-5-6 face.
27	Nonuniform load on 2-1-5-6 face; magnitude and direction supplied in the FORCEM user subroutine.
28	Uniform pressure on 3-2-6-7 face.
29	Nonuniform load on 3-2-6-7 face; magnitude and direction supplied in the FORCEM user subroutine.
30	Uniform pressure on 4-3-7-8 face.
31	Nonuniform load on 4-3-7-8 face; magnitude and direction supplied in the FORCEM user subroutine.
32	Uniform pressure on 1-4-8-5 face.
33	Nonuniform load on 1-4-8-5 face; magnitude and direction supplied in the FORCEM user subroutine.
40	Uniform shear 1-2-3-4 face in the 1-2 direction.
41	Nonuniform shear 1-2-3-4 face in the 1-2 direction.
42	Uniform shear 1-2-3-4 face in the 2-3 direction.

Load Type	Description
43	Nonuniform shear 1-2-3-4 face in the 2-3 direction.
48	Uniform shear 6-5-8-7 face in the 5-6 direction.
49	Nonuniform shear 6-5-8-7 face in the 5-6 direction.
50	Uniform shear 6-5-8-7 face in the 6-7 direction.
51	Nonuniform shear 6-5-8-7 face in the 6-7 direction.
52	Uniform shear 2-1-5-6 face in the 1-2 direction.
53	Nonuniform shear 2-1-5-6 face in the 1-2 direction.
54	Uniform shear 2-1-5-6 face in the 1-5 direction.
55	Nonuniform shear 2-1-5-6 face in the 1-5 direction.
56	Uniform shear 3-2-6-7 face in the 2-3 direction.
57	Nonuniform shear 3-2-6-7 face in the 2-3 direction.
58	Uniform shear 3-2-6-7 face in the 2-6 direction.
59	Nonuniform shear 2-3-6-7 face in the 2-6 direction.
60	Uniform shear 4-3-7-8 face in the 3-4 direction.
61	Nonuniform shear 4-3-7-8 face in the 3-4 direction.
62	Uniform shear 4-3-7-8 face in the 3-7 direction.
63	Nonuniform shear 4-3-7-8 face in the 3-7 direction.
64	Uniform shear 1-4-8-5 face in the 4-1 direction.
65	Nonuniform shear 1-4-8-5 face in the 4-1 direction.
66	Uniform shear 1-4-8-5 in the 1-5 direction.
67	Nonuniform shear 1-4-8-5 face in the 1-5 direction.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in the x-, y-, and z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Pressure forces are positive into element face.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

$1 = \varepsilon_{xx}$

$2 = \varepsilon_{yy}$

$3 = \varepsilon_{zz}$

$4 = \gamma_{xy}$

$5 = \gamma_{yz}$

$6 = \gamma_{zx}$

Output of Stresses

Output of stresses is the same as for Output of Strains .

Transformation

Standard transformation of three global degrees of freedom to local degrees of freedom.

Tying

No special tying available. An automatic constraint is available for brick-to-shell transition meshes (see Geometry).

Output Points

Centroid or the eight integration points as shown in Figure 3-9.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 43. See Element 43 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux generated by dissipated plastic energy is specified with type 101.

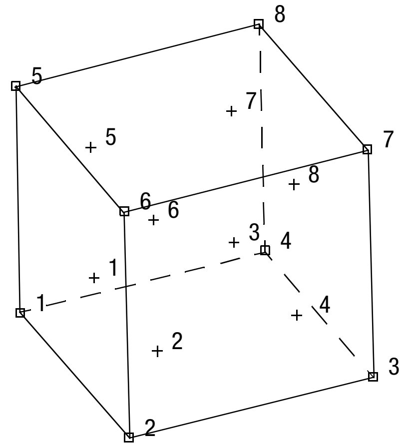


Figure 3-9 Eight-Point Gauss Integration Scheme for Element 7

Assumed Strain

The assumed strain formulation is available to improve the bending behavior. This increases the stiffness assembly costs per element, but it improves the accuracy.

Notes:	The element can be collapsed to a tetrahedron.
	By collapsing one plane of the element to a line (see Figure 3-8), a transition element for connecting bricks with four-node shell element type 75 is generated. Thickness of the shell must be specified in the geometry field of the brick element.

Element 8

Curved Triangular Shell Element

This element is an isoparametric, curved, triangular, thin-shell element based on the Koiter-Sanders shell theory, which fulfills continuity requirements and represents rigid-body motions exactly. This element cannot be used with the CONTACT option.

Geometry

The middle surface of the shell is defined by the equations:

$$
\begin{aligned}
x &= x(\theta^1, \theta^2) \\
y &= y(\theta^1, \theta^2) \\
z &= z(\theta^1, \theta^2)
\end{aligned}
\tag{3-1}
$$

where (x, y, and z) are Cartesian coordinates.

(θ^1, θ^2) denote Gaussian coordinates on the middle surface of the shell.

The domain of definition in the plane (θ^1, θ^2) is divided into a mesh of triangles which are mapped onto curved elements on the middle surface Σ. The actual middle surface is approximated by a smooth surface Σ which has the same coordinates (x-y-z) and the same tangent plane at each nodal point of the mesh. Practically, the mesh is defined by the Caussian coordinates (θ_i^1, θ_i^2) of the nodal points, and the surface Σ is defined by the values of the functions (Equation (3-1)) and their first derivatives at these points. According to the terminology of Marc, the coordinates are, therefore, the set:

$$
\begin{aligned}
&\theta_i^1,\ \theta_i^2,\ x(p_i),\ \partial x(p_i)/\partial\theta^1,\ \partial x(p_i)/\partial\theta^2 \\
&\qquad y(p_i),\ \partial y(p_i)/\partial\theta^1,\ \partial y(p_i)/\partial\theta^2 \\
&\qquad z(p_i),\ \partial z(p_i)/\partial\theta^1,\ \partial z(p_i)/\partial\theta^2
\end{aligned}
\tag{3-2}
$$

where $x(p_i)$ stand for $x(\theta_i^1, \theta_i^2)$, (θ_i^1, θ_i^2) being the coordinates of the node p_i.

In the general case, these 11 coordinates must be given. Particular shapes are available through the FXORD option (described in Marc Volume A: Theory and User Information). Often the UFXORD user subroutine can be used to generate the coordinates from a reduced set (see Marc Volume D: User Subroutines and Special Routines). The thickness of the shell can vary linearly in an element: the values at the three nodes are given in EGEOM1, EGEOM2, EGEOM3. If EGEOM2 or EGEOM3 is given as zero, that thickness is set equal to EGEOM1.

There are nine degrees of freedom for each nodal point p_i. These degrees of freedom are defined in terms of the Cartesian components of displacement u, v, and w, and rates of change with respect to the Gaussian coordinates:

$$u(p_i),\ \partial u(p_i)/\partial\theta^1,\ \partial u(p_i)/\partial\theta^2$$
$$v(p_i),\ \partial v(p_i)/\partial\theta^1,\ \partial v(p_i)/\partial\theta^2$$
$$w(p_i),\ \partial w(p_i)/\partial\theta^1,\ \partial w(p_i)/\partial\theta^2$$

The displacements within an element are defined by interpolation functions. These interpolation functions $\phi(\theta^1,\ \theta^2)$ are such that compatibility of displacements and their first derivatives is insured between adjacent elements. Hence, for an element whose vertices are the nodal points p_i, p_j, and p_k, the components u, v, w are defined as:

$$u(\theta^1,\ \theta^2) = (\underline{U}_i^T,\ \underline{U}_j^T,\ \underline{U}_k^T)\ .\ \phi(\theta^1,\ \theta^2)$$
$$v(\theta^1,\ \theta^2) = (\underline{V}_i^T,\ \underline{V}_j^T,\ \underline{V}_k^T)\ ,\ \phi(\theta^1,\ \theta^2)$$
$$w(\theta^1,\ \theta^2) = (\underline{W}_i^T,\ \underline{W}_j^T,\ \underline{W}_k^T)\ ,\ \phi(\theta^1,\ \theta^2)$$

Numerical Integration

For this element, seven integration points are used with an integration rule which is exact for all polynomials up to the fifth order. See Figure 3-10.

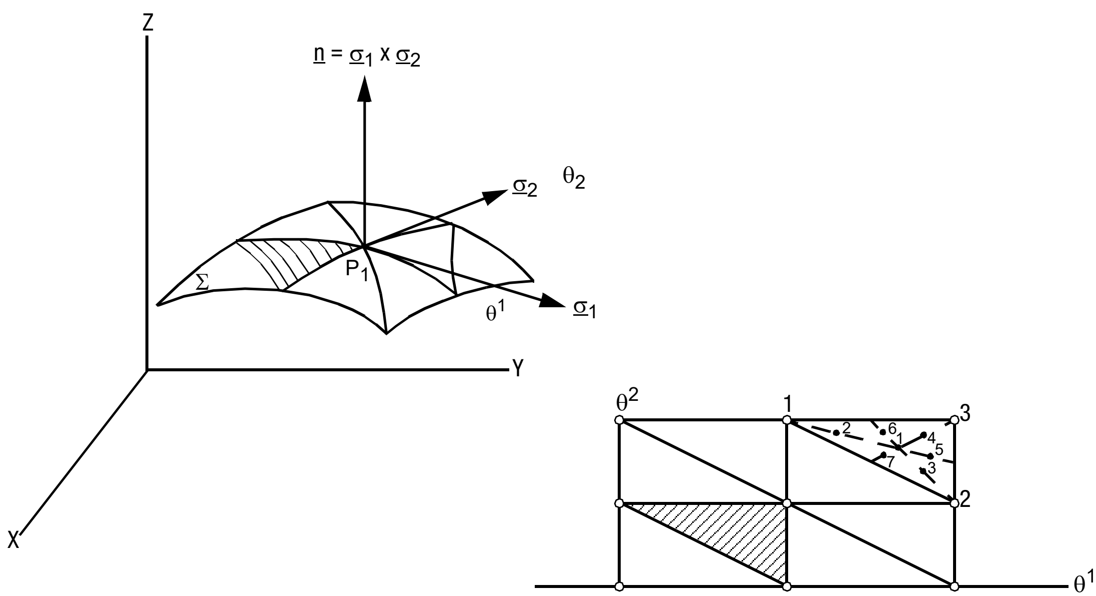


Figure 3-10 Uniform Loads On and Integration Points for Element 8

Quick Reference

Type 8

Arbitrary doubly curved triangular shell.

Connectivity

Three nodes per element. Numbering can be clockwise or counterclockwise for this element.

Geometry

The element can have linear variation of thickness in the θ^1, θ^2 plane. Thickness at the first node is input at the first data field (EGEOM1). Thickness at the second node in the second data field (EGEOM2). In the third data field (EGEOM3), thickness is at the third node. If only the first data field is used, the element defaults to a constant thickness.

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

Coordinates

The coordinates are defined by Gaussian coordinates (θ^1, θ^2) on the middle surface of the shell with 11 coordinates per node required in the general case:

$1 = \theta^1$	$5 = \dfrac{\partial x}{\partial \theta^2}$	$9 = z$
$2 = \theta^2$	$6 = y$	$10 = \dfrac{\partial z}{\partial \theta^1}$
$3 = x$	$7 = \dfrac{\partial y}{\partial \theta^1}$	$11 = \dfrac{\partial z}{\partial \theta^2}$
$4 = \dfrac{\partial x}{\partial \theta^1}$	$8 = \dfrac{\partial y}{\partial \theta^2}$	

Degrees of Freedom

Global displacement degrees of freedom:

$1 = u$ *displacement*	$4 = v$ *displacement*	$7 = w$ *displacement*

$2 = \dfrac{\partial u}{\partial \theta^1}$	$5 = \dfrac{\partial v}{\partial \theta^1}$	$8 = \dfrac{\partial w}{\partial \theta^1}$
$3 = \dfrac{\partial u}{\partial \theta^2}$	$6 = \dfrac{\partial v}{\partial \theta^2}$	$9 = \dfrac{\partial w}{\partial \theta^2}$

Tractions

Distributed loading types are as follows:

Load Type	Description
1	Uniform weight per surface area in the negative z-direction.
2	Uniform pressure; magnitude of pressure is positive when applied in negative normal vector direction.
3	Nonuniform pressure; magnitude given by the FORCEM user subroutine.
4	Nonuniform load per unit volume in arbitrary direction; magnitude and direction supplied in the FORCEM user subroutine.
11	Uniform load per unit length on the 1-2 edge in the x-direction.
12	Uniform load per unit length on the 1-2 edge in the y-direction.
13	Uniform load per unit length on the 1-2 edge in the z-direction.
21	Uniform load per unit length on the 2-3 edge in the x-direction.
22	Uniform load per unit length on the 2-3 edge in the y-direction.
23	Uniform load per unit length on the 2-3 edge in the z-direction.
31	Uniform load per unit length on the 3-1 edge in the x-direction.
32	Uniform load per unit length on the 3-1 edge in the y-direction.
33	Uniform load per unit length on the 3-1 edge in the z-direction.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in the x-, y-, and z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Load Type	Description
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Components of stretch and curvature in surface coordinate directions.

Output of Stresses

Physical components in surface coordinate directions at the points through the thickness: The first point on surface in the direction of positive normal; last point on surface in the direction of negative normal. Stress components are physical components in the θ^1, θ^2 directions:

$$1 = \sigma^{11}$$
$$2 = \sigma^{22}$$
$$3 = \sigma^{12}$$

Transformation

Cartesian displacement components and their derivatives can be transformed to a local system. The surface coordinate system is not affected by this transformation.

Special Transformation

The shell transformation option type 2 can be used to permit easier application of point loads, moments and/or boundary conditions of a node. For a description of the transformation type, see Marc Volume A: Theory and User Information . Note that if the FOLLOW FOR parameter is invoked, the transformation is based on the updated configuration of the element.

Tying

Typing type 18 is used for shell intersection. Tying type 19, 20, and 21 are provided for tying beam type 13 as a stiffener on this shell element.

Output Points

Centroid or seven integration points. These are located in the θ^1, θ^2 plane as shown in Figure 3-10.

Note: These results are sensitive to correct boundary condition specifications.

Section Stress Integration

Use SHELL SECT parameter to set number of points for Simpson rule integration through the thickness.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – output of stress and strain as for total Lagrangian approach. Thickness is updated.

Note:	Shell theory only applies if strain variation through the thickness is small.

Large Deformation Analysis

The large deformation analysis allows either the Lagrangian or updated Lagrangian description used in Marc. In the present version, however, only large deflection terms corresponding to the stretching strains have been introduced. This approximation is usually acceptable even for nonlinear buckling analysis.

Element 9

Three-dimensional Truss

Element type 9 is a simple linear straight truss with constant cross section. The strain-displacement relations are written for large strain, large displacement analysis. All constitutive relations can be used with this element. This element can be used as an actuator in mechanism analyses.

The stiffness matrix of this element is formed using a single point integration. The mass matrix is formed using two-point Gaussian integration.

Note:	This element has no bending stiffness.

Quick Reference

Type 9

Two- or three-dimensional, two-node, straight truss. Used by itself or in conjunction with any 3-D element, this element has three coordinates and three degrees of freedom. Otherwise, it has two coordinates and two degrees of freedom.

Connectivity

Two nodes per element (see Figure 3-11).

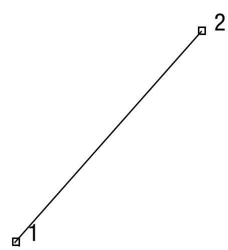


Figure 3-11 Two-node Truss

Geometry

The cross-sectional area is input in the first data field (`EGEOM1`). Default area is equal to 1.0. The second and the third data fields are not used. If the element is used as an actuator, then for table driven input the fourth field is used to define the multiplication factor for the table specified on the ACTUATOR model definition option. For non table driven input, the fourth field is used to define the initial length of the actuator. The ACTUATOR history definition option, or the UACTUAT user subroutine can then be used to modify the length of the actuator.

If beam-to-beam contact is switched on (see CONTACT option), the radius used when the element comes in contact with other beam or truss elements must be entered in the 7th data field (`EGEOM7`).

Coordinates

Three coordinates per node in the global x-, y-, and z-direction.

Degrees of Freedom

Global displacement degrees of freedom:

1 = u displacement
2 = v displacement
3 = w displacement (optional)

Tractions

Distributed loads according to the value of IBODY are as follows:

Load Type	Description
0	Uniform load (force per unit length) in the direction of the global x-axis.
1	Uniform load (force per unit length) in the direction of the global y-axis.
2	Uniform load (force per unit length) in the direction of the global z-axis.
11	Fluid drag/buoyancy loading – fluid behavior specified in the FLUID DRAG option.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter two magnitudes of gravity acceleration in the x- and y-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Uniaxial in the truss member.

Output of Stresses

Uniaxial in the truss member.

Transformation

The three global degrees of freedom for any node can be transformed to local degrees of freedom.

Tying

Use the UFORMSN user subroutine.

Output Points

Only one integration point available.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output as for total Lagrangian approach. Cross section is updated.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 36. See Element 36 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The cross-sectional area can be considered a design variable for this element.

Element 10

Arbitrary Quadrilateral Axisymmetric Ring

Element type 10 is a four-node, isoparametric, arbitrary quadrilateral written for axisymmetric applications. As this element uses bilinear interpolation functions, the strains tend to be constant throughout the element. This results in a poor representation of shear behavior.

In general, you need more of these lower-order elements than the higher-order elements such as types 28 or 55. Hence, use a fine mesh.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using four-point Gaussian integration.

For nearly incompressible behavior, including plasticity or creep, it is advantageous to use an alternative integration procedure. This constant dilatation method which eliminates potential element locking is flagged through the GEOMETRY option.

This element can be used for all constitutive relations. When using incompressible rubber materials (for example, Mooney and Ogden), the element must be used within the Updated Lagrange framework.

For rubber materials with total Lagrange procedure, element type 82 can be used. This is slightly more expensive because of the extra pressure degrees of freedom associated with element type 82.

Quick Reference

Type 10

Axisymmetric, arbitrary ring with a quadrilateral cross section.

Connectivity

Four nodes per element. Node numbering must be counterclockwise (see Figure 5-12).

Geometry

If a nonzero value is entered in the second data field (`EGEOM2`), the volume strain is constant throughout the element. This is particularly useful for analysis of approximately incompressible materials, and for analysis of structures in the fully plastic range. It is also recommended for creep problems in which it is attempted to obtain the steady-state solution.

Coordinates

Two coordinates in the global z- and r-direction.

Degrees of Freedom

1 = u (displacement in the global z-direction)
2 = v (displacement in the global r-direction).

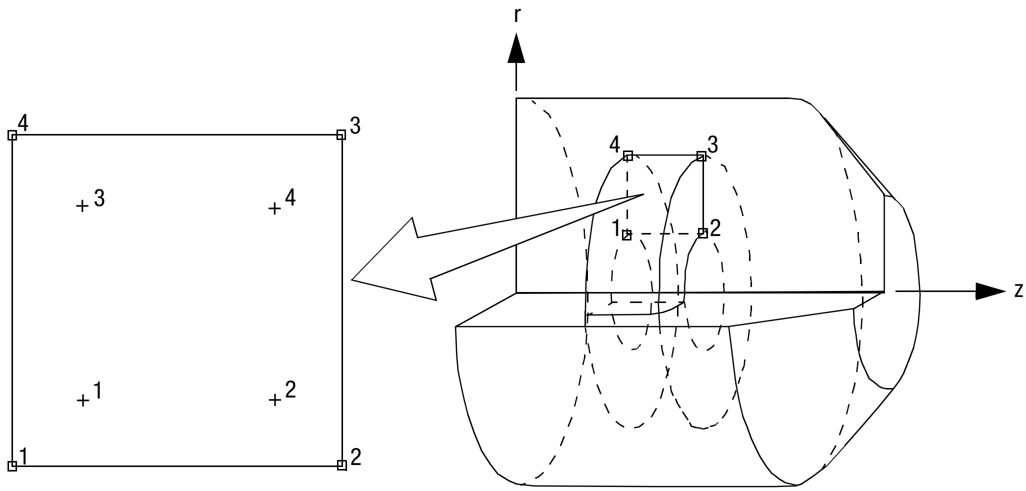


Figure 5-12 Integration Points for Element 10

Distributed Loads

Load types for distributed loads are as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force by unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element; magnitude supplied through the FORCEM user subroutine.
4	Nonuniform body force per unit volume in first coordinate direction; magnitude supplied through the FORCEM user subroutine.
5	Nonuniform body force per unit volume in second coordinate direction; magnitude supplied through the FORCEM user subroutine.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element; magnitude supplied through the FORCEM user subroutine.
8	Uniform pressure on 3-4 face of the element.
9	Nonuniform pressure on 3-4 face of the element; magnitude supplied through the FORCEM user subroutine.
10	Uniform pressure on 4-1 face of the element.
11	Nonuniform pressure on 4-1 face of the element; magnitude supplied through the FORCEM user subroutine.

Load Type	Description
20	Uniform shear force on side 1-2 (positive from 1 to 2).
21	Nonuniform shear force on side 1-2; magnitude supplied through the FORCEM. user subroutine
22	Uniform shear force on side 2-3 (positive from 2 to 3).
23	Nonuniform shear force on side 2-3; magnitude supplied through the FORCEM user subroutine.
24	Uniform shear force on side 3-4 (positive from 3 to 4).
25	Nonuniform shear force on side 3-4; magnitude supplied through the FORCEM user subroutine.
26	Uniform shear force on side 4-1 (positive from 4 to 1).
27	Nonuniform shear force on side 4-1; magnitude supplied through the FORCEM user subroutine.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in the x-, y-, and z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

All pressures are positive when directed into the element. In addition, point loads can be applied at the nodes. The magnitude of point loads must correspond to the load integrated around the circumference.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid of the element in global coordinates is:

$1 = \varepsilon_{zz}$

$2 = \varepsilon_{rr}$

$3 = \varepsilon_{\theta\theta}$

$4 = \gamma_{rz}$

Output of Stresses

Same as for **Output of Strains**.

Transformation

Two global degrees of freedom can be transformed into local coordinates.

Tying

Can be tied to axisymmetric shell type 1 using standard tying type 23.

Output Points

Output is available at the centroid or at the four Gaussian points shown in Figure 5-12.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinate directions. Reduced volume strain integration recommended. (See **Geometry**.)

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 40. See Element 40 for a description of the conventions used for entering the flux and film data for this element.

Element 11

Arbitrary Quadrilateral Plane-strain

Element type 11 is a four-node, isoparametric, arbitrary quadrilateral written for plane strain applications. As this element uses bilinear interpolation functions, the strains tend to be constant throughout the element. This results in a poor representation of shear behavior. The shear (or bending) characteristics can be improved by using alternative interpolation functions. This assumed strain procedure is flagged through the GEOMETRY option.

In general, you need more of these lower-order elements than the higher-order elements such as types 27 or 54. Hence, use a fine mesh.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using four-point Gaussian integration.

For nearly incompressible behavior, including plasticity or creep, it is advantageous to use an alternative integration procedure. This constant dilatation method, which eliminates potential element locking, is flagged through the GEOMETRY option.

This element can be used for all constitutive relations. When using incompressible rubber materials (for example, Mooney and Ogden), the element must be used within the Updated Lagrange framework.

For rubber materials with total Lagrange procedure, element type 80 can be used. This is slightly more expensive because of the extra pressure degrees of freedom associated with element type 80.

Quick Reference

Type 11

Plane-strain quadrilateral.

Connectivity

Four nodes per element. Node numbering must be counterclockwise (see Figure 5-1).

Geometry

The thickness is entered in the first data field (`EGEOM1`). Default thickness is one.

If a nonzero value is entered in the second data field (`EGEOM2`), the volume strain is constant throughout the element. That is particularly useful for analysis of approximately incompressible materials and for analysis of structures in the fully plastic range. It is also recommended for creep problems in which it is attempted to obtain the steady-state solution.

If a one is entered in the third field, the assumed strain formulation is used.

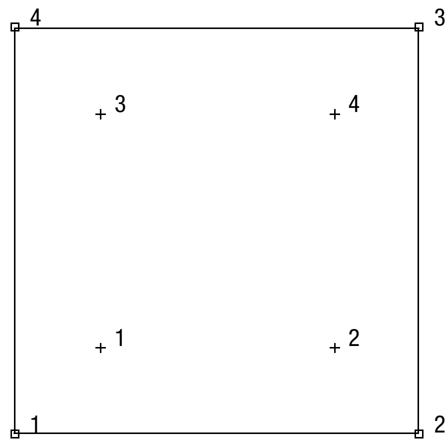


Figure 5-1 Gaussian Integration Points for Element Type 11

Coordinates

Two coordinates in the global x- and y-direction.

Degrees of Freedom

1 = u displacement (x-direction)
2 = v displacement (y-direction)

Distributed Loads

Load types for distributed loads are as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force by unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element; magnitude supplied through the FORCEM user subroutine.
4	Nonuniform body force per unit volume in first coordinate direction; magnitude supplied through the FORCEM user subroutine.
5	Nonuniform body force per unit volume in second coordinate direction; magnitude supplied through the FORCEM user subroutine.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element; magnitude supplied through the FORCEM user subroutine.
8	Uniform pressure on 3-4 face of the element.

Load Type	Description
9	Nonuniform pressure on 3-4 face of the element; magnitude supplied through the FORCEM user subroutine.
10	Uniform pressure on 4-1 face of the element.
11	Nonuniform pressure on 4-1 face of the element; magnitude supplied through the FORCEM user subroutine.
20	Uniform shear force on side 1-2 (positive from 1 to 2).
21	Nonuniform shear force on side 1-2; magnitude supplied through the FORCEM user subroutine
22	Uniform shear force on side 2-3 (positive from 2 to 3).
23	Nonuniform shear force on side 2-3; magnitude supplied through the FORCEM user subroutine.
24	Uniform shear force on side 3-4 (positive from 3 to 4).
25	Nonuniform shear force on side 3-4; magnitude supplied through the FORCEM user subroutine.
26	Uniform shear force on side 4-1 (positive from 4 to 1).
27	Nonuniform shear force on side 4-1; magnitude supplied through the FORCEM user subroutine.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter the magnitude of gravity acceleration in the z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

All pressures are positive when directed into the element. In addition, point loads can be applied at the nodes.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid of the element in global coordinates is:

$1 = \varepsilon_{xx}$

$2 = \varepsilon_{yy}$

$3 = \varepsilon_{zz}$

$4 = \gamma_{xy}$

Output of Stresses

Same as for **Output of Strains.**

Transformation

Two global degrees of freedom can be transformed into local coordinates.

Tying

Use the UFORMSN user subroutine.

Output Points

Output is available at the centroid or at the four Gaussian points shown in Figure 5-1.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinate directions. Reduced volume strain integration is recommended. (See **Geometry.**)

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 39. See Element 39 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux due to dissipation of plastic work specified with type 101.

Assumed Strain

The assumed strain formulation is available to improve the bending characteristics of this element. Although this increases the stiffness assembly costs per element, it improves the accuracy.

Element 12

Friction And Gap Link Element

This element provides frictional and gapping connection between any two nodes of a structure. Essentially, the element is based on imposition of a gap closure constraint and frictional stick or slip via Lagrange multipliers.

The element can be used with any other elements in Marc by invoking suitable tying (UFORMSN user subroutine), if necessary.

Three different options for the definition of the gap have been included. The default formulation is a gap in a fixed direction. This option is useful for geometrically linear analysis or geometrically nonlinear analysis if a body is not to penetrate a given flat surface.

The second formulation constrains the true distance between the two end-points of the gap to be greater or less than a specified value. This option is useful for geometrically nonlinear analysis or for analysis in which a body is not to penetrate a given circular (2-D) or spherical (3-D) surface. This option is activated by specification of a “1” in the seventh data field of the GAP DATA model definition option.

The third formulation allows specification of closure distance and gap direction in the GAPU user subroutine. The gap direction and distance can then be updated during analysis to model sliding along a curved surface. The fixed direction gap option in the GAP DATA option must be activated if GAPU is used.

Note: For general contact, the CONTACT option is the preferred solution.

Fixed Direction Gap

Description of the Fixed Direction Gap for Two-Dimensional Problems

The element is implemented in Marc as a four-node element (link). The first and fourth nodes have (u, v) Cartesian displacements to couple to the rest of the structure.

Node 2 is the gap node.It has one degree of freedom, F_n, the force being carried across the link. The coordinate data for this node is used to input (n_x, n_y), the direction of $\underset{\sim}{n}$, the gap closure direction. If these data are not given (or are all zero), Marc defines:

$$\underset{\sim}{n} = (\underset{\sim}{X}_4 - \underset{\sim}{X}_1) / |\underset{\sim}{X}_4 - \underset{\sim}{X}_1| ;$$

that is, the gap closure direction is along the element in its original configuration. You should note that, in many cases, the gap is very small (or, indeed, can be of zero length if the two surfaces are initially touching), so that inaccuracies can be introduced by taking a small difference between two large values. This is the reason for allowing separate input of (n_x, n_y).

Node 3 is the friction node. It has degrees of freedom F; the frictional force being carried across the link, and s, the net frictional slip. The coordinate data for this node can be given as (t_x, t_y), the frictional direction. If you do not input this data, $\underset{\sim}{t}$ is defined by Marc as:

$$\underset{\sim}{t} = \underset{\sim}{k} \times \underset{\sim}{n}$$

where $\underset{\sim}{n}$ is the gap direction (see above) and $\underset{\sim}{k}$ is the unit vector normal to the plane of analysis.

Description of the Fixed Direction Gap for Three-Dimensional Problems

The first and fourth nodes have (u, v, w) Cartesian displacements to couple to the rest of the structure.

Node 2 is the gap node. It has one degree of freedom, F_n; the normal force being carried across the link. The coordinate data for this node is used to input (n_x, n_y, n_z), the direction of $\underset{\sim}{n}$, the gap closure direction. If these data are not given (or are all zero), Marc defines:

$$\underset{\sim}{n} = (\underset{\sim}{X}_4 - \underset{\sim}{X}_1) / \left|\underset{\sim}{X}_4 - \underset{\sim}{X}_1\right| ;$$

that is, the gap closure direction is along the element in its original configuration. You should note that in many cases, the gap is very small (or, indeed, can be of zero length if the two surfaces are initially touching), so that inaccuracies can be introduced by taking a small difference between two large values. This is the reason for allowing separate input of (n_x, n_y, n_z).

Node 3 is the friction node. It has degrees of freedom (F_1, F_2); the frictional forces being carried cross the link and s, the net frictional slip.

The coordinate data for this note can be given as (t_x^1, t_y^1, t_z^1) , the first frictional direction. If you do not input this data, $\underset{\sim}{t}^1$ is defined by Marc as:

$$\underset{\sim}{t}^1 = \underset{\sim}{i} \times \underset{\sim}{n}$$

where $\underset{\sim}{n}$ is the gap direction (see above) and $\underset{\sim}{i}$ is the unit vector in the global $\underset{\sim}{x}\textit{-direction}$. If $\underset{\sim}{n}$ is parallel to $\underset{\sim}{i}$, the first friction direction is defined as:

$$\underset{\sim}{t}^1 = -\underset{\sim}{j}$$

where j is the global y-direction. The second friction direction $\underset{\sim}{t}^2$ is calculated by Marc as:

$$\underset{\sim}{t}^2 = \underset{\sim}{n} \times \underset{\sim}{t}^1$$

Note on the Fixed Directional Gap

The gap is closed when $(\underset{\sim}{u}_1 - \underset{\sim}{u}_4) \bullet \underset{\sim}{n} = u_{c1}$, and this relative displacement in direction $\underset{\sim}{n}$ cannot be exceeded. The closure distance u_{c1} is given in the first data field of the GAP DATA option. A negative number insures that a closed gap condition is prescribed during increment zero. Note that no nonlinearity is accounted for in increment zero; i.e., the gap should be either open or closed as defined by you. The closure distance can be updated by you through the GAPU user subroutine.

The second data field of the GAP DATA option is used to input the coefficient of friction, μ. If the coefficient of friction is set to zero, the friction calculations are skipped and the element acts as a gap only.

True Distance Gap

Description of the True Distance Gap

In this formulation, the nodes have the same meaning as in the fixed direction formulation. Nodes 1 and 4 have (u, v) Cartesian displacements to couple to the rest of the structure. For three-dimensional problems, nodes 1 and 4 have (u, v, w) Cartesian displacements. Node 2 has one degree of freedom, the gap force F_n. In contrast to the fixed direction gap, the constraint enforced by this true distance gap is as follows:

$$\left|\underset{\sim}{x}_4 - \underset{\sim}{x}_1\right| \geq d, \qquad \text{if } d > 0$$

or

$$\left|\underset{\sim}{x}_4 - \underset{\sim}{x}_1\right| \leq -d \qquad \text{if } d < 0$$

where $|d|$ is the minimum or maximum distance between the end-points defined in the first data field of the GAP DATA option. Note that this distance must always be positive. From the above equation follows that the gap closure direction is defined as follows:

$$\underset{\sim}{n} = \pm(\underset{\sim}{x}_4 - \underset{\sim}{x}_1) / \left|\underset{\sim}{x}_4 - \underset{\sim}{x}_1\right|;$$

i.e., the gap closure direction is along the element in its current configuration. You cannot specify any different direction.

General Comments

Since it is very important that the degrees of freedom of the gap element are eliminated in an appropriate order, automatic internal renumbering of the nodes connected to the gap is carried out prior to the analysis (but after eventual optimization). You cannot influence this procedure. For very small models this process may not work, and the results are unreliable. In problems with large numbers of gaps and/or high friction coefficients, the convergence of the gap and friction algorithm is sometimes rather slow. Very often, this is due to iterations of elements in areas on the borderline of opening-closing and/or slipping-sticking. This is a local effect that shows itself in the often good convergence of other measures, such as the displacements. Hence, even if gap convergence is not reached completely in the specified number of cycles, the solution can still be sufficiently accurate for all practical purposes. In that case, Marc continues with the analysis after issuing a warning message. If no convergence problems occur in subsequent increments, such a nonconvergence usually has no significant effect in the subsequent results. To obtain information regarding gap convergence, use the PRINT,5 parameter.

Quick Reference

Type 12

Four-node friction/gap element. Can be used with any other element types, if necessary through appropriate tying.

Connectivity

Four nodes per element. Nodes 1 and 4 are the ends of the link to connect to the rest of the structure, node 2 is the gap node and node 3 is the friction node.

Marc automatically renumbers the internal node numbers to avoid equation solver problems. This occurs after optimization and can lead to a nonoptimal bandwidth.

Coordinates

Nodes 1 and 4 - Cartesian coordinates (x, y) for two-dimensional problems, otherwise (x, y, z).

Node 2 and 3 - (fixed direction gap only):

Node 2 - gap direction cosines (n_x, n_y) or (n_x, n_y, n_z) for 3-D problems
Node 3 - friction direction cosines (t_x, t_y) or (t_x, t_y, t_z) for 3-D problems

Gap Data

First Data Field

For the fixed direction gap, this field is used to define u_{c1}, the closure distance.

For the true distance gap, this field is used to define d, the minimum distance between the end-points. If $d > 0$, the two end-points are at least a distance d apart. If $d < 0$, the two end-points do not move further apart than a distance -d.

Second Data Field

This data field is used to define the coefficient of friction.

Third Data Field

The data field is used to define the elastic stiffness (spring stiffness) of the closed gap in the gap direction. If the field is left blank, the gap is assumed to be rigid when closed.

Fourth Data Field

This data field is used to define the elastic stiffness (spring stiffness) of the closed gap in friction direction. If the field is left blank, the nonslipping gap is rigid in the slip direction.

Fifth Data Field

User supplied momentum ratio for the first gap node.

Sixth Data Field

User supplied momentum ratio for the fourth gap node.

Seventh Data Field

User enters a “1” for true distance gap; a “0” for fixed direction gap.

Eighth Data Field

User enters a “1” for the condition that the gap is closed during increment 0; a “0” for the condition that the gap is open during increment 0.

Degrees Of Freedom

Nodes 1 and 4 (u, v) Cartesian components of displacement in two-dimensional problems, (u, v, w) Cartesian components of displacement in three-dimensional problems.

Node 2 - F_n, force in the gap direction.

Node 3 - F_s, frictional force and, s, net frictional slip in two-dimensional problems, or F_1, F_2, frictional forces and s, net frictional slip in three-dimensional problems.

s is the accumulated total slip under nonzero frictional forces.

Special Considerations

The TRANSFORMATION option should not be invoked at nodes 2 and 3 of this element.

Updated Lagrange And Finite Strain Plasticity

Use true distance gap or the GAPU user subroutine if necessary – large strain option not relevant.

Element 13

Open Section Thin-Walled Beam

This element is an open-section, curved, thin-walled beam of arbitrary section. The geometry is interpolated cubically from coordinate and direction information at two nodes. The element is illustrated in Figure 5-2. The following pages describe how you can set up the cross section and the orientation of the beam and its section in space, and define the degrees of freedom, strains and distributed loads associated with the element. The element is based on classical theory of thin-walled beams with nondeforming sections.

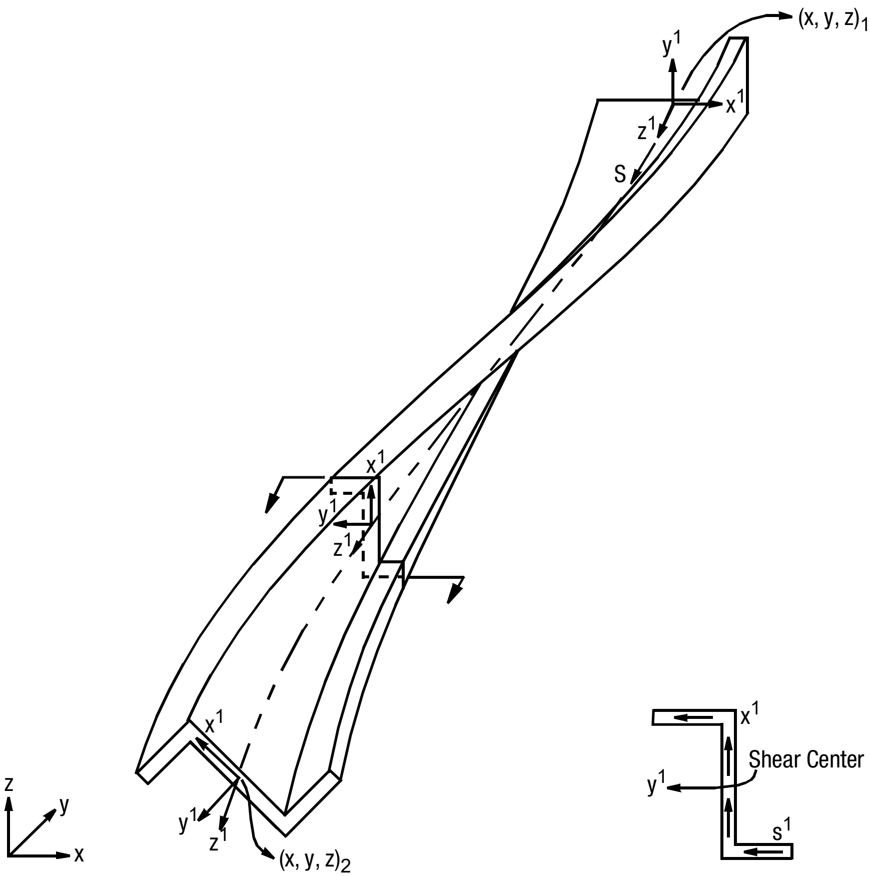


Figure 5-2 Typical Beam Element, (x^l, y^l, z^l)

Primary warping effects are included, but twisting is always assumed to be elastic as follows:

$$M_T = GJ\frac{d\phi}{ds}$$

where G = shear modulus and $\frac{d\phi}{ds}$ = rate of twist per unit length along the beam axis.

Thus, the shear stress is assumed small compared to the axial stress and is neglected in the formation of elastic-plastic and creep relations. The twisting stiffness is formed directly by numerical integration:

$$J = \int_{o}^{\Sigma} \frac{1}{3}(t\ (\sigma))^3\ d\sigma$$

where σ is the distance along the section, 0<σ<Σ, and t(σ) is the wall thickness.

The axial force, bending moments and bimoment are formed by numerical integration of the direct stress, which is obtained via one-dimensional elastic-plastic creep constitutive theory.

The cross section is assumed to remain undeformed during loading of the beam. Large displacement effects are included in the direct strain only.

Definition of Open Section Beam Geometry

Axis System

The convention adopted in Element 13 for the director set at a point of the beam is as follows:

The first and second directions (local x and y) at a point are normal to the beam axis.

The third director (local z) is tangent to the beam axis and is in the direction of increasing distance s along the beam.

The director set must form a right-handed system.

Orientation of the Section in Space

The beam axis in an element is interpolated by a cubic from the first six coordinates at the two nodes of an element. The coordinates are as follows:

$$x,\ y,\ z,\ \frac{dx}{ds},\ \frac{dy}{ds},\ \frac{dz}{ds}$$

where s is the continuous distance along the beam and is given as the thirteenth coordinate at each node. The orientation of the beam section in an element is defined by the direction of the first director at a point (local x), and this direction is interpolated by a cubic from the seventh through the twelfth coordinates at the two nodes of an element. The coordinates are as follows:

$$a_1, a_2,\ a_3,\ \frac{da_2}{ds},\ \frac{da_3}{ds},\ \frac{da_3}{ds}$$

where a_1, a_2, a_3, are the components in the global directions of the first director at the node. Since the interpolated director $\underline{a}$ is not, in general, orthogonal to the interpolated beam axis tangent $\underline{a}_3$, an internal correction is applied at each numerical integration point of the element according to the formula:

$$\underline{b}_1^c = \frac{1}{\sqrt{1-(\underline{b}_1\cdot\underline{b}_3)^2}}(\underline{b}_1\bar{\underline{b}}_1^c\cdot\underline{b}_3)\underline{b}_3)$$

where $\underset{\sim}{b}_1$, $\underset{\sim}{b}_3$, are the unit vectors along $\underset{\sim}{a}_1$, a_3, and $\underset{\sim}{b}_1^c$ is the corrected unit vector along the first director. The second director is then obtained from the cross product $\underset{\sim}{b}_3 \times \underset{\sim}{b}_1$.

Displacements

There are eight degrees of freedom at each node. These are as follows:

$$u,\ \frac{du}{ds},\ v,\ \frac{dv}{ds},\ w,\ \frac{dw}{ds},\ \phi,\ \frac{d\phi}{ds}$$

where u, v, w, are the components of displacement in the global directions and ϕ is the rotation about the beam axis. Here, $\frac{d}{ds}$ represents differentiation with respect to distance along the beam.

Strains

Five generalized strains are associated with each integration point along the beam axis. These are oriented with respect to the interpolated director set (1, 2, 3) at the integration point are defined as follows:

1 = Direct strain on the beam axis (along third director).
2 = Curvature about the first director.
3 = Curvature about the second director.
4 = $\frac{d^2\phi}{ds^2}$ where ϕ is rotation about the third direction; i.e., warping of the section.
5 = $\frac{d\phi}{ds}$ = twist of the section.

Distributed Loads

Distributed loads are available as uniform load per unit length in the global (x, y, z) directions. The type of load is given as 1(x), 2(y), or 3(z) and the magnitude is given as the load per unit length along the beam.

Quick Reference

Type 13

Open-section, thin-walled beam of arbitrary section, including twist and warping.

Connectivity

Two nodes per element.

Geometry

Section is defined by you in an additional set of data blocks. The section number is given for an element by input in the second data field (EGEOM2). The other data fields are not used.

Coordinates

Beam axis and cross-section orientation interpolated cubically from 13 coordinates per node:

$$x,\ y,\ z,\ \frac{dx}{ds},\ \frac{dy}{ds},\ \frac{dz}{ds},\ a_1,\ a_2,\ a_3,\ \frac{da_1}{ds},\ \frac{da_2}{ds},\ \frac{da_3}{ds},\ s$$

where (a_1, a_2, a_3) is a vector defining the direction of the first local axis of the cross section.

Degrees of Freedom

Eight degrees of freedom per node: $u,\ \frac{du}{ds},\ v,\ \frac{dv}{ds},\ w,\ \frac{dw}{ds},\ \phi,\ \frac{d\phi}{ds}$ where s is the distance along the beam.

Tractions

Distributed loads are per unit length of beam in the three global directions.

Load Type	Description
1	Uniform load per unit length in x-direction.
2	Uniform load per unit length in y-direction.
3	Uniform load per unit length in z-direction.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in the x-, y-, and z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Generalized strains:

1 = ε_{xx} = axial stretch
2 = κ_{xx} = curvature about local x-axis of cross section.
3 = κ_{yy} = curvature about local y-axis of cross section.
4 = η = warping
5 = γ = twist

Output of Stresses

Generalized stresses:

1 = axial stress
2 = local xx-moment
3 = local yy-moment
4 = bimoment
5 = axial torque

Transformation

The displacement vectors can be transformed into local degrees of freedom.

Tying

Use tying type 13 to join two elements under an arbitrary angle. It can be used as a stiffener on the arbitrary curved shell element (element type 8) by tying types 19, 20, 21.

Output Points

Centroid or three Gaussian integration points along the beam. First point is closest to the first node of the beam.

For all beam elements, the default printout gives section forces and moments, plus stress at any layer with plastic, creep strain, or nonzero temperature. This default printout can be changed via the PRINT CHOICE option.

Beam Sect

The BEAM SECT parameter is required.

Updated Lagrange Procedure and Finite Strain Plasticity

Updated Lagrange capability is not available. Use element types 77 or 79.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is 36. See Element 36 for a description of the conventions used for entering the flux and film data for this element.

Element 14

Thin-walled Beam in Three Dimensions without Warping

This is a simple, straight beam element with no warping of the section, but including twist. The default cross section is a thin-walled circular closed-section beam. You can specify alternative cross sections through the BEAM SECT parameter.

The degrees of freedom associated with each node are three global displacements and three global rotations; all defined in a right-handed convention. The generalized strains are stretch, two curvatures, and twist per unit length. Stresses are direct (axial) and shear given at each point of the cross section. The local coordinate system which establishes the positions of those points on the section is defined in Geometry fields 4, 5, and 6. Using the GEOMETRY option, a vector in the plane of the local x-axis and the beam axis must be specified. If no vector is defined here, the local coordinate system can alternatively be defined by the fourth, fifth, and sixth coordinates at each node, which give the (x,y,z) global Cartesian coordinates of a point in space which locates the local x-axis of the cross section. This axis lies in the plane defined by the beam nodes and this point, pointing from the beam toward this point. The local z-axis is along the beam from the first to the second node and the local y-axis forms a right-handed set with the local x and local z.

For other than the default (circular) section, the stress points are defined by you in the local x-y set through the BEAM SECT parameter set. For the circular hollow section, `EGEOM1` is the wall thickness, `EGEOM2` is the radius. Otherwise, `EGEOM2` gives the section choice from the BEAM SECT input. Section properties are obtained by numerical integration over the stress points of the section.

All constitutive models can be used with this element.

Standard (Default) Circular Section

The positions of the 16 numerical integration points are shown in Figure 5-15. The contributions of the 16 points to the section quantities are obtained by numerical integration using Simpson's rule.

Note:	For noncircular sections, the BEAM SECT parameter must be used to describe the section.

Special Considerations

Note that element 25 is the same as this element, but with axial strain as an additional degree of freedom at each node. This yields superior results for large displacement problem or problems involving axial temperature gradients. Elements of types 14, 25, 52, 76, 77, 78, 79, and 98 can be used together directly.

For all beam elements, the default printout gives section forces and moments plus stress at any layer with plastic, creep strain or nonzero temperature. This default printout can be changed via the PRINT ELEMENT option.

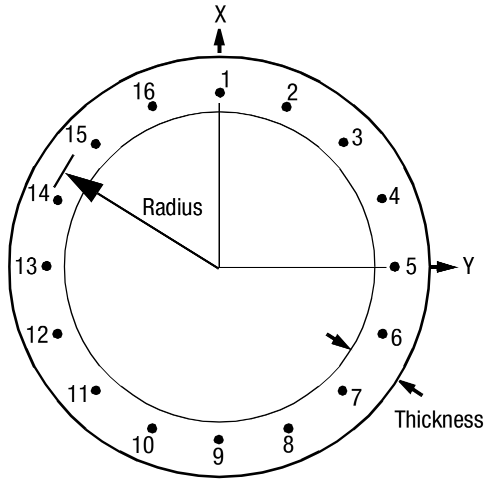


Figure 5-15 Default Cross Section

Quick Reference

Type 14

Closed section beam, Euler-Bernoulli theory.

Connectivity

Two nodes per element (see Figure 5-16).

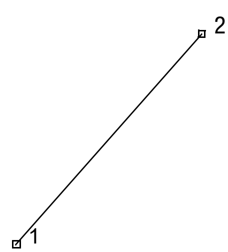


Figure 5-16 Closed-section Beam

Geometry

In the default section of a hollow, circular cylinder, the first data field is for the thickness (EGEOM1). For noncircular section, set EGEOM1 to 0. For circular section, set EGEOM2 to radius. For noncircular section, set EGEOM2 to the section number needed. (Sections are defined using the BEAM SECT parameter.) EGEOM4-EGEOM6: Components of a vector in the plane of the local x-axis and the beam axis. The local x-axis lies on the same side as the specified vector.

If beam-to-beam contact is switched on (see CONTACT option), the radius used when the element comes in contact with other beam or truss elements must be entered in the seventh data field (EGEOM7).

For segment-segment beam contact, various flags that control the creation of patches are entered in the seventh data file (`EGEOM7`).

`EGEOM7 = real(100 * NSEG + 10 * IPTCH + IESCAP)`

where

`NSEG` = number of segments for hollow tubes (default = 32)		
`IPTCH` = patch flag for hollow tubes		
	1	Both outer and inner patches (default).
	3	Only outer patches (at radius + 1/2 * thickness).
	-3	Only outer patches (at radius - i.e., ignore thickness).
	5	Only inner patches (at radius - 1/2 * thickness).
	-5	Only inner patches (at radius - i.e., ignore thickness).
`IESCAP` = end cap/side cap flag		
	0	No end or side cap (default).
	1	End cap only.
	2	Side cap only.
	3	Average side only.
	4	End and side cap.
	5	End cap and average side.

Beam Offsets Pin Codes and Local Beam Orientation

If an offset is applied to the beam geometry or pin codes are applied to the degrees of freedom of the end point of the beams or the beam axis is defined with respect to a local coordinate system, then a code is placed in the 8th field of the 3rd data block of the GEOMETRY option (`EGEOM8`). This code is formed by the negative sum of ioffset, iorien, and ipin where:

ioffset	=	0	-	no offset
		1	-	beam offset
iorien	=	0	-	conventional definition of local beam orientation; beam axis given in 4th through 6th field with respect to global system
		10	-	the local beam orientation entered in the 4th through 6th field is given with respect to the coordinate system of the first beam node. This vector is transformed to the global coordinate system before any further processing.
ipin	=	0	-	no pin codes are used
		10 0	-	pin codes are used.

If ioffset = 1, an additional line is read in the GEOMETRY option (4th data block).

If ipin = 100, an additional line is read in the GEOMETRY option (5th data block)

The offset vector at the first corner node is provided in the first three fields and the offset vector at the second corner node is provided in the next three fields of the extra line. The coordinate system used for the offset vectors at the two nodes are indicated by the corresponding flags in the eighth and ninth fields of the extra line. A flag setting of 0 indicates global coordinate system, 1 indicates local element coordinate system, 2 indicates local shell normal and 3 indicates local nodal coordinate system. When the flag setting is 1, the local beam coordinate system is constructed by taking the local z axis along the beam, the local x axis as the user-specified vector (obtained from `EGEOM4-EGEOM6`) and the local y axis as perpendicular to both. The flag setting of 2 can be used when the beam nodes are also attached to a shell. In this case, the offset magnitude at the first and second nodes are specified in the first and fourth data fields of the extra line respectively. The automatically calculated shell normals at the nodes are used to construct the offset vectors.

Coordinates

Six coordinates per node. The first three are global (x,y,z). The fourth, fifth, and sixth are the global x,y,z coordinates of a point in space which locates the local x-axis of the cross section. The local x axis is a vector normal to the beam axis through the point described by the fourth, fifth and sixth coordinates. The local x axis is positive progressing from the beam to the point. The fourth, fifth, and sixth coordinates are only used if the local x-axis direction is not specified in the GEOMETRY option.

Degrees of Freedom

1 = u	4 = θ_x
2 = v	5 = θ_y
3 = w	6 = θ_z

Tractions

Distributed load types are as follows:

Load Type	Description
1	Uniform load per unit length in the global x-direction.
2	Uniform load per unit length in the global y-direction.
3	Uniform load per unit length in the global z-direction.
4	Nonuniform load per unit length; magnitude and direction supplied via the FORCEM user subroutine.
11	Fluid drag/buoyancy loading – fluid behavior specified in the FLUID DRAG option.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in the x-, y-, and z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Load Type	Description
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Point loads and moments can be applied at the nodes.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Generalized strains:

1 = ε_{zz} = axial stretch
2 = κ_{xx} = curvature about local x-axis of cross section.
3 = κ_{yy} = curvature about local y-axis of cross section.
4 = γ = twist

Output of Stresses

Generalized stresses:

1 = axial stress
2 = local xx-moment
3 = local yy-moment
4 = axial torque

Transformation

Displacement and rotations at the nodes can be transformed to a local coordinate reference.

Tying

Special tying types exist for use of this element with element 17 to form complete pipelines for nonlinear piping system analysis. See element 17 description. Use tying type 100 for fully moment-carrying joints and tying type 52 for pinned joints.

Output Points

Centroid or three Gaussian integration points. The first point is near the first node in the connectivity description of the element. The second point is at the midspan location of the beam. The third point is near the second node in the connectivity description of the element.

Updated Lagrange Procedure and Finite Strain Plasticity

Updated Lagrange procedure is available for this element. This element does not have a finite strain capability.

Coupled Analysis

In a coupled thermal-mechanical, analysis the associated heat transfer element is type 36. See Element 36 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

For the default hollow circular section only, the wall thickness and the radius can be considered as design variables.

Element 15 — Axisymmetric Shell, Isoparametric Formulation

Element type 15 is a two-node, axisymmetric, thin-shell element, with a cubic displacement assumption based on the global displacements and their derivatives with respect to distance along the shell. The strain-displacement relationships used are suitable for large displacements with small strains. The stress-strain relationship is integrated through the thickness using Simpson's rule, the first and last points being on the surfaces. Three-point Gaussian integration is used along the element. All constitutive relations can be used with this element.

Quick Reference

Type 15

Axisymmetric, curved, thin-shell element.

Connectivity

Two nodes per element (see Figure 5-17).

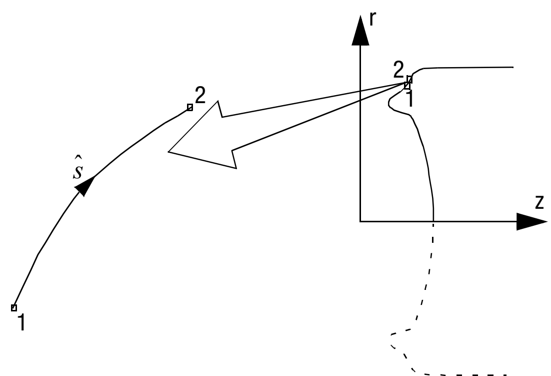


Figure 5-17 Axisymmetric, Curved Thin-shell Element

Geometry

Linear thickness variation along length of the element. Thickness at first node of the element store in the first data field (`EGEOM1`).

Thickness at second node store in the third data field (`EGEOM3`).

If `EGEOM3=0`, constant thickness is assumed. Notice that the linear thickness variation is only taken into account if the ALL POINTS parameter is used since, in the other case, section properties formed at the centroid of the element are used for all integration points.

The second data field is not used (`EGEOM2`).

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

Coordinates

1 = z

2 = r

$3 = \frac{dz}{ds}$

$4 = \frac{dr}{ds}$

5 = s

Note:	The redundancy in the coordinate specification is retained for simplicity of use with generators.

Degrees of Freedom

1 = u = axial (parallel to symmetry axis)

2 = v = radial (normal to symmetry axis)

$3 = \frac{du}{ds}$

$4 = \frac{dv}{ds}$

Tractions

Distributed loads selected with IBODY are as follows:

Load Type (IBODY)	Description
0	Uniform pressure.
1	Uniform load in 1 direction (force per unit area).
2	Uniform load in 2 direction (force per unit area).
3	Nonuniform load in 1 direction (force per unit area).
4	Nonuniform load in 2 direction (force per unit area).
5	Nonuniform pressure.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter the magnitudes of gravity acceleration in the z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Load Type (IBODY)	Description
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Pressure assumed positive in direction opposite of the normal obtained by rotation of 90° from direction of increasing s (see Figure 5-17).

In the nonuniform cases (`IBODY = 3`, `4`, or `5`), the load magnitude must be supplied by the FORCEM. user subroutine

Concentrated loads applied at the nodes must be integrated around the circumference.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Generalized strains are as follows:

1 = meridional membrane (stretch)
2 = circumferential membrane (stretch)
3 = meridional curvature
4 = circumferential curvature

Output of Stresses

Stresses are output at the integration points through the thickness of the shell. The first point is on the surface of the positive normal.

1 = meridional stress
2 = circumferential stress

Transformation

The degrees of freedom can be transformed to local directions.

Special Transformation

The shell transformation option type 1 can be used to permit easier application of moments and/or boundary conditions on a node. For a description of this transformation type, see Marc Volume A: Theory and User Information . Note that if the FOLLOW FOR parameter is invoked, the transformations is based on the updated configuration of the element.

Output Points

Centroid or three Gaussian integration points. The first Gaussian integration point is closest to the first node as defined in the connectivity data. The second integration point is at the midspace location. The third integration point is at the second node as defined in the connectivity data.

Section Stress Integration

Simpson's rule is used to integrate through the thickness. Use the SHELL SECT parameter to specify the number of integration points.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – output of stress and strain in meridional and circumferential direction. Thickness is updated.

Note:	Shell theory only applies if strain variation through the thickness is small.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is element type 88. See Element 88 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered as a design variable.

Element 16

Curved Beam in Two-dimensions, Isoparametric Formulation

Element type 16 is a two-node curved beam, with displacements interpolated cubically from the global displacements and their derivatives with respect to distance along the beam at the two end nodes. The strain-displacement relations used are suitable for large displacements with small strains. The stress-strain relationship is integrated through the thickness by a Simpson rule; the first and last points being on the two surfaces. Three-point Gaussian integration is used along the element. The cross section is a solid rectangle. All constitutive relations can be used with this element.

Quick Reference

Type 16

Two-node curved beam element.

Connectivity

Two nodes per element (see Figure 5-18).

Figure 5-18 Two-node Curved Beam Element

Geometry

Linear thickness variation along the element. Thickness at the first node of the element is stored in the first data field (`EGEOM1`). Thickness at second node is stored in the third data field (`EGEOM3`). If `EGEOM3 = 0`, constant thickness is assumed.

Notice that the linear thickness variation is only taken into account if the ALL POINTS parameter is used; since in the other case, section properties formed at the centroid of the element are used for all integration points.

The beam width is in the second data field (`EGEOM2`). The default width is unity.

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

Coordinates

Nodal coordinates in right-hand set (x,y)

$$1 = x$$
$$2 = y$$
$$3 = \frac{dx}{ds}$$
$$4 = \frac{dy}{ds}$$
$$5 = s$$

where s is the distance along the beam measured continuously starting from one end of the beam.

Degrees of Freedom

Degrees of freedom in right-hand set (u,v):

1 = u
2 = v

$3 = \frac{du}{ds}$

$4 = \frac{dv}{ds}$

where s = distance along the beam.

Tractions

Distributed loads selected with IBODY are as follows:

Load Type (IBODY)	Description
0	Uniform pressure.
1	Uniform load (force per unit length) in 1 direction.
2	Uniform load (force per unit length in 2 direction.
3	Nonuniform load (force per unit length) in 1 direction.
4	Nonuniform load (force per unit length) in 2 direction.
5	Nonuniform pressure.
11	Fluid drag/buoyancy loading – fluid behavior specified in the FLUID DRAG option.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter the magnitudes of gravity acceleration in the x- and y-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Pressure assumed positive in direction opposite of the normal obtained by rotation of 90° from direction of increasing s (see Figure 5-18).

In the nonuniform cases (`IBODY = 3`, `4`, or `5`), the load magnitude must be supplied by the FORCEM user subroutine.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Generalized strains:

1 = membrane (stretch)
2 = curvature

Output of Stresses

Output of axial stress at points through thickness (first and last points are on surfaces). First point is on surface up positive normal. Points proceed down normal at equal intervals.

Transformation

All degrees of freedom can be transformed to local directions.

Special Transformation

The shell transformation option type 1 can be used to permit easier application of moments and/or boundary conditions on a node. For a description of this transformation type, see Marc Volume A: Theory and User Information, Chapter 9. Note that if the FOLLOW FOR parameter is invoked, the transformations is based on the updated configuration of the element.

Tying

Requires the UFORMSN user subroutine.

Output Points

The first Gaussian integration point is closest to the first node defining the element in the connectivity data. The second point is at the midspan beam location. The third point is closest to the second node describing the element in the connectivity data.

Section Stress Integration

Integration through-the-thickness is performed numerically using Simpson's rule. The number of integration points is specified with the SHELL SECT parameter. This number must be odd.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – output of stress and strain in beam direction. Thickness is updated, but beam width is assumed to be constant.

Note:	Beam theory only applies if strain variation over the thickness is small.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is element type 36. See Element 36 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness (beam height) and/or the beam width can be considered as design variables.

Element 17

Constant Bending, Three-node Elbow Element

This element modifies Element type 15 into a pipe-bend approximation. The main purpose of the element is to provide nonlinear analysis of complete piping loops at realistic cost: The straight pipe sections are modeled by Element type 14, and the bends are built up by using several sections of these modified axisymmetric elements. Each such section has a beam mode, with constant stretch and curvatures, superposed on the axisymmetric shell modes so that ovalization of the cross section is admitted. The element has no flexibility in torsion so that the twisting of a pipe-bend section is ignored, and the rotations along the section secant are made equal by tying. Pipe-bend sections are coupled together and into straight beam elements by extensive use of special default tying types and are described later in this section. Thus, a complete pipe-bend might consist of several sections of elements tied together with each section being build up from several elements and all sharing a common "elbow" node and having the usual two nodes on the shell surface. All constitutive relations can be used with this element. This element cannot be used with the CONTACT option.

A pipe-bend section is shown in Figure 5-19. In the plane of the section (the z-r plane), there are several elements. At the first two nodes of each element of the section are the shell degrees of freedom associated with Element type 15:

where u and v are displacements in the z and r directions in the plane of the section. The third node is the same at all elements of the section and with this node are associated the beam modes:

Δu -normal motion of one end plane with the other end plane fixed.

$\Delta\phi$ -in-plane rotation of one end plane with the other end plane fixed, that is, rotation about the z-axis in Figure 5-19 (ϕ positive closes elbow).

$\Delta\psi$ -out-of-plane rotation of one end plane with the other end plane fixed (ψ positive gives tension in z > 0 in Figure 5-19).

It is assumed that these motions create a stress state in the section independent of position around the bend.

The geometry of the elbow is defined in two ways: in the section and in space. In the section, the geometry is input by placing the pipe surface nodes around the pipe in a z-r section, so that the axis of the pipe-bend torus is on r = 0, and the pipe center line is on z = 0.

At each pipe surface node, the coordinates are those for Element type 15:

$$z,\ r,\ \frac{dz}{ds},\ \frac{dr}{ds},\ s$$

where s is the distance around the pipe surface. Notice that since the section is close, there is discontinuity in s (s = 0 is the same point as s = $2\pi a$). Two nodes must be placed at this point and constrained to the same displacement by tying. Notice also that it is permissible to have unequal sized elements in this plane.

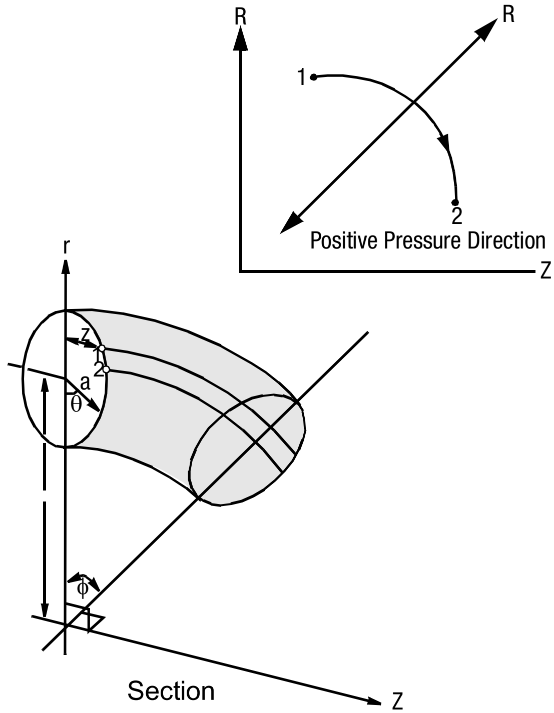


Figure 5-19 Typical Pipe-bend Section

The geometry of the section in space is defined by the GEOMETRY option as follows. For each element on the section:

`EGEOM1` = pipe thickness.

`EGEOM2` = angle ϕ (Figure 5-19) in degrees; i.e., angular extent of pipe-bend section around the pipe-bend torus.

`EGEOM3` = torus radius; i.e., radius to center of pipe in r-z plane.

There are no coordinates associated with the "elbow" (shared) node of the section.

The section is oriented in space and linked to other such sections or straight pipes by the introduction of additional nodes placed at the actual location in space of the center point of the ends of the section. These nodes are not associated with any element, but serve to connect the complete pipeline through the special default tying types developed for this element and described below. Each of these nodes has six coordinates. The first three are its (x,y,z) position in space and the second three are the (x,y,z) position of the center of the pipe-bend torus with which the node is associated.

Tying of the Pipe-Bend Section

Two tying types are required to complete a pipe-bend section. A further tying is required to link the section to the two nodes introduced at the ends of the section. This latter is not necessary if the pipe-bend section is used alone; e.g., to study nonlinear behavior of an individual component under in-plane loadings (see Marc Volume A: Theory and User Information). The data required for each of the above tying types are described as follows:

1. The pipe must be closed around the discontinuity in s (in the z-r section) by tying all degrees of freedom at the last shell node of the section to the corresponding degrees of freedom at the first shell node of the section. This can be achieved by specifying tying type 100, with one retained node. The last node on the section (s = 2πa) is given as tied node, and the first node on the section (s = 0) is given as the retained node.
2. The rigid body modes in the r-z section must be eliminated. This is achieved by the special default tying type 16, which ensures that the integrated u- and v-displacements around the section are zero, based on the cubic displacement assumption in this plane within each element. This tying drops u and v at the first node of the section in terms of the four shell degrees of freedom at all other nodes of the section and $\frac{du}{ds}, \frac{dv}{ds}$ at the first node, according to:

$$\bar{u} = o = \frac{1}{(n= 1)}\{u_1 + u_2 + \ldots + u_{n-1} + \frac{1}{12}(s_2 - s_1 - s_n + s_{n-1})\frac{du_1}{ds} + (s_1 - 2s_2 + s_3)\frac{du_2}{ds} + \ldots + (s_{n-2} - 2s_{n-1} + s_n)\frac{du_{n-1}}{ds}\}$$

3. with n = number of nodes on the section. Notice that this tying specifies v = 0. Net radial motion of the torus section is accounted for by the first degree of freedom at the elbow (shared) node of the section. This means that load coupling between these nodes must be created externally by you: for example, with pressure in the pipe, the net out-of-balance force on the section must be appropriately applied at the first degree of freedom of the “elbow” node of the section.

   The data defining this tying type is: tying type 16, with the number of retained nodes equal to the number of shell nodes in the z-r plane of the section (counting the two coincident end nodes separately). The tied node is the first shell node of the section (at s = 0) and the list of retained nodes gives all other shell nodes of the section in order of increasing s (including the end node at s = 2πa); then, lastly, the first shell node of the section again.
4. For out-of-plane bending, the low stiffness mode associated with the section rotating about the beam axis should be removed. This is done by dropping the u_z degree of freedom at the second node on the section according to:

$$\sum_{n_i = 1}^{n} u_i^t = 0$$

5. where u^t is the displacement tangent to the pipe wall in the section,

$$u^t = u_z \cos\phi + u_R \sin\phi$$

6. The data for this tying type is: tying type 15, number of retained nodes equal to one less than that used for tying type 16. The tied node is the second shell node of the section and the retained nodes are the first, third, fourth, etc., up to the next to the last node of the section; then, lastly, repeat the tied node.
7. If the section is being tied to the two external nodes introduced at the center points of its end planes, tying type 17 must be used with two retained nodes and one of the external nodes as tied node, and the other external node, then, the “elbow” node of the section as retained nodes. This tying removes all degrees of freedom at the external node given as tied node. Care must be taken with multiple-section bends to ensure these nodes are removed in an appropriate order.

Basis of the Element

The element is based on a superposition of purely axisymmetric strains generated by the shell modes, and the beam bending strains introduced through the degrees of freedom at the "elbow" node. Defining the latter as Δu, $\Delta\psi$ and $\Omega\chi$ for stretch, relative in-plane and out-of-plane rotations, respectively. An additional strain is superposed on the circumferential strain of the axisymmetric shell:

$$\varepsilon^{22} = \left[\frac{\Delta u}{r} + \Delta\phi\left(1 - \frac{\bar{r}}{r}\right) + \frac{z}{r}\Delta\Psi\right]\frac{1}{\phi}$$

where $\bar{r}$, z, and ϕ are defined in Figure 5-19.

Since $\frac{\Delta u}{\phi}$ causes the same strain as a mean radial motion of the torus in the (z-r) plane, the constraint

$\bar{v} = \frac{1}{2\pi}\int_{o}^{2\pi} v d\theta = 0$ is applied. Similarly, $\bar{u} = \frac{1}{2\pi}\int_{o}^{2\pi} u d\theta = 0$ is applied to remove this rigid body mode.

The pipe-bend section is linked externally by coupling relative motions of the external nodes introduced at the centers of the end planes to the relative displacements used as degrees of freedom at the "elbow" node of the section. Referring to Figure 5-20, in terms of the local set:

$$(u_x, u_y, u_z, \phi_x, \phi_y, \phi_z)$$

introduced in the local coordinates (x^1, y^1, z^1) at the two external nodes A and B and $\Delta\phi$, $\Delta\phi$, $\Delta\psi$ at the "elbow" node, these couplings are:

$$u_x^B = u_x^A \cos\phi = u_y^A \sin\phi - \theta_z^A \bar{r}(1 - \cos\phi) + \Delta u + \frac{\bar{r}}{\phi}(1 - \sin\phi)\Delta\phi$$

$$u_y^B = u_x^A \sin\phi + u_y^A \cos\phi + \theta_z^A \sin\phi - \frac{\bar{r}}{\phi}(1 - \cos\phi)\Delta\phi$$

$$u_z^B = u_z^A - \theta_x^A \bar{r}(1 - \cos\phi) - \theta_y^A \bar{r} \sin\phi - \bar{r}\,\phi\,\Delta\psi$$

$$\theta_y^B = \theta_y^A + \Delta\psi$$

$$\theta_z^B = \theta_z^A - \Delta\phi$$

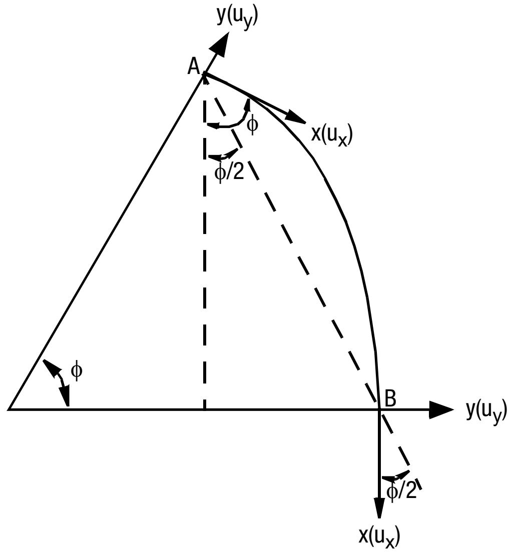


Figure 5-20 Pipe-bend Section, Relative Motion Couplings

Additionally, in order to provide some coupling of the θ_x rotations, the rotation along the secant is made the same at each end:

$$\theta_x^B \; cos\frac{\phi}{2} + \theta_y^B \; sin\frac{\phi}{2} \; = \; \theta_x^A \; cos\frac{\phi}{2} \; \theta_y^A \; sin\frac{\phi}{2}$$

These constraints are transformed internally to corresponding constraints in terms of the six degrees of freedom at A and B in the global coordinate system.

Quick Reference

Type 17

Constant bending, three-node elbow element.

Connectivity

Three nodes per element. Last node is a common point for the bend.

Geometry

For each element section, the geometry is defined as follows:

The pipe thickness is input in the first data field (`EGEOM1`).

The angular extent of the pipe-bend section around the pipe-bend truss is input in the second data field (`EGEOM2`), in degrees.

The radius to the center of the pipe in the r-z plane is input in the third data field (`EGEOM3`).

No coordinates associated with the common node.

Coordinates

At each pipe surface node, the coordinates are those for Element 15:

$$z,\ r\ \frac{dz}{ds},\ \frac{dr}{ds},\ s$$

where s is the distance around the pipe surface.

Degrees of Freedom

The degrees of freedom for the first two nodes are:

$$u,\ v,\ \frac{du}{ds},\ \frac{dv}{ds}$$

For the common third node, they are in-plane stretch and out-of-plane curvature of the section.

Tractions

Distributed load selected with IBODY are as follows:

Load Type (IBODY)	Description
0	Uniform pressure.
1	Uniform load in 1 direction.
2	Nonuniform load in 1 direction.
3	Uniform load in 2 direction.
4	Nonuniform load in 2 direction.
5	Nonuniform pressure.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in the x-, y-, and z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Pressure assumed positive in the direction of normal obtained by rotation of -90° from direction of increasing s (see Figure 5-19).

In the nonuniform cases (`IBODY-3`, `4`, or `5`), the load magnitude must be supplied by the FORCEM user subroutine.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strain on the center line of the element is:

1 = meridional membrane
2 = circumferential membrane
3 = meridional curvature
4 = circumferential curvature

Output of Stresses

Output of stress is at points through the thickness. The stresses are given in pairs as:

1 = meridional stress
2 = circumferential stress

proceeding from the top surface (up the positive local normal, see Figure 5-19) to the bottom surface in equal divisions.

Transformation

All degrees of freedom are transformed to local directions.

Shell Transformation

The shell transformation option type 1 can be used for the pipe surface nodes. It permits an easier application of symmetry conditions in case you want to use a half section only. For a description of this transformation type, see Marc Volume A: Theory and User Information. Note that if the FOLLOW FOR parameter is invoked, the transformations are based on the updated configuration of the element.

Tying

Special tying to form complete pipeline available with Element type 14.

Output Points

Centroid or three Gaussian integration points. The first Gaussian point is closest to first node of the element. The second Gaussian point is at the centroidal section. The third Gaussian point is closest to the second node.

Plotting

This element can be plotted, undeformed or deformed circumference. The tying data must be included with the connectivity and coordinate data.

Notes:	For dynamics, the element mass matrix is only associated with the ovalization degree of freedom in the (r-z) plane. An equivalent straight beam (element type 14) should be used across the elbow segment, with zero stiffness (set Young's modulus to zero) and the same density to obtain the mass terms associated with the beam modes of elbow movement. With this approach, the element provides satisfactory modeling for dynamic as well as static response.
	Adjacent elbow segments should have the same curvature and lies in the same plane. A short straight segment (element 14) can be used as a link in situations where this is not the case.
	As to torus radius increases, the element exhibits less satisfactory behavior; the extreme case of modeling a straight pipe with this element should, therefore, be avoided.

Section Stress Integration

Use the SHELL SECT parameter to set number of points for Simpson rule integration through the thickness. Three points are enough for linear material response. Seven points are enough for simple plasticity or creep analysis. Eleven points are enough for complex plasticity or creep (for example, dynamic plasticity). The default is 11 points.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Element 18

Four-node, Isoparametric Membrane

Element 18 is a four-node, isoparametric, arbitrary quadrilateral written for membrane applications. As a membrane has no bending stiffness, the element is very unstable.

As this element uses bilinear interpolation functions, the strains tend to be constant throughout the element.

In general, you need more of these lower-order elements than the higher-order elements such as 30. Hence, use a fine mesh.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using four-point Gaussian integration.

All constitutive models can be used with this element.

This element is usually used with the LARGE DISP parameter, in which case the (tensile) initial stress stiffness increase the rigidity of the element.

Geometric Basis

The element is defined geometrically by the (x,y,z) coordinates of the four corner nodes. Due to the bilinear interpolation, the surface forms a hyperbolic paraboloid which is allowed to degenerate to a plate. The element thickness is specified in the GEOMETRY option.

The stress output is given in local orthogonal surface directions, $\underset{\sim}{V}_1$, $\underset{\sim}{V}_2$, and $\underset{\sim}{V}_3$, which for the centroid are defined in the following way: (see Figure 5-21).

At the centroid the vectors tangent to the curves with constant isoparametric coordinates are normalized.

$$t_1 = \frac{\partial x}{\partial \xi} / \left|\frac{\partial x}{\partial \eta}\right|,\ t_2 = \frac{\partial x}{\partial \eta} / \left|\frac{\partial x}{\partial \eta}\right|$$

Now a new basis is being defined as:

$$\underset{\sim}{s} = \underset{\sim}{t}_1 + \underset{\sim}{t}_2, \underset{\sim}{d} = \underset{\sim}{t}_1 - \underset{\sim}{t}_2$$

After normalizing these vectors by:

$$\bar{\underset{\sim}{s}} = \underset{\sim}{s} / \sqrt{2}|\underset{\sim}{s}| \quad \bar{\underset{\sim}{d}} = \underset{\sim}{d} / \sqrt{2}|\underset{\sim}{d}|$$

The local orthogonal directions are then obtained as:

$$\underset{\sim}{V}_1 = \bar{\underset{\sim}{s}} + \bar{\underset{\sim}{d}}, \underset{\sim}{V}_2 = \bar{\underset{\sim}{s}} - \bar{\underset{\sim}{d}}, \text{ and } \underset{\sim}{V}_3 = \underset{\sim}{V}_1 \times \underset{\sim}{V}_2$$

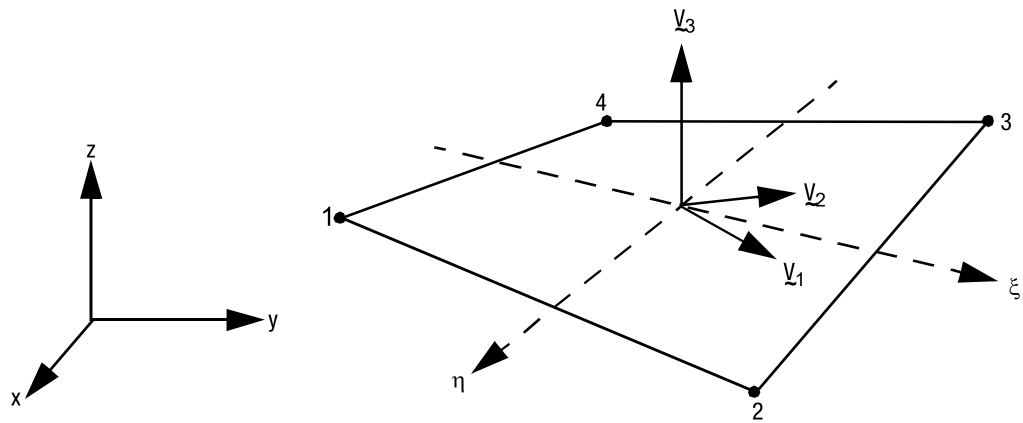


Figure 5-21 Form of Element 18

In this way, the vectors $\frac{\partial \underset{\sim}{x}}{\partial \xi}$, $\frac{\partial \underset{\sim}{x}}{\partial \eta}$ and $\underset{\sim}{V}_1$, $\underset{\sim}{V}_2$ have the same bisecting plane.

The local directions at the Gaussian integration points are found by projection of the centroid directions. Hence, if the element is flat, the directions at the Gauss points are identical to those at the centroid.

Quick Reference

Type 18

Four-node membrane element (straight edge) in three-dimensional space.

Connectivity

Four nodes per element (see Figure 5-21).

Geometry

The thickness is input in the first data field (`EGEOM1`). The other two data fields are not used.

Coordinates

Three global Cartesian coordinates x, y, z.

Degrees of Freedom

Three global degrees of freedom u, v, and w.

Tractions

Four distributed load types are available, depending on the load type definition:

Load Type	Description
1	Gravity load, proportional to surface area, in negative global z direction.
2	Pressure (load per unit area) positive when in direction of normal V_3.
3	Nonuniform gravity load, proportional to surface area, in negative global z direction (use the FORCEM user subroutine).
4	Nonuniform pressure, proportional to surface area, positive when in direction of normal $\underset{\sim}{V}_3$ (use the FORCEM user subroutine).
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in the x-, y-, and z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains and Stresses

Output of stress (σ_{11}, σ_{22}, τ_{12}) and strain (ε_{11}, ε_{22}, γ_{12}) is in the local (V_1, V_2) directions defined above.

Transformation

Nodal degrees of freedom can be transformed to local degrees of freedom.

Output Points

Centroid or four Gaussian integration points (see Figure 5-22).

Notes:	Sensitive to excessive distortions, use essentially a rectangular mesh.
	An eight-node membrane distorted quadrilateral (Element 30) is available and is the preferred element.
	Membrane analysis is extremely difficult due to rigid body modes. For example, a circular cylinder shape is particularly numerically sensitive.
	This element has no bending stiffness.

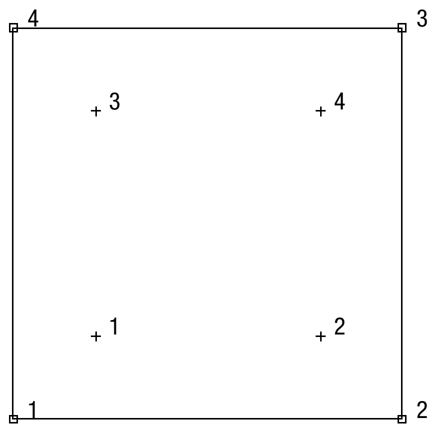


Figure 5-22 Gaussian Integration Points for Element 18

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 39. See Element 39 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered a design variable for this element.

Element 19

Generalized Plane Strain Quadrilateral

This element is an extension of the plane strain isoparametric quadrilateral (Element type 11) to the generalized plane strain case. As this element uses bilinear interpolation functions, the strains tend to be constant throughout the element. This results in a poor representation of shear behavior. The generalized plane strain condition is obtained by allowing two extra nodes in each element. One of these nodes has the single degree of freedom of change of distance between the top and bottom of the structure at one point (i.e., change in thickness at that point) while the other additional node contains the relative rotations θ_{xx} and θ_{yy} of the to p surface with respect to the bottom surface about the same point. The generalized plane strain condition is attained by allowing these two nodes to be shared by all elements forming the structure. Note that this is an extension of the usual generalized plane strain condition which is obtained by setting the relative rotations to zero (by kinematic boundary conditions) and from there the element could become a plane strain element if the relative displacement is also made zero. Note that the sharing of the two extra nodes by all elements implies two full nodal rows in the generated plane strain part of the assembled stiffness matrix considerable computational savings are achieved if these two nodes are given the highest node numbers in that part of the structure.

In general, you need more of these lower-order elements than the higher-order elements such as types 29 or 56. Hence, use a fine mesh.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using four-point Gaussian integration.

For nearly incompressible behavior, including plasticity or creep, it is advantageous to use an alternative integration procedure. This constant dilatation method, which eliminates potential element locking, is flagged through the GEOMETRY option.

This element can be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 82 instead. Element type 81 is also preferable for small strain incompressible elasticity.

These elements cannot be used with the CASI iterative solver.

Quick Reference

Type 19

Generalized plane strain quadrilateral.

Connectivity

Node Numbering:

First four nodes are the corners of the element in the x-y plane, and must proceed counterclockwise around the element when the x-y plane is viewed from the positive z side.

Fifth and sixth nodes are shared by all generalized plane strain elements in this part of the structure. These two nodes should have the highest node numbers in the generalized plane strain part of the structure, to reduce matrix solution time (see Figure 5-23).

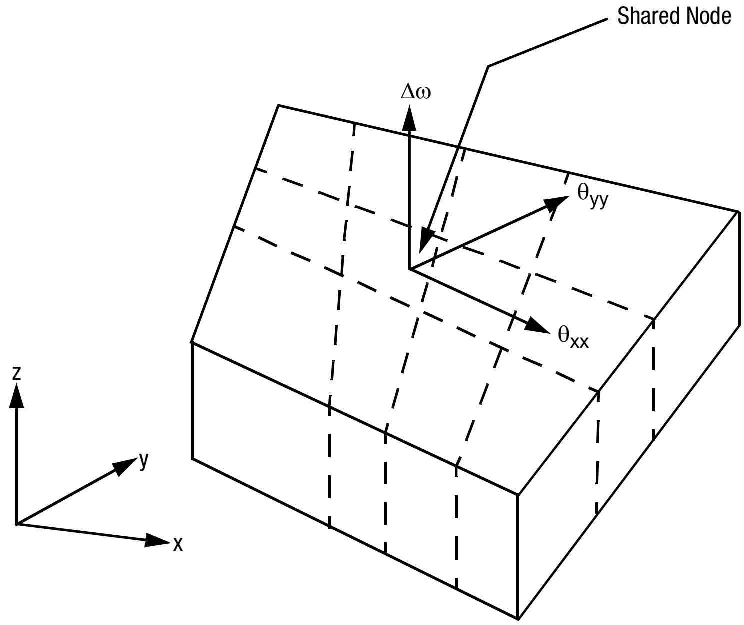


Figure 5-23 Generalized Plane Strain Element

Geometry

The thickness is entered in the first data field (EGEOM1). Default is unit thickness.

If a nonzero value is entered in the second data field (EGEOM2), the volumetric strain is constant throughout the element. This is particularly useful for analysis of approximately incompressible materials, and for analysis of structures in the fully plastic range. It is also recommended for creep problems in which it is attempted to obtain the steady state solution.

Coordinates

Coordinates are X and Y at all nodes. Note the position of the first shared node (node 5 of each element) determines the point where the thickness change is measured. You choose the location of nodes 5 and 6. These nodes should be at the same location in space.

Degrees of Freedom

Global displacement degrees of freedom:

1 = u displacement (parallel to x-axis)
2 = v displacement (parallel to y-axis)

at all nodes except the two shared nodes (nodes 5 and 6 of each element).

For the first shared node (node 5 of each element):

1 = Δw = thickness change at that point
2 – is not used

For the second shared node (node 6 of each element):

$1 = \Delta\theta_{xx}$ = relative rotation of top surface of generalized plane strain section of structure, with respect to its bottom surface, about the x-axis.

$2 = \Delta\theta_{yy}$ = relative rotation of the top surface with respect to the bottom surface about the y-axis.

Distributed Loads

Load types for distributed loads as follows:

Load Type	Description
* 0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force per unit volume in second coordinate direction.
* 3	Nonuniform pressure on 1-2 face of the element; magnitude supplied through the FORCEM user subroutine.
4	Nonuniform body force per unit volume in first coordinate direction; magnitude supplied through the FORCEM user subroutine.
5	Nonuniform body force per unit volume in second coordinate direction; magnitude supplied through the FORCEM. user subroutine
* 6	Uniform pressure on 2-3 face of the element.
* 7	Nonuniform pressure on 2-3 face of the element; magnitude supplied through the FORCEM user subroutine.
* 8	Uniform pressure on 3-4 face of the element.
* 9	Nonuniform pressure on 3-4 face of the element; magnitude supplied through the FORCEM user subroutine.
* 10	Uniform pressure on 4-1 face of the element.
* 11	Nonuniform pressure on 4-1 face of the element; magnitude supplied through the FORCEM user subroutine.
* 20	Uniform shear force on side 1 - 2 (positive from 1 to 2).
* 21	Nonuniform shear force on side 1 - 2; magnitude supplied through the FORCEM user subroutine.
* 22	Uniform shear force on side 2 - 3 (positive from 2 to 3).
* 23	Nonuniform shear force on side 2 - 3; magnitude supplied through the FORCEM user subroutine.
* 24	Uniform shear force on side 3 - 4 (positive from 3 to 4).
* 25	Nonuniform shear force on side 3 - 4; magnitude supplied through the FORCEM user subroutine.
* 26	Uniform shear force on side 4 - 1 (positive from 4 to 1).
* 27	Nonuniform shear force on side 4 - 1; magnitude supplied through the FORCEM user subroutine.

Load Type	Description
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in the x-, y-, and z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

All pressures are positive when directed into the element. Load types shown with an asterisk (*) require the magnitude of the load to be entered as force per unit area. To prescribe these loads in force per unit length, add 50 to the load type. This is often useful in design optimization where the thickness changes, but it is desired that the applied force remain the same. In addition, point loads can be applied at the nodes.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains and Stresses

Output of strain and stress:

$1 = \varepsilon_{xx}\ (\sigma_{xx})$

$2 = \varepsilon_{yy}\ (\sigma_{yy})$

$3 = \varepsilon_{zz}\ (\sigma_{zz})$

$4 = \gamma_{xy}\ (\tau_{xy})$

Transformation

Only in the x-y plane.

Tying

Use the UFORMSN user subroutine.

Output Points

Centroid or four Gaussian integration points.

Updated Lagrange Procedure And Finite Strain Plasticity

Capability is available – stress and strain output in global coordinate directions. Thickness is updated. Reduced volume strain integration is recommended – see Geometry .

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 39. See Element 39 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered a design variable for this element type.

Element 20

Axisymmetric Torsional Quadrilateral

Element type 20 is a four-node, isoparametric, arbitrary quadrilateral written for axisymmetric applications including torsional strains. It is assumed that there are no variations in behavior in the circumferential direction.

As this element uses bilinear interpolation functions, the strains tend to be constant throughout the element. This results in a poor representation of shear behavior.

In general, one needs more of these lower order elements than the higher order elements such as types 67 or 73. Hence, use a fine mesh.

This element is preferred over higher order elements when used in a contact analysis. Note, in a contact analysis, there is no friction associated with the torsion.

The stiffness of this element is formed using four-point Gaussian integration.

For nearly incompressible behavior, including plasticity or creep, it is advantageous to use an alternative integration procedure. This constant dilatation method, which eliminates potential element locking, is flagged through the GEOMETRY option.

This element can be used for all constitutive relations. When using incompressible rubber materials (for example, Mooney and Ogden), the element must be used within the Updated Lagrange framework.

For rubber materials with total Lagrange procedure, element type 83 can be used. This is slightly more expensive because of the extra pressure degrees of freedom associated with element type 83.

Notice that there is no friction contribution in the torsional direction when the CONTACT option is used.

If the analysis involves large rotations about the symmetry axis, it is recommended to use a full 3-D model.

Quick Reference

Type 20

Axisymmetric, arbitrary, ring with a quadrilateral cross section.

Connectivity

Four nodes per element. Numbering in a right-handed manner (counterclockwise).

Geometry

If a nonzero value is entered in the second data field (`EGEOM2`), the volume strain is constant throughout the element. This is particularly useful for analysis of approximately incompressible materials, and for analysis of structures in the fully plastic range. It is also recommended for creep problems in which it is attempted to obtain the steady state solution.

Coordinates

Two coordinates in the global z- and r-directions, respectively.

Degrees of Freedom

Global displacement degrees of freedom:

1 = u displacement (along symmetric axis)
2 = radial displacement
3 = angular displacement about symmetric axis (measured in radians)

Distributed Loads

Load types for distributed loads as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force per unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element; magnitude supplied through the FORCEM. user subroutine
4	Nonuniform body force per unit volume in first coordinate direction; magnitude supplied through the FORCEM user subroutine.
5	Nonuniform body force per unit volume in second coordinate direction; magnitude supplied through the FORCEM user subroutine.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element; magnitude supplied through the FORCEM user subroutine.
8	Uniform pressure on 3-4 face of the element.
9	Nonuniform pressure on 3-4 face of the element; magnitude supplied through the FORCEM user subroutine.
10	Uniform pressure on 4-1 face of the element.
11	Nonuniform pressure on 4-1 face of the element; magnitude supplied through the FORCEM user subroutine.
20	Uniform shear force on side 1 - 2 (positive from 1 to 2).
21	Nonuniform shear force on side 1 - 2; magnitude supplied through the FORCEM user subroutine.
22	Uniform shear force on side 2 - 3 (positive from 2 to 3).
23	Nonuniform shear force on side 2 - 3; magnitude supplied through the FORCEM user subroutine.
24	Uniform shear force on side 3 - 4 (positive from 3 to 4).
25	Nonuniform shear force on side 3 - 4; magnitude supplied through the FORCEM user subroutine.
26	Uniform shear force on side 4 - 1 (positive from 4 to 1).
27	Nonuniform shear force on side 4 - 1; magnitude supplied through the FORCEM user subroutine.

Load Type	Description
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter the magnitude of gravity acceleration in the z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

All pressures are positive when directed into the element. In addition, point loads and torques can be applied at the nodes. The magnitude of concentrated loads must correspond to the load integrated around the circumference.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid of the element in global coordinates:

$1 = \varepsilon_{zz}$

$2 = \varepsilon_{rr}$

$3 = \varepsilon_{\theta\theta}$

$4 = \gamma_{zr}$

$5 = \gamma_{r\theta}$

$6 = \gamma_{\theta z}$

Output of Stresses

Output for stress is the same direction as for Output of Strains .

Transformation

First two degrees of freedom can be transformed to local degrees of freedom.

Tying

Use the UFORMSN user subroutine.

Output Points

Centroid or four Gaussian integration points (see Figure 5-24).

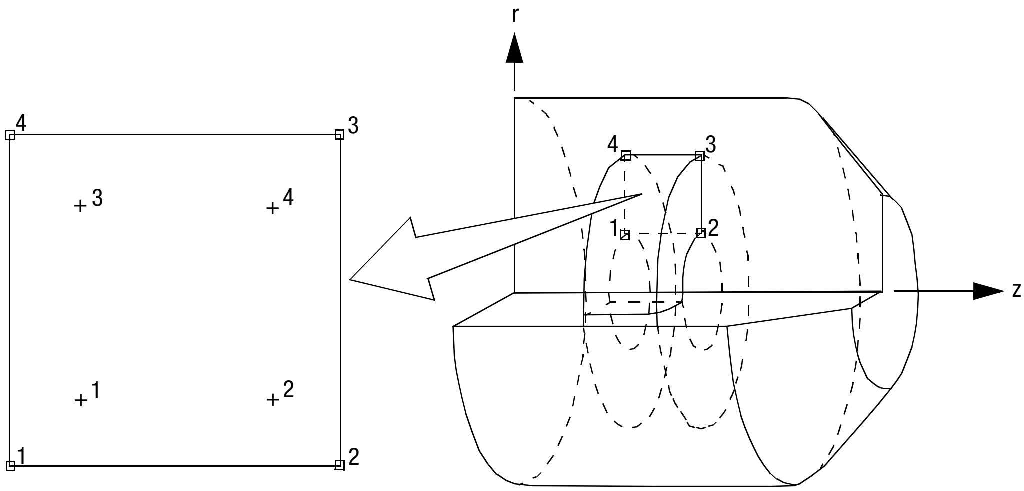


Figure 5-24 Integration Points for Element 20

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinate directions. Reduced volume strain integration is recommended – see Geometry .

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 40. See Element 40 for a description of the conventions used for entering the flux and film data for this element.

Element 21

Three-dimensional 20-node Brick

Element type 21 is a 20-node, isoparametric, arbitrary hexahedral. This element uses triquadratic interpolation functions to represent the coordinates and displacements; hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses.

Lower-order elements, such as type 7, are preferred in contact analyses.

The stiffness of this element is formed using 27-point Gaussian integration.

This element can be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 35 instead. Element type 35 is also preferable for small strain incompressible elasticity.

Note:	Reduction to Wedge or Tetrahedron – By simply repeating node numbers on the same spatial position, the element can be reduced as far as a tetrahedron. Element type 127 is preferred for tetrahedrals.

Quick Reference

Type 21

Twenty-nodes, isoparametric, arbitrary, distorted cube.

Connectivity

Twenty nodes numbered as described in the connectivity write-up for this element and as shown in Figure 5-25.

Geometry

In general, not required. The first field contains the transition thickness when the automatic brick to shell transition constraints are used (see Figure 5-26). In a coupled analysis, there are no constraints for the temperature degrees of freedom.

Coordinates

Three global coordinates in the x-, y-, and z-directions.

Degrees of Freedom

Three global degrees of freedom, u, v and w.

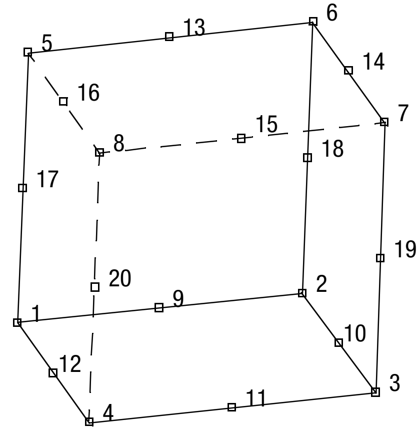


Figure 5-25 Form of Element 21

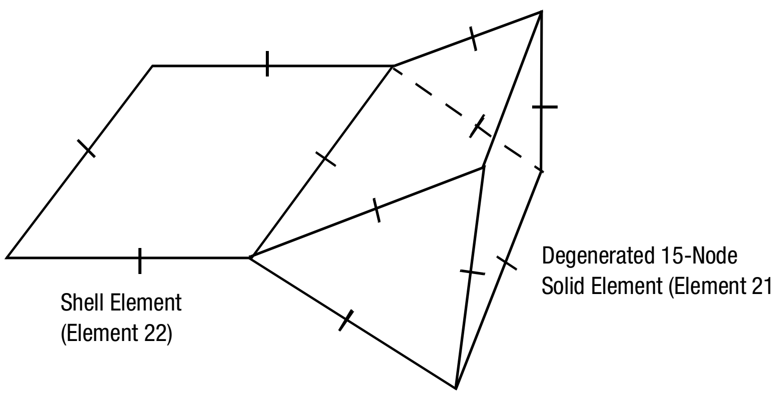


Figure 5-26 Shell-to-solid Automatic Constraint

Distributed Loads

Distributed loads chosen by value of IBODY are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3-4 face.
1	Nonuniform pressure on 1-2-3-4 face; magnitude supplied through the FORCEM user subroutine.
2	Uniform body force per unit volume in -z-direction.
3	Nonuniform body force per unit volume (e.g., centrifugal force); magnitude and direction supplied through the FORCEM user subroutine.
4	Uniform pressure on 6-5-8-7 face.
5	Nonuniform pressure on 6-5-8-7 face.
6	Uniform pressure on 2-1-5-6 face.

Load Type	Description
7	Nonuniform pressure on 2-1-5-6 face.
8	Uniform pressure on 3-2-6-7 face.
9	Nonuniform pressure on 3-2-6-7 face.
10	Uniform pressure on 4-3-7-8 face.
11	Nonuniform pressure on 4-3-7-8 face.
12	Uniform pressure on 1-4-8-5 face.
13	Nonuniform pressure on 1-4-8-5 face.
20	Uniform pressure on 1-2-3-4 face.
21	Nonuniform load on 1-2-3-4 face; magnitude and direction supplied in the FORCEM user subroutine.
22	Uniform body force per unit volume in -z-direction.
23	Nonuniform body force per unit volume (e.g., centrifugal force; magnitude supplied through the FORCEM user subroutine).
24	Uniform pressure on 6-5-8-7 face.
25	Nonuniform load on 6-5-8-7 face; magnitude and direction supplied in the FORCEM user subroutine.
26	Uniform pressure on 2-1-5-6 face.
27	Nonuniform load on 2-1-5-6 face; magnitude and direction supplied in the FORCEM user subroutine.
28	Uniform pressure on 3-2-6-7 face.
29	Nonuniform load on 3-2-6-7 face; magnitude and direction supplied in the FORCEM user subroutine.
30	Uniform pressure on 4-3-7-8 face.
31	Nonuniform load on 4-3-7-8 face; magnitude and direction supplied in the FORCEM user subroutine.
32	Uniform pressure on 1-4-8-5 face.
33	Nonuniform load on 1-4-8-5 face; magnitude and direction supplied in the FORCEM user subroutine.
40	Uniform shear 1-2-3-4 face in 12 direction.
41	Nonuniform shear 1-2-3-4 face in 12 direction.
42	Uniform shear 1-2-3-4 face in 23 direction.
43	Nonuniform shear 1-2-3-4 face in 23 direction.
48	Uniform shear 6-5-8-7 face in 56 direction.
49	Nonuniform shear 6-5-8-7 face in 56 direction.
50	Uniform shear 6-5-8-7 face in 67 direction.
51	Nonuniform shear 6-5-8-7 face in 67 direction.
52	Uniform shear 2-1-5-6 face in 12 direction.
53	Nonuniform shear 2-1-5-6 face in 12 direction.
54	Uniform shear 2-1-5-6 face in 15 direction.

Load Type	Description
55	Nonuniform shear 2-1-5-6 face in 15 direction.
56	Uniform shear 3-2-6-7 face in 23 direction.
57	Nonuniform shear 3-2-6-7 face in 23 direction.
58	Uniform shear 3-2-6-7 face in 26 direction.
59	Nonuniform shear 2-3-6-7 face in 26 direction.
60	Uniform shear 4-3-7-8 face in 34 direction.
61	Nonuniform shear 4-3-7-8 face in 34 direction.
62	Uniform shear 4-3-7-8 face in 37 direction.
63	Nonuniform shear 4-3-7-8 face in 37 direction.
64	Uniform shear 1-4-8-5 face in 41 direction.
65	Nonuniform shear 1-4-8-5 face in 41 direction.
66	Uniform shear 1-4-8-5 face in 15 direction.
67	Nonuniform shear 1-4-8-5 face in 15 direction.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

The FORCEM user subroutine is called once per integration point when flagged. The magnitude of load defined by DIST LOADS is ignored and the FORCEM value is used instead.

For nonuniform body force, force values must be provided for the twenty-seven integration points.

For nonuniform surface pressure, force values need only be supplied for the nine integration points on the face of application. Nodal (concentrated) loads can also be supplied.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

$1 = \varepsilon_{xx}$

$2 = \varepsilon_{yy}$

$3 = \varepsilon_{zz}$

$4 = \gamma_{xy}$

$5 = \gamma_{yz}$

$6 = \gamma_{zx}$

Output of Stresses

Same as for Output of Strains .

Transformation

Three global degrees of freedom can be transformed to local degrees of freedom.

Tying

Use the UFORMSN user subroutine. An automatic constraint is available for brick-to-shell transition meshes. (See Geometry .)

Note:	There is an automatic constraint option for transitions between bricks and shells in element type 22. By collapsing a one-sided plane to a line as shown in Figure 5-26, this transition is created. Thickness of the shell must be specified in the GEOMETRY option of the brick element.

Output Points

Centroid or 27 Gaussian integration points (see Figure 5-27).

Notes:	A large bandwidth results in a lengthy central processing time.
	You should invoke the appropriate OPTIMIZE option in order to minimize the matrix solution time.

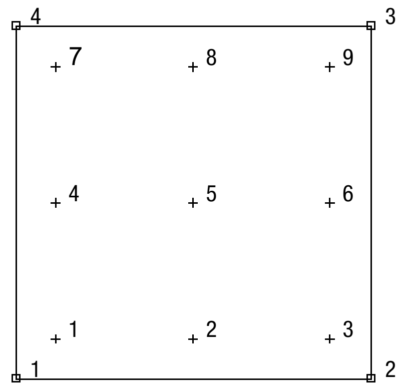


Figure 5-27 Element 21 Integration Plane

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinate directions.

Note:	Distortion of element during analysis can cause bad solution. Element type 7 is preferred.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 44. See Element 44 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux due to dissipation of plastic work specified with type 101.

Element 22

Quadratic Thick Shell Element

Element type 22 is an eight-node thick shell element with global displacements and rotations as degrees of freedom. Second-order interpolation is used for coordinates, displacements and rotations. The membrane strains are obtained from the displacement field; the curvatures from the rotation field. The transverse shear strains are calculated at ten special points and interpolated to the integration points. In this way, this element behaves correctly in the limiting case of thin shells. The element can be degenerated to a triangle by collapsing one of the sides. Tying type 22, which connects shell and solid, is available for this element.

Lower-order elements, such as type 75, are preferred in contact analysis.

The stiffness of this element is formed using four-point Gaussian integration. The mass matrix of this element is formed using nine-point Gaussian integration.

All constitutive relations can be used with this element.

Geometric Basis

The element is defined geometrically by the (x, y, z) coordinates of the four corner nodes and four midside nodes. The element thickness is specified in the GEOMETRY option. The stress output is given with respect to local orthogonal surface directions, $\underset{\sim}{V}_1$, $\underset{\sim}{V}_2$, and $\underset{\sim}{V}_3$ for which each integration point is defined in the following way (see Figure 5-28).

At each of the integration points, the vectors tangent to the curves with constant isoparametric coordinates are normalized:

$$\underset{\sim}{t}_1 = \frac{\partial \underset{\sim}{x}}{\partial \xi} \bigg/ \left|\frac{\partial x}{\partial \eta}\right|, \ \underset{\sim}{t}_2 = \frac{\partial \underset{\sim}{x}}{\partial \eta} \bigg/ \left|\frac{\partial x}{\partial \eta}\right|$$

Now, a new basis is being defined as follows:

$$\underset{\sim}{s} = \underset{\sim}{t}_1 + \underset{\sim}{t}_2, \underset{\sim}{d} = \underset{\sim}{t}_1 - \underset{\sim}{t}_2$$

After normalizing these vectors by $\bar{\underset{\sim}{s}} = \underset{\sim}{s} / \sqrt{2}|\underset{\sim}{s}| \quad \bar{\underset{\sim}{d}} = \underset{\sim}{d} / \sqrt{2}|\underset{\sim}{d}|$, the local orthogonal directions are then obtained as follows:

$$\underset{\sim}{V}_1 = \bar{\underset{\sim}{s}} + \bar{\underset{\sim}{d}}, \ \underset{\sim}{V}_2 = \bar{\underset{\sim}{s}} - \bar{\underset{\sim}{d}}, \text{ and } \ \underset{\sim}{V}_3 = \underset{\sim}{V}_1 \times \underset{\sim}{V}_2$$

In this way, the vectors $\frac{\partial \underset{\sim}{x}}{\partial \xi}, \frac{\partial \underset{\sim}{x}}{\partial \eta}$ and $\underset{\sim}{V}_1$, $\underset{\sim}{V}_2$ have the same bisecting plane.

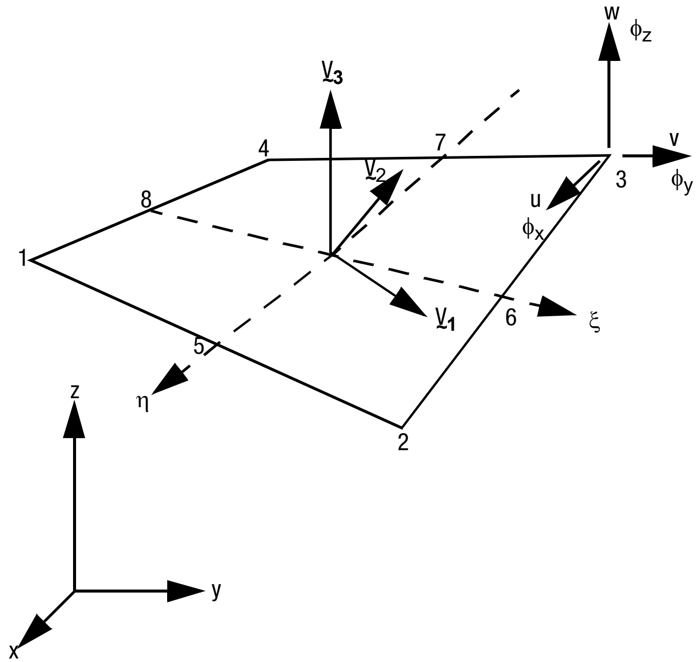


Figure 5-28 Form of Element 22

Displacements

The six nodal displacement variables are as follows:

u, v, w Displacement components defined in global Cartesian x,y,z coordinate system.
ϕ_x, ϕ_y, ϕ_z Rotation components about global x-, y-, and z-axis, respectively.

Quick Reference

Type 22

Bilinear, eight-node shell element including transverse shear effects.

Connectivity

Eight nodes per element. The element can be collapsed to a triangle.

Geometry

Bilinear thickness variation is allowed in the plane of the element. Thicknesses at first, second, third, and fourth nodes of the element are stored for each element in the first (`EGEOM1`), second (`EGEOM2`), third (`EGEOM3`) and fourth (`EGEOM4`), geometry data fields, respectively. If `EGEOM2 = EGEOM3 = EGEOM4 = 0`, then a constant thickness (`EGEOM1`) is assumed for the element.

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

If the element needs to be offset from the user-specified position, the eighth data field (EGEOM8) is set to -2.0. In this case, an extra line containing the offset information is provided (see the GEOMETRY option in Marc Volume C: Program Input). The offset magnitudes along the element normal for the four corner nodes are provided in the first, second, third, and fourth data fields of the extra line. If the interpolation flag (fifth data field) is set to 1, then the offset values at the midside nodes are obtained by interpolating from the corresponding corner values. A uniform offset for all nodes can be set by providing the offset magnitude in the first data field and then setting the constant offset flag (sixth data field of the extra line) to 1.

Coordinates

Three coordinates per node in the global x-, y-, and z-direction.

Degrees of Freedom

Six degrees of freedom per node:

1 = u = global (Cartesian) x-displacement
2 = v = global (Cartesian) y-displacement
3 = w = global (Cartesian) z-displacement
4 = ϕ_x = rotation about global x-axis
5 = ϕ_y = rotation about global y-axis
6 = ϕ_z = rotation about global z-axis

Distributed Loads

Distributed load types follow below:

Load Type	Description
1	Uniform gravity load per surface area in -z-direction.
2	Uniform pressure with positive magnitude in -V_3-direction.
3	Nonuniform gravity load per surface area in -z-direction; magnitude given in the FORCEM user subroutine.
4	Nonuniform pressure with positive magnitude in -V_3-direction; magnitude given in the FORCEM user subroutine.
5	Nonuniform load per surface area in arbitrary direction; magnitude given in the FORCEM user subroutine.
11	Uniform edge load in the plane of the surface on the 1-2 edge and perpendicular to this edge.
12	Nonuniform edge load; magnitude given in the FORCEM user subroutine in the plane of the surface on the 1-2 edge.
13	Nonuniform edge load; magnitude and direction given in the FORCEM user subroutine on 1-2 edge.
14	Uniform load on the 1-2 edge of the shell, tangent to, and in the direction of the 1-2 edge
15	Uniform load on the 1-2 edge of the shell. Perpendicular to the shell; i.e., -v_3 direction
21	Uniform edge load in the plane of the surface on the 2-3 edge and perpendicular to this edge.
22	Nonuniform edge load; magnitude given in the FORCEM user subroutine in the plane of the surface on 2-3 edge.

Load Type	Description
23	Nonuniform edge load; magnitude and direction given in the FORCEM user subroutine on 2-3 edge.
24	Uniform load on the 2-3 edge of the shell, tangent to and in the direction of the 2-3 edge
25	Uniform load on the 2-3 edge of the shell. Perpendicular to the shell; i.e., $-v_3$ direction
31	Uniform edge load in the plane of the surface on the 3-4 edge and perpendicular to this edge.
32	Nonuniform edge load; magnitude given in the FORCEM user subroutine in the plane of the surface on 3-4 edge.
33	Nonuniform edge load; magnitude and direction given in the FORCEM user subroutine on 3-4 edge.
34	Uniform load on the 3-4 edge of the shell, tangent to and in the direction of the 3-4 edge
35	Uniform load on the 3-4 edge of the shell. Perpendicular to the shell; i.e., $-v_3$ direction
41	Uniform edge load in the plane of the surface on the 4-1 edge and perpendicular to this edge.
42	Nonuniform edge load; magnitude given in the FORCEM user subroutine in the plane of the surface on 4-1 edge.
43	Nonuniform edge load; magnitude and direction given in the FORCEM user subroutine on 4-1 edge.
44	Uniform load on the 4-1 edge of the shell, tangent to and in the direction of the 4-1 edge
45	Uniform load on the 4-1 edge of the shell. Perpendicular to the shell; i.e., $-v_3$ direction
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter 3 magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

All edge loads require the input as force per unit length.

Point loads and moments can also be applied at the nodes.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Generalized strain components are as follows:

Middle surface stretches:	ε_{11} ε_{22} ε_{12}
Middle surface curvatures:	κ_{11} κ_{22} κ_{12}
Transverse shear strains:	γ_{23} γ_{31}

in local (V_1 , V_2 , V_3) system.

Output of Stresses

σ_{11}, σ_{22}, σ_{12}, σ_{23}, σ_{31} in local (V_1 , V_2 , V_3) system given at equally spaced layers though thickness. First layer is on positive V_3 direction surface.

Transformation

Displacement and rotation at each node can be transformed to local direction.

Tying

Use the UFORMSN user subroutine.

Updated Lagrange Procedure and Finite Strain Plasticity

Updated Lagrange capability is available. Note, however, that since the curvature calculation is linearized, you have to select your loadsteps such that the rotation remains small within a load step. Thickness is updated if the LARGE STRAIN parameter is specified.

Section Stress – Integration

Integration through the shell thickness is performed numerically using Simpson's rule. Use the SHELL SECT parameter to specify the number of integration points. This number must be odd.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 85. See Element 85 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux due to dissipation of plastic work specified with type 101.

Design Variables

The thickness can be considered as a design variable.

Element 23

Three-dimensional 20-node Rebar Element

This element is an isoparametric, three-dimensional, 20-node empty brick in which you can place single strain members such as reinforcing rods or cords (i.e., rebars). The element is then used in conjunction with the 20-node brick continuum element (e.g., elements 21, 35, 57, or 61) to represent cord reinforced composite materials. This technique allows the rebar and the filler to be represented accurately with respect to their stress distribution, so that separate constitutive theories can be used in each (e.g., cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via the REBAR option or via the REBAR user subroutine.

Integration

It is assumed that several “layers” of rebars are presented. The number of such “layers” is input by you via the REBAR option or, if the REBAR user subroutine is used, in the second element geometry field. A maximum number of five layers can be used within a rebar element. Each layer is assumed to be similar to a pair of opposite element faces (although the rebar direction is arbitrary), so that the “thickness” direction is from one of the element faces to its opposite one. For instance (see Figure 5-29), if the layer is similar to the 1, 2, 3, 4 and 5, 6, 7, 8 faces of the element, the “thickness” direction is from the 1, 2, 3, 4 face to 5, 6, 7, 8 face of the element. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains four integration points (see Figure 5-29). At each such integration point on each layer, you must input (via either the REBAR option or the REBAR user subroutine) the position, equivalent thickness (or, alternatively, spacing and area of cross section), and orientation of the rebars. See Marc Volume C: Program Input for REBAR option or Marc Volume D: User Subroutines and Special Routines for the REBAR user subroutine.

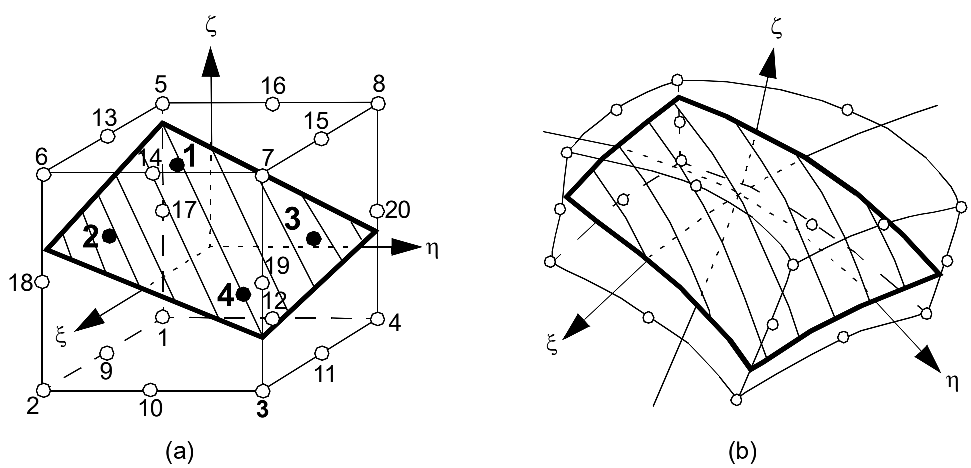


Figure 5-29 Twenty-node Rebar Element

Quick Reference

Type 23

Twenty-node, isoparametric rebar element to be used with 20-node brick continuum element.

Connectivity

Twenty nodes per element. Node numbering of the element is same as that for elements 21, 35, 57, or 61.

Geometry

The number of rebar layers and the isoparametric direction of the layers are defined using the REBAR model definition option.

Coordinates

Three global coordinates in the x-, y-, and z-directions.

Degrees of Freedom

Displacement output in global components is as follows:

1 – u
2 – v
3 – w

Tractions

Point loads can be applied at the nodes, but no distributed loads are available. Distributed loads are applied only to corresponding 20-node brick elements (e.g., element types 21, 35, 57, or 61).

Output of Strains and Stresses

One stress and one strain are output at each integration point – the axial rebar value. For the case of large deformations, the stress is the second Piola-Kirchhoff stress and the strain is the Green strain.

By using post code 471 and 481 (representing Second Piola Kirchhoff stress in undeformed configuration and Cauchy stress in deformed configuration, respectively), the rebar stress can be written into the post file in the form of a stress tensor defined in the global coordinate directions. Mentat can be used to plot the principal directions of the stress tensor to show the magnitude of rebar stress, rebar orientation, and their changes based on deformation.

Transformation

Any local set (u,v,w) can be used at any node.

Special Consideration

Either the REBAR option or the REBAR user subroutine is needed to input the position, size, and orientation of the rebars.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Element 24

Curved Quadrilateral Shell Element

This is an arbitrary, double-curved, isoparametric, quadrilateral shell element, based on De Veubeke interpolation function (Marc Volume A: Theory and User Information). The element is based on Koiter-Sander's shell theory. It is a fully compatible, complete thin-shell element. Because of its isoparametric form, it represents rigid body modes exactly, and is well suited to shell analysis. The element can have any (straight-sided) quadrilateral shape in the mapped surface coordinate $(\theta^1 - \theta^2)$ plane with the restriction that the quadrilateral must not be re-entrant. Any surface coordinate system can be used. The element is extremely rapidly convergent. Note that the generation of the large displacement stiffness terms requires a lot of computing. For this reason, the element is not recommended for use in large displacement problems. This element cannot be used with the CONTACT option.

Geometry

The element is isoparametric, so the surface used for analysis is interpolated from nodal coordinates. The mesh is defined in the $\theta^1 - \theta^2$ plane of surface coordinates. Then the surface in (x, y, z) space is interpolated from the following coordinate set:

$$\theta^1,\ \theta^2,\ x,\ \frac{\partial x}{\partial \theta^1},\ \frac{\partial x}{\partial \theta^2},\ y,\ \frac{\partial y}{\partial \theta^1},\ \frac{\partial y}{\partial \theta^2},\ z,\ \frac{\partial z}{\partial \theta^1},\ \frac{\partial z}{\partial \theta^2}$$

at all nodes, where (x, y, z) are rectangular, Cartesian coordinates and (θ^1, θ^2) are surface coordinates.

Usually, the mesh is first defined in the (θ^1, θ^2) plane, using, say, the MESH2D option. Then a suitable mapping generates the remaining coordinates, either using the FXORD option for default surface types (Marc Volume A: Theory and User Information), or through the UFXORD user subroutine.

The thickness of the element is given in `EGEOM1`. Note that tying must be used when shells of different thickness come together.

Displacements

The following set forms the nodal displacement unknowns:

Corner Nodes: $u,\ \frac{\partial u}{\partial \theta^1},\ \frac{\partial u}{\partial \theta^2},\ v,\ \frac{\partial v}{\partial \theta^2},\ \frac{\partial v}{\partial \theta^2},\ w,\ \frac{\partial w}{\partial \theta^1},\ \frac{\partial w}{\partial \theta^2}$ (nine degrees of freedom)

Midside Nodes: $\frac{\partial u}{\partial n},\ \frac{\partial v}{\partial n},\ \frac{\partial w}{\partial n}$ (3 degrees of freedom)

where (u, v, w) are the Cartesian components of displacement and n is normal distance in the $\theta^1 - \theta^2$ plane; n has positive projection on the θ^1 axis $(n^1>0)$ or, if $n^1 = 0$, $n^2>0$.

The De Veubeke interpolation function ensures continuity of all displacement components and their first derivatives $\frac{\partial u_i}{\partial \theta^1}, \frac{\partial u_i}{\partial \theta^2}$ between elements when the above sets of nodal degrees of freedom take identical values at shared nodes on element edges.

Care should be taken with the application of kinematic boundary conditions since they must be fully, but not over-fully, specified and in the application of edge moments, so that they are conjugate to the appropriate degrees of freedom and so generate a mechanical work rate.

Connectivity

The node numbering order is given in Figure 5-30. Corners should be numbered first, in counterclockwise order in the $\theta^1 - \theta^2$ plane, then midside nodes, the fifth node being between the first and second, etc.

Numerical Integration

The element uses a 28-point integration scheme, based on a 7-point, fifth order scheme in each of the contributing triangles. The locations of the points are shown in Figure 5-31. The distance ratios are $(1 + \sqrt{15})/7$ and $(1 - \sqrt{15})/7$ of the distance from the centroid to a vertex in each triangle.

Note that the density of integration points takes full advantage of the high order of the element.

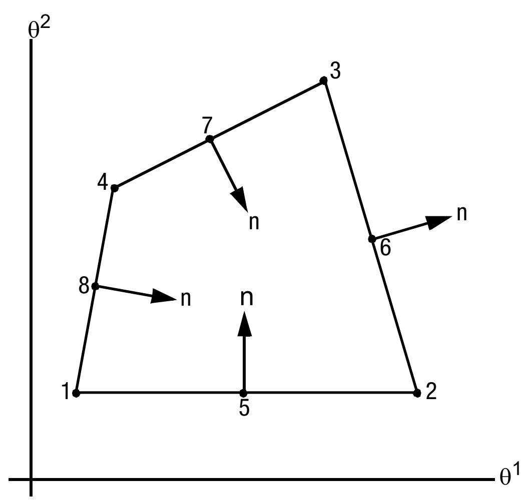


Figure 5-30 Definitions of the Positive Midside Normal Direction

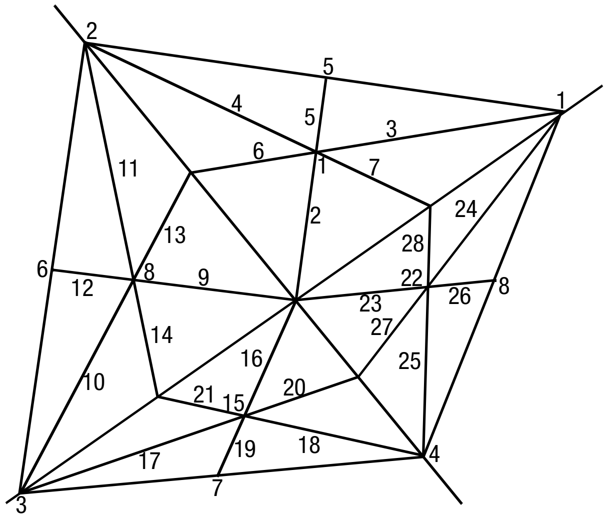


Figure 5-31 Integration Points for Element 24 (not to scale)

Quick Reference

Type 24

Arbitrary, doubly-curved, quadrilateral shell element.

Connectivity

Eight nodes per element. Four corner nodes and four midside nodes numbered as shown in Figure 5-30.

Geometry

The element thickness is input in the first data field (EGEOM1). The other two data fields are not used.

Coordinates

Eleven coordinates for nodes:

$$1 = \theta^1 ; 2 = \theta^2$$

$$3 = x ; \quad 4 = \frac{\partial x}{\partial \theta^1} ; \quad 5 = \frac{\partial x}{\partial \theta^2}$$

$$6 = y ; \quad 7 = \frac{\partial y}{\partial \theta^1} ; \quad 8 = \frac{\partial y}{\partial \theta^2}$$

$$9 = z ; \quad 10 = \frac{\partial z}{\partial \theta^1} ; \quad 11 = \frac{\partial z}{\partial \theta^2}$$

This coordinate set is input at all nodes.

Degrees of Freedom

Nine degrees of freedom for the corner nodes:

$$1 = u; \quad 2 = \frac{\partial u}{\partial \theta^1}; \quad 3 = \frac{\partial u}{\partial \theta^2}$$

$$4 = v; \quad 5 = \frac{\partial v}{\partial \theta^1}; \quad 6 = \frac{\partial v}{\partial \theta^2}$$

$$7 = w; \quad 8 = \frac{\partial w}{\partial \theta^2}; \quad 9 = \frac{\partial w}{\partial \theta^2}$$

Three degrees of freedom for the midside nodes:

$$1 = \frac{\partial u}{\partial n}; \quad 2 = \frac{\partial v}{\partial n}; \quad 3 = \frac{\partial w}{\partial n}$$

n = normal to side in $\theta^1 - \theta^2$ plane, directed such that the projection of n on the θ^1 axis is positive, or, if the side is parallel to the θ^1-axis, n is in the direction of increasing θ^1.

Tractions

Point loads on nodes (care should be used with moments, since derivative degrees of freedom cannot measure rotation in radians).

Distributed Loads

Type 1	Uniform self-weight (magnitude = load per unit surface area) in negative z-direction.
Type 2	Uniform pressure in negative surface normal direction (surface normal $\underset{\sim}{a}_3 = \underset{\sim}{a}_1 \times \underset{\sim}{a}_2$, where $\underset{\sim}{a}_1$ and $\underset{\sim}{a}_2$ are base vectors along θ^1 and θ^2 surface coordinate lines).
Type 3	Nonuniform pressure in negative surface normal direction; magnitude specified by the FORCEM user subroutine.
Type 4	Nonuniform load unit volume in arbitrary direction; magnitude and direction defined by the FORCEM user subroutine.
	This subroutine is called once per integration point for all elements of type 24 listed with load type 4. For these elements the magnitude supplied in the DIST LOADS option (associated with load type 4) is overwritten by the value defined in the subroutine.
Types 11-43	These loads represent edge loads per unit length, applied in any of the global (x, y, z) directions on any of the four edges of the element. The first digit of the load type chooses the edge, the second chooses the global force directions as follows:

Load Type	Description
11	Uniform load per unit length on the 1-2 edge in the x-direction.
12	Uniform load per unit length on the 1-2 edge in the y-direction.

Load Type	Description
13	Uniform load per unit length on the 1-2 edge in the z-direction.
21	Uniform load per unit length on the 2-3 edge in the x-direction.
22	Uniform load per unit length on the 2-3 edge in the y-direction.
23	Uniform load per unit length on the 2-3 edge in the z-direction.
31	Uniform load per unit length on the 3-4 edge in the x-direction.
32	Uniform load per unit length on the 3-4 edge in the y-direction.
33	Uniform load per unit length on the 3-4 edge in the z-direction.
41	Uniform load per unit length on the 4-1 edge in the x-direction.
42	Uniform load per unit length on the 4-1 edge in the y-direction.
43	Uniform load per unit length on the 4-1 edge in the z-direction.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Six generalized strain components, three surface stretches and three surface curvature changes, are given as components with respect to the surface coordinate system.

Output of Stresses

σ^{11}, σ^{22}, σ^{12}, physical components of stress in the $(\theta^1 - \theta^2)$ directions at points through the thickness of the shell. First point is on surface in the direction of positive surface normal. The last point is on opposite surface.

Transformation

Cartesian components of displacement and displacement derivatives can be transformed to local directions. Surface coordinate directions remain unchanged.

Special Transformation

The shell transformation options type 2 (for the corner nodes) and type 3 (for the midside nodes) can be used to permit easier application of point loads, moments and/or boundary conditions on a node. For a description of these transformation types, see Marc Volume A: Theory and User Information. Note that if the FOLLOW FOR parameter is invoked, the transformation is based on the updated configuration of the element.

Tying

No special type of tying. Use the UFORMSN user subroutine.

Output Points

Centroid or 28 integration points as shown in Figure 5-31.

Notes:	Quadrilaterals must not be re-entrant.
	This is a very expensive element.
	For buckling analysis, elements 4 and 8 are preferable due to the expense of element 24.

Section Stress Integration

Use the SHELL SECT parameter to set number of points for Simpson rule integration through the thickness.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – output of stress and strain as for total Lagrangian approach. Thickness is updated.

Note:	Shell theory only applies if strain variation over the thickness is small.

Coupled Analysis

Coupled analysis is not available for this element. Use element types 22, 72, 75, or 139.

Element 25

Thin-walled Beam in Three-dimensions

This is a straight beam element with no warping of the section, but including twist. It is obtained by modifying element type 14 to give a linear variation of strain along its axis. This improves the element for large displacement analysis and for cases where linear axial strain is necessary (for example, thermal gradient along the axis).

The degrees of freedom associated with each node are three global displacements and three global rotations, all defined in a right-handed convention. In addition, a seventh degree of freedom measures the rates of change of displacement along the beam axis. The generalized strains are stretch, two curvatures and twist per unit length. Stress is direct axial and shear given at each point of the cross section. The local coordinate system which establishes the orientation of the cross section is defined in GEOMETRY fields 4, 5, and 6. Using the GEOMETRY option, a vector in the plane of the local x-axis must be specified. If no vector is defined through this option, the local coordinate system can alternatively be defined by giving the coordinates of a point in space which locates the local x-axis of the cross section. This axis lies in the plane defined by the beam nodes and this point, pointing from the beam towards this point. The default cross section is circular. You can specify alternative cross sections through the BEAM SECT parameter.

All constitutive relations can be used with this element.

Quick Reference

Type 25

Closed-section beam; Euler-Bernoulli theory.

Connectivity

Two nodes per element (see Figure 5-32).

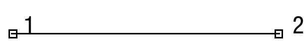


Figure 5-32 Two-node Closed-section Beam

Geometry

For the default section of a hollow circular cylinder, the first data field is for the thickness (EGEOM1).

For noncircular section, set EGEOM1 to 0.

For circular section, set EGEOM2 to the radius.

For noncircular section, set EGEOM2 to the section number needed.

EGEOM4-EGEOM6:	Components of a vector in the plane of the local x-axis and the beam axis. The local x-axis lies on the same side as the specified vector.

If beam-to-beam contact is switched on (see CONTACT option), the radius used when the element comes in contact with other beam or truss elements must be entered in the seventh data field (EGEOM7).

For segment-segment beam contact, various flags that control the creation of patches are entered in the seventh data file (EGEOM7).

```
EGEOM7 = real(100 * NSEG + 10 * IPTCH + IESCAP)
```

where

`NSEG` = number of segments for hollow tubes (default = 32)		
`IPTCH` = patch flag for hollow tubes		
	1	Both outer and inner patches (default).
	3	Only outer patches (at radius + 1/2 * thickness).
	-3	Only outer patches (at radius - i.e., ignore thickness).
	5	Only inner patches (at radius - 1/2 * thickness).
	-5	Only inner patches (at radius - i.e., ignore thickness).
`IESCAP` = end cap/side cap flag		
	0	No end or side cap (default).
	1	End cap only.
	2	Side cap only.
	3	Average side only.
	4	End and side cap.
	5	End cap and average side.

Beam Offsets Pin Codes and Local Beam Orientation

If an offset is applied to the beam geometry or pin codes are applied to the degrees of freedom of the end point of the beams or the beam axis is defined with respect to a local coordinate system, then a code is placed in the 8th field of the 3rd data block of the GEOMETRY option (`EGEOM8`). This code is formed by the negative sum of ioffset, iorien, and ipin where:

ioffset	=	0	-	no offset
		1	-	beam offset
iorien	=	0	-	conventional definition of local beam orientation; beam axis given in 4th through 6th field with respect to global system
		10	-	the local beam orientation entered in the 4th through 6th field is given with respect to the coordinate system of the first beam node. This vector is transformed to the global coordinate system before any further processing.
ipin	=	0	-	no pin codes are used
		10 0	-	pin codes are used.

If ioffset = 1, an additional line is read in the GEOMETRY option (4th data block).

If ipin = 100, an additional line is read in the GEOMETRY option (5th data block)

In this case, an extra line containing the offset information is provided (see the GEOMETRY option in Marc Volume C: Program Input). The offset vector at the first corner node is provided in the first three fields and the offset vector at the second corner node is provided in the next three fields of the extra line. The coordinate system used for the offset vectors at the two nodes are indicated by the corresponding flags in the eighth and ninth fields of the extra line. A flag setting of 0 indicates global coordinate system, 1 indicates local element coordinate system, 2 indicates local shell normal, and 3 indicates local nodal coordinate system. When the flag setting is 1, the local beam coordinate system is constructed by taking the local z axis along the beam, the local x axis as the user specified vector (obtained from EGEOM4-EGEOM6) and the local y axis as perpendicular to both. The flag setting of 2 can be used when the beam nodes are also attached to a shell. In this case, the offset magnitude at the first and second nodes are specified in the first and fourth data fields of the extra line, respectively. The automatically calculated shell normals at the nodes are used to construct the offset vectors.

Coordinates

The first three coordinates at each node are the global (x,y,z) coordinates.

The fourth, fifth, and sixth coordinates are the (x,y,z) coordinates of a point in space which locates the local x-axis of the cross section. This axis lies in the plane defined by the beam nodes and this point. The local x-axis is normal to the beam vector and is positive moving from the beam vector to the point. The fourth, fifth, and sixth coordinates is only used if the local x-axis direction is not specified in the GEOMETRY option.

Degrees of Freedom

1 = u
2 = v
3 = w
4 = θ_x
5 = θ_y
6 = θ_z
7 = $d\bar{u}/ds$ (local)

Tractions

Distributed load types are as follows:

Load Type	Description
1	Uniform load per unit length in global x-direction.
2	Uniform load per unit length in global y-direction.
3	Uniform load per unit length in global z-direction.
4	Nonuniform load per unit length, with magnitude and direction supplied via the FORCEM. user subroutine
11	Fluid drag/buoyancy loading – fluid behavior specified in the FLUID DRAG model definition option.

Load Type	Description
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Point loads and moments can be applied at the nodes.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Generalized strains:

1 = ε_{zz} = axial stretch
2 = κ_{xx} = curvature about local x-axis of cross section.
3 = κ_{yy} = curvature about local y-axis of cross section.
4 = γ = twist

Output of Stresses

Generalized stresses:

1 = axial stress
2 = local xx-moment
3 = local yy-moment
4 = axial torque

Transformation

Displacements and rotations at the nodes can be transformed to a local coordinate reference.

Tying

Use tying type 53 for fully moment-carrying joint. Use tying type 52 for pinned joint.

Output Points

Centroid or three Gaussian integration points.

Special Considerations

The seventh degree of freedom is only shared between two adjacent elements when the beam section and properties are the same for both elements. Tying should be used in other cases. Elements of types 13, 14, 52, 76, 77, 78, 79, or 98 can be used together directly.

For all beam elements, the default printout gives section forces and moments, plus stress at any layer with plastic, creep strain or nonzero temperature. This default printout can be changed via the PRINT ELEMENT option.

Updated Lagrange Procedure and Finite Strain Plasticity

Updated Lagrange procedure is available for this element. This element does not have a finite strain capability.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 36. See Element 36 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

For the default hollow circular section only, the wall thickness and/or the radius can be considered as design variables.

Element 26

Plane Stress, Eight-node Distorted Quadrilateral

Element type 26 is an eight-node, isoparametric, arbitrary quadrilateral written for plane stress applications. This element uses biquadratic interpolation functions to represent the coordinates and displacements. This allows for a more accurate representation of the strain fields in elastic analyses than lower order elements.

Lower-order elements, such as type 3, are preferred in contact analyses.

The stiffness of this element is formed using eight-point Gaussian integration.

All constitutive models can be used with this element.

Quick Reference

Type 26

Second-order, isoparametric, distorted quadrilateral. Plane stress.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention). Then the fifth node between first and second; the sixth node between second and third, etc. See Figure 5-33.

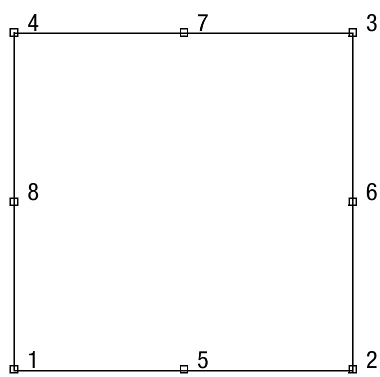


Figure 5-33 Nodes of Eight-node, 2-D Element

Geometry

Thickness stored in first data field (EGEOM1). Default thickness is unity. Other fields are not used.

Coordinates

Two global coordinates, x and y, at each node.

Degrees of Freedom

Two at each node:

1 = u = global x-direction displacement
2 = v = global y-direction displacement

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type (IBODY)	Description
* 0	Uniform pressure on 1-5-2 face.
* 1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.
4	Uniform body force in y-direction.
5	Nonuniform body force in the y-direction.
* 6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
* 7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
* 8	Uniform pressure on 2-6-3 face.
* 9	Nonuniform pressure on 2-6-3 face.
* 10	Uniform pressure on 3-7-4 face.
* 11	Nonuniform pressure on 3-7-4 face.
* 12	Uniform pressure on 4-8-1 face.
* 13	Nonuniform pressure on 4-8-1 face.
* 20	Uniform shear force on 1-2-5 face in the 1⇒2⇒5 direction.
* 21	Nonuniform shear force on side 1-5-2.
* 22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
* 23	Nonuniform shear force on side 2-6-3.
* 24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
* 25	Nonuniform shear force on side 3-7-4.
* 26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
* 27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Load Type (IBODY)	Description
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

All pressures are positive when directed into the element. Load types shown with an asterisk (*) require the magnitude of the load to be entered as force per unit area. To prescribe these loads in force per unit length, add 50 to the load type. This is often useful in design optimization where the thickness changes, but it is desired that the applied force remain the same. For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid or element integration points (see Figure 5-34 and Output Points) in the following order:

1 = ε_{xx}, direct

2 = ε_{yy}, direct

3 = γ_{xy}, shear

Note:	Although $\varepsilon_{zz} = \frac{-\nu}{E}(\sigma_{xx} + \sigma_{yy})$, it is not printed and is posted as 0 for isotropic materials. For Mooney or Ogden (TL formulation) Marc post code 49 provides the thickness strain for plane stress elements. See Marc Volume A: Theory and User Information , Chapter 12 Output Results, Element Information for von Mises intensity calculation for strain.

Output of Stresses

Output of stresses is the same as Output of Strains .

Transformation

Only in x-y plane.

Tying

Use the UFORMSN user subroutine.

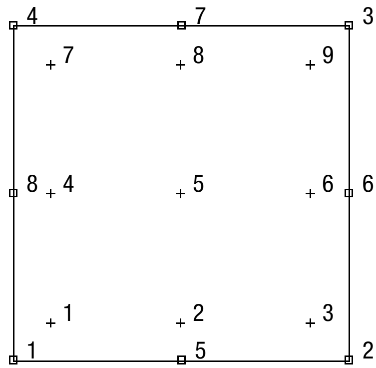


Figure 5-34 Integration Points of Eight-node, 2-D Element

Output Points

If the CENTROID parameter is used, the output occurs at the centroid of the element, given as point 5 in Figure 5-34.

If the ALL POINTS parameter is used, nine output points are given, as shown in Figure 5-34. This is the usual option for a second order element.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – output of stress and strain in global coordinates. Thickness is updated.

Note:	Distortion of element during analysis can cause bad solution. Element type 3 is preferred.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 41. See Element 41 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered a design variable for this element.

Element 27

Plane Strain, Eight-node Distorted Quadrilateral

Element type 27 is an eight-node, isoparametric, arbitrary quadrilateral written for plane strain applications. This element uses biquadratic interpolation functions to represent the coordinates and displacements; hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses.

Lower-order elements, such as type 11, are preferred in contact analyses.

The stiffness of this element is formed using nine-point Gaussian integration.

This element can be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 32 instead. Element type 32 is also preferable for small strain incompressible elasticity.

Quick Reference

Type 27

Second-order, isoparametric, distorted quadrilateral. Plane strain.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention). Then the fifth node between first and second; the sixth node between second and third, etc. See Figure 5-35.

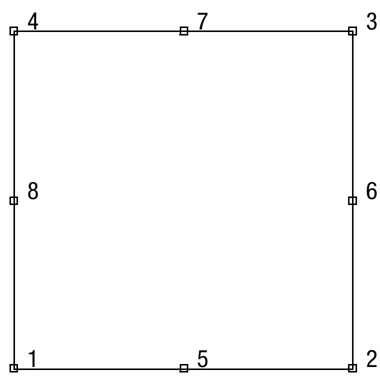


Figure 5-35 Nodes of Eight-node, 2-D Element

Geometry

Thickness stored in first data field (EGEOM1). Default thickness is unity. Other fields are not used.

Coordinates

Two global coordinates, x and y, at each node.

Degrees of Freedom

Two at each node:

1 = u = global x-direction displacement
2 = v = global y-direction displacement

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.
4	Uniform body force in y-direction.
5	Nonuniform body force in the y-direction.
6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
8	Uniform pressure on 2-6-3 face.
9	Nonuniform pressure on 2-6-3 face.
10	Uniform pressure on 3-7-4 face.
11	Nonuniform pressure on 3-7-4 face.
12	Uniform pressure on 4-8-1 face.
13	Nonuniform pressure on 4-8-1 face.
20	Uniform shear force on 1-5-2 face in the 1⇒5⇒2 direction.
21	Nonuniform shear force on side 1-5-2.
22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
23	Nonuniform shear force on side 2-6-3.
24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
25	Nonuniform shear force on side 3-7-4.
26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
27	Nonuniform shear force on side 4-8-1.

Load Type (IBODY)	Description
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Four strain components are printed in the order listed below:

1 = ε_{xx}, direct

2 = ε_{yy}, direct

3 = ε_{zz}, direct

4 = γ_{xy}, shear

Output of Stresses

Output for stresses is the same as for Output of Strains .

Transformation

Only in x-y plane.

Tying

Use the UFORMSN user subroutine.

Output Points

If the CENTROID parameter is used, the output occurs at the centroid of the element, given as point 5 in Figure 5-36.

If the ALL POINTS parameter is used, nine output points are given, as shown in Figure 5-36. This is the usual option for a second-order element.

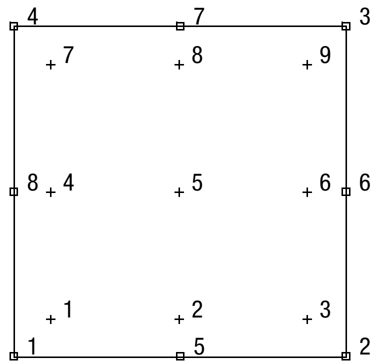


Figure 5-36 Integration Points of Eight-node, 2-D Element

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinates.

Note:	Distortion of element during analysis can cause bad solution. Element type 6 or 11 is preferred.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 41. See Element 41 for a description of the conventions used for entering the flux and film data for this element.

Element 28

Axisymmetric, Eight-node Distorted Quadrilateral

Element type 28 is an eight-node, isoparametric, arbitrary quadrilateral written for axisymmetric applications. This element uses biquadratic interpolation functions to represent the coordinates and displacements; hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses.

Lower-order elements, such as type 10, are preferred in contact analyses.

The stiffness of this element is formed using nine-point Gaussian integration.

This element can be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 33 instead. Element type 33 is also preferable for small strain incompressible elasticity.

Quick Reference

Type 28

Second-order, isoparametric, distorted quadrilateral. Axisymmetric formulation.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention in z-r plane). Then the fifth node between first and second; the sixth node between second and third, etc. See Figure 5-37.

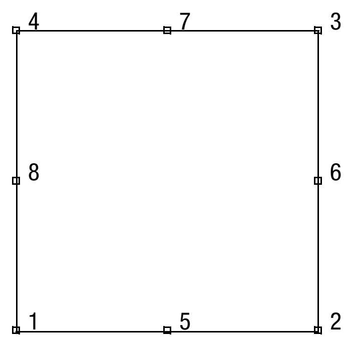


Figure 5-37 Nodes of Eight-node Axisymmetric Element

Geometry

No geometry input for this element.

Coordinates

Two global coordinates, z and r, at each node.

Degrees of Freedom

Two at each node:

1 = u = global z-direction displacement (axial)
2 = v = global r-direction displacement (radial)

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.
4	Uniform body force in y-direction.
5	Nonuniform body force in the y-direction.
6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
8	Uniform pressure on 2-6-3 face.
9	Nonuniform pressure on 2-6-3 face.
10	Uniform pressure on 3-7-4 face.
11	Nonuniform pressure on 3-7-4 face.
12	Uniform pressure on 4-8-1 face.
13	Nonuniform pressure on 4-8-1 face.
20	Uniform shear force on 1-5-2 face in the 1⇒5⇒2 direction.
21	Nonuniform shear force on side 1-5-2.
22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
23	Nonuniform shear force on side 2-6-3.
24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
25	Nonuniform shear force on side 3-7-4.
26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.

Load Type	Description
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

Concentrated nodal loads must be the value of the load integrated around the circumference.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Four strain components are printed in the order listed below:

1 = ε_{zz}, direct

2 = ε_{rr}, direct

3 = $\varepsilon_{\theta\theta}$, hoop direct

4 = γ_{zr}, shear in the section

Output of Stresses

Output for stresses is the same as for Output of Strains .

Transformation

Only in z-r plane.

Tying

Use the UFORMSN user subroutine.

Output Points

If the CENTROID parameter is used, the output occurs at the centroid of the element, given as point 5 in Figure 5-38.

If the ALL POINTS parameter is used, nine output points are given as shown in Figure 5-38. This is the usual option for a second-order element.

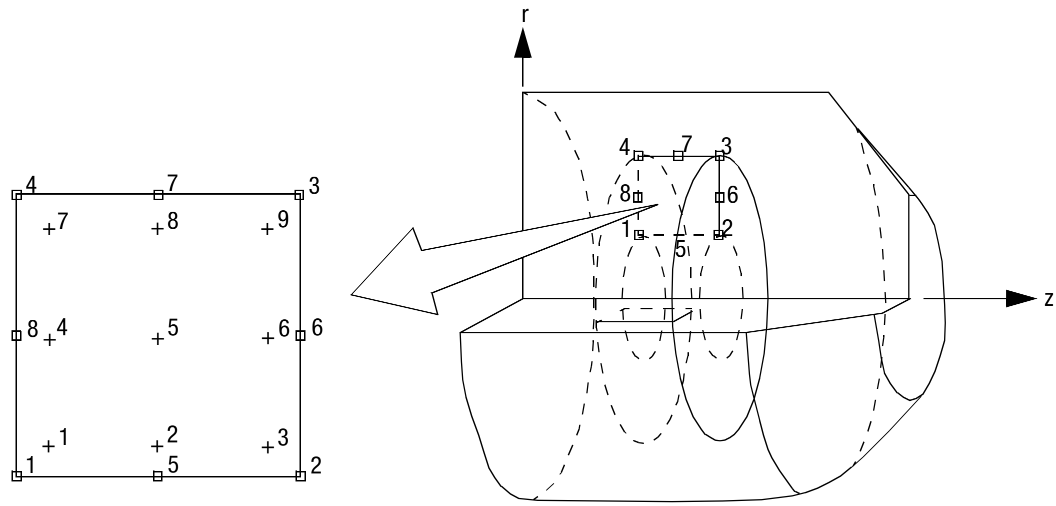


Figure 5-38 Integration Points of Eight-node, Axisymmetric Element

Updating Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinates.

Note:	Distortion of element during analysis can cause bad solution. Element type 2 or 10 is preferred.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 42. See Element 42 for a description of the conventions used for entering the flux and film data for this element.

Element 29

Generalized Plane Strain, Distorted Quadrilateral

This element is an extension of the plane strain isoparametric quadrilateral (element type 27) to the generalized plane strain case. The generalized plane strain condition is obtained by allowing two extra nodes in each element. One of these nodes has the single degree of freedom of change of distance between the top and bottom of the structure at one point (i.e., change in thickness at that point) while the other additional node contains the relative rotations θ_{xx} and θ_{yy} of the top surface with respect to the bottom surface about the same point. The generalized plane strain condition is attained by allowing these two nodes to be shared by all elements forming the structure. Note that this is an extension of the usual generalized plane strain condition which is obtained by setting the relative rotations to zero (by kinematic boundary conditions) and from there the element could become a plane strain element if the relative displacement is also made zero. Note that the sharing of the two extra nodes by all elements implies two full nodal rows in the generated plane strain part of the assembled stiffness matrix considerable computational savings are achieved if these two nodes are given the highest node numbers in that part of the structure.

This element uses biquadratic interpolation functions to represent the coordinates and displacements; hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses.

Lower-order elements, such as type 19, are preferred in contact analyses.

The stiffness of this element is formed using nine-point Gaussian integration.

This element can be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 34 instead. Element type 34 is also preferable for small strain incompressible elasticity.

This element cannot be used with the CASI iterative solver.

Quick Reference

Type 29

Second-order, isoparametric, distorted quadrilateral. Generalized plane strain formulation.

Connectivity

Corners numbered first in a counterclockwise direction (right-handed convention in x-y plane). Then the fifth node between first and second; the sixth node between second and third, etc. The ninth and tenth nodes are the generalized plane strain nodes shared with all elements in the mesh.

Geometry

Thickness stored in first data field (EGEOM1). Default thickness is unity. Other fields are not used.

Coordinates

Global x and global y coordinate at each of the ten nodes. The ninth and tenth nodes can be anywhere in the (x, y) plane.

Degrees of Freedom

Two at each of the first eight nodes:

1 = u = global x-direction displacement
2 = v = global y-direction displacement

One at the ninth node:

1 = Δz = relative z-direction displacement of front and back surfaces. See Figure 5-39.

Two at the tenth node:

1 = $\Delta\theta_x$ = relative rotation of front and back surfaces about global x- axis.
2 = $\Delta\theta_y$ = relative rotation of front and back surfaces about global y-axis. See Figure 5-39.

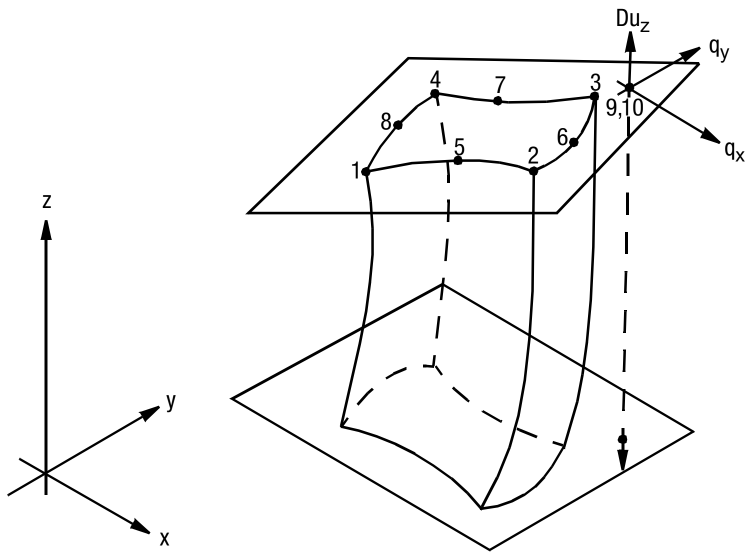


Figure 5-39 Generalized Plane Strain Distorted Quadrilateral

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type	Description
* 0	Uniform pressure on 1-5-2 face.
* 1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.
4	Uniform body force in y-direction.
5	Nonuniform body force in the y-direction.
* 6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
* 7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.

Load Type	Description
* 8	Uniform pressure on 2-6-3 face.
* 9	Nonuniform pressure on 2-6-3 face.
* 10	Uniform pressure on 3-7-4 face.
* 11	Nonuniform pressure on 3-7-4 face.
* 12	Uniform pressure on 4-8-1 face.
* 13	Nonuniform pressure on 4-8-1 face.
* 20	Uniform shear force on 1-2-5 face in the 1⇒2⇒5 direction.
* 21	Nonuniform shear force on side 1-5-2.
* 22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
* 23	Nonuniform shear force on side 2-6-3.
* 24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
* 25	Nonuniform shear force on side 3-7-4.
* 26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
* 27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

All pressures are positive when directed into the element. Load types shown with an asterisk (*) require the magnitude of the load to be entered as force per unit area. To prescribe these loads in force per unit length, add 50 to the load type. This is often useful in design optimization where the thickness changes, but it is desired that the applied force remain the same. For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Strains are printed in the order listed below:

1 = ε_{xx}, direct
2 = ε_{yy}, direct
3 = ε_{zz}, thickness direction direct
4 = γ_{xy}, shear in the (x-y) plane

No γ_{xz}, γ_{yz}, or shear – relative rotations of front and back surfaces.

Output of Stresses

Output of stresses is the same as Output of Strains .

Transformation

Only in x-y plane.

Tying

Use the UFORMSN user subroutine.

Output Points

If the CENTROID parameter is used, the output occurs at the centroid of the element, given as point 5 in Figure 5-40.

If the ALL POINTS parameter is used, nine output points are given as shown in Figure 5-40. This is the usual option for a second order element.

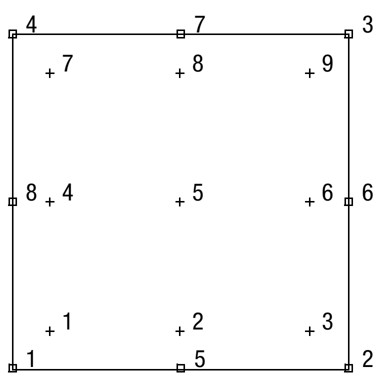


Figure 5-40 Integration Points of Eight-node, 2-D Element

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinates. Thickness is updated.

Note:	Distortion of element during analysis can cause bad solution. Element type 19 is preferred.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 41. See Element 41 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered a design variable for this element type.

Element 30

Membrane, Eight-node Distorted Quadrilateral

Element type 30 is an eight-node, isoparametric, arbitrary quadrilateral written for membrane applications. As a membrane has no bending stiffness, the element is very unstable.

This element uses biquadratic interpolation functions to represent the coordinates and displacements. This allows for a more accurate representation of the strain fields in elastic analyses than lower order elements.

Lower-order elements, such as type 18, are preferred in contact analyses.

The stiffness of this element is formed using nine-point Gaussian integration.

All constitutive models can be used with this element.

This element is usually used with the LARGE DISP parameter, in which case the (tensile) initial stress stiffness increases the rigidity of the element.

Geometric Basis

Similar to element type 22, the element is defined geometrically by the (x, y, z) coordinates of the four corner nodes and four midside nodes. The element thickness is specified in the GEOMETRY option. Local orthogonal surface directions ($\underset{\sim}{V}_1$, $\underset{\sim}{V}_2$, and $\underset{\sim}{V}_3$) for each integration point are defined below (see Figure 5-41):

> At each of the integration points, the vectors tangent to the curves with constant isoparametric coordinates are normalized.
>
> $$t_1 = \frac{\partial x}{\partial \xi} \Big/ \left|\frac{\partial x}{\partial \xi}\right|, \quad t_2 = \frac{\partial x}{\partial \eta} \Big/ \left|\frac{\partial x}{\partial \eta}\right|$$
>
> Now a new basis is being defined as:
>
> $$s = t_1 + t_2, \quad d = t_1 - t_2$$
>
> After normalizing these vectors by:
>
> $$\bar{s} = s/(\sqrt{2}|s|) \qquad \bar{d} = d/(\sqrt{2}|d|) \;,$$
>
> the local orthogonal directions are then obtained as:
>
> $$\underset{\sim}{V}_1 = \bar{s} + \bar{d}, \underset{\sim}{V}_2 = \bar{s} - \bar{d}, \text{ and } \underset{\sim}{V}_3 = \underset{\sim}{V}_1 \times \underset{\sim}{V}_2$$
>
> In this way, the vectors $\frac{\partial x}{\partial \xi}$, $\frac{\partial x}{\partial \eta}$ and $\underset{\sim}{V}_1$, $\underset{\sim}{V}_2$ have the same bisecting plane.

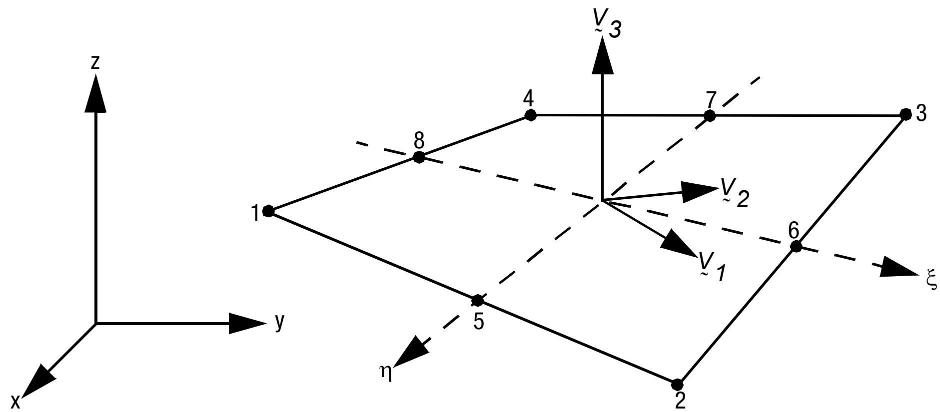


Figure 5-41 Local Coordinate System for Eight-node Membrane Element

The local directions for the Gaussian integrations points are found by projection of the centroid directions. Hence, if the element is flat, the directions at the Gauss points are identical to those at the centroid.

In addition, this element can be used for an electrostatic problem. A description of this option can be found in Marc Volume A: Theory and User Information.

Quick Reference

Type 30

Eight-node, second-order, isoparametric membrane element (Figure 5-42). Plane stress.

Connectivity

The corners are numbered first in a counterclockwise direction. The fifth node is located between nodes 1 and 2; the sixth node is located between nodes 2 and 3, etc.

Geometry

The thickness is input in the first data field (`EGEOM1`). Other fields are not used.

Coordinates

Three global coordinates (x, y, z) at each node.

Degrees of Freedom

Three at each node – (u, v, w) in the global rectangular Cartesian system.

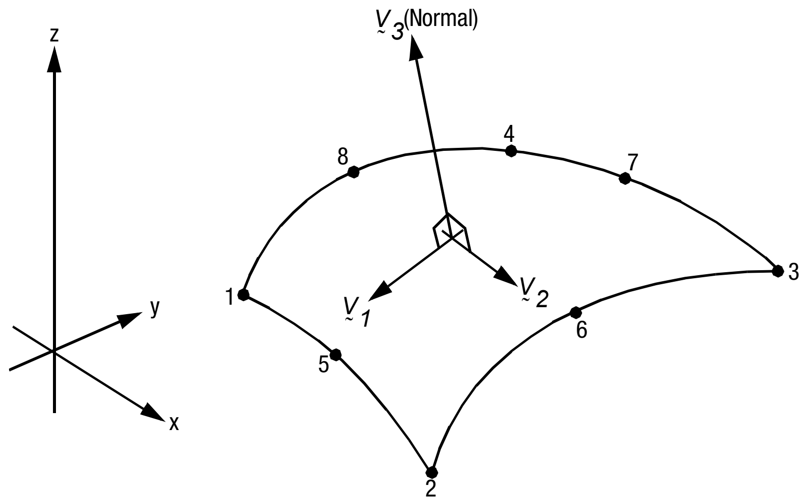


Figure 5-42 Eight-node, Second Order Membrane Element

Tractions

Pressure	Pressure is specified as load type 2 and is positive in the direction of $\underset{\sim}{V}_3$, the surface normal.
Self-weight	Self-weight is a force in the negative global z-direction proportional to surface areas. It is chosen as load type 1.
Nonuniform pressure	Specified as load type 4, positive in the direction of $\underset{\sim}{V}_3$, the surface normal (use the FORCEM user subroutine).
Nonuniform self-weight	Specified as load type 3, force in the negative global z-direction, proportional to surface area (use the FORCEM user subroutine).

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Stresses

Output of stress $(\sigma_{11}, \sigma_{22}, \tau_{12})$ and strain $(\varepsilon_{11}, \varepsilon_{22}, \gamma_{12})$ is in the local $(\underset{\sim}{V}_1, \underset{\sim}{V}_2)$ directions defined above.

Transformation

The global degrees of freedom can be transformed at any node.

Tying

Use the UFORMSN user subroutine.

Output Points

If the CENTROID parameter is used, the output occurs at point 5 in Figure 5-43. If the ALL POINTS parameter is included, the output is given at all nine points shown in Figure 5-43. The latter is the usual option.

Notes:	Sensitive to excessive distortions. Use a rectangular mesh.
	Membrane analysis is extremely difficult due to rigid body modes. For example, a circular cylinder shape is particularly numerically sensitive.
	This element has no bending stiffness.

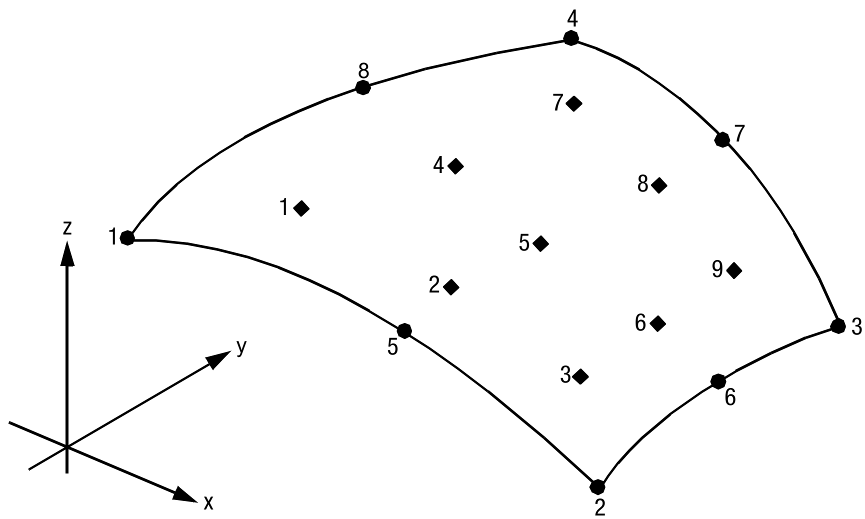


Figure 5-43 Integration Point Numbers for Eight-node Membrane Element

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 41. See Element 41 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered the design variable for this element.

Element 31

Elastic Curved Pipe (Elbow)/Straight Beam

This is a linear, elastic, curved-pipe beam element with flexibility of the elbow based on an analytical elastic solution of the elbow segment. The flexibility and resulting stress distribution is not only a function of the cross-sectional properties, but also of the internal pressure in the curved elbow.

In addition, arbitrary cross sections can be specified for this element thus allowing for the possibility of analyzing arbitrary curved beams. The element can be degenerated into a straight elastic beam with elastic properties. The effects of axial stiffness, bending stiffness and shear stiffness are included in this straight beam element.

A curved beam is specified by the coordinates of the end points of the beam segment (COORDINATES option), the bending radius and center of rotation (GEOMETRY option) (Figure 5-44). If a zero is specified for the bending radius, the element is assumed to be straight. Cross-sectional properties are specified via the GEOMETRY option or the BEAM SECT parameter for arbitrary cross-sectional properties.

Element 31 has three displacement and three rotation components of degrees of freedom with respect to the global system. On the output, however, the local quantities such as forces and moments in the plane and normal to the plane for each element are also given.

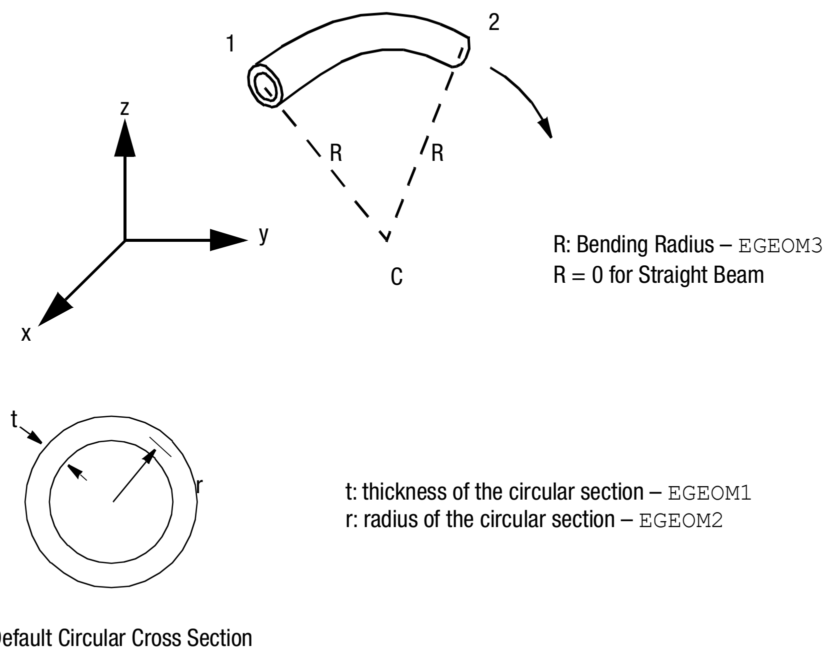


Figure 5-44 Elastic Curved (Elbow)/Straight Beam Element

Quick Reference

Type 31

Curved-pipe elbow or straight beam; elastic behavior.

Connectivity

Two nodes per element.

Geometry

The default cross section is a hollow, circular cylinder. Other cross sections are specified on the BEAM SECT parameter and are cross-referenced here.

EGEOM1

Thickness of the hollow cylinder if default cross section is used.
Enter a zero if a noncircular pipe is defined through the BEAM SECT parameter.

EGEOM2

Radius of the circular cross section if default cross section is used. Otherwise, the BEAM SECT identifier is specified on the parameter.

EGEOM3

Bending radius of the elbow if a curved beam has to be used.
Enter a zero if the element is straight.

For a curved beam, the coordinates of the center of curvature (C) are given in `EGEOM4`, `EGEOM5`, and `EGEOM6`. Nodes 1 and 2 and the center of curvature define the in-plane bending plane of the curved beam. The local x-direction is normal to this plane and the local y-direction of every cross section points to the center curvature. For a straight beam, the values `EGEOM4`, `EGEOM5`, and `EGEOM6` are used to define the components of a vector in the plane of the local y-axis and the beam axis (constructed by connecting node 1 and 2). The local y-axis lies on the same side as the specified vector. The vector is pointing to an imaginary center of curvature which, for a straight beam, lies at infinity.

Note:	This definition of the local directions for the straight beam is different from the definition used for other straight beam elements (the principal directions are swapped). It is consistent with the definition of the curved beam in its limiting case having its center of curvature at infinity.

Coordinates

It is sufficient to specify only three coordinates per node. They represent the spatial global x, y, z position of a node.

Degrees of Freedom

1 = u global displacement in x direction
2 = v global displacement in y direction
3 = w global displacement in z direction
4 = θ_x global rotation about the x-axis
5 = θ_y global rotation about the y-axis
6 = θ_z global rotation about the z-axis

Distributed Loads

Distributed loads can be entered as follows:

Load Type	Description
3	Internal pressure with closed-end caps (stress stiffening and axial loading).
-3	Internal pressure with open-end caps (stress stiffening effect only).

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Point Loads

Point loads and moments can be applied at the nodes.

Output

For each element, the following output is obtained in a local reference system:

1	In-plane shear force
2	Axial force
3	Out-of-plane shear force
4	Out-of-plane moment
5	Torque around beam axis
6	In-plane moment

In addition, the following (maximum) stress quantities are printed:

1	Shear stress due to in-plane shear force
2	Axial stress
3	Shear stress due to out-of-plane force
4	Maximum out-of-plane bending stress
5	Shear stress due to torsion

6	Maximum in-plane bending stress
7	Hoop stress due to internal pressure

Output Points

Output of stress quantities is at the nodal points. All quantities are based on analytical solutions. No numerical integration is required.

Transformation

Displacement and rotations at the node can be transformed to a local coordinate system.

Updated Lagrange Procedure and Finite Strain

This element does not have a large displacement or finite strain capability.

Coupled Analysis

This element cannot be used for a coupled analysis; it has no heat transfer equivalent.

Dynamic Analysis

This element can be used in a dynamic analysis. The mass matrix is based on a subdivision of the total mass onto the displacement degrees of freedom of the two end nodes.

Element 32

Plane Strain Eight-node Distorted Quadrilateral, Herrmann Formulation

Element type 32 is an eight-node, isoparametric, arbitrary quadrilateral written for incompressible plane strain applications. This element uses biquadratic interpolation functions to represent the coordinates and displacements. This allows for an accurate representation of the strain fields in elastic analyses. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is represented using bilinear interpolation functions based upon the extra degree of freedom at the corner nodes.

Lower-order elements, such as type 80, are preferred in contact analyses.

The stiffness of this element is formed using nine-point Gaussian integration.

This element is designed to be used for incompressible elasticity only. It can be used for either small strain behavior or large strain behavior using the Mooney or Ogden models.This element can be used in conjunction with other elements such as type 27 when other material behavior, such as plasticity, must be represented.

This element can also be used in coupled soil-pore pressure analyses.

Quick Reference

Type 32

Second-order, isoparametric, distorted quadrilateral. Plane strain. Hybrid formulation for incompressible and nearly incompressible elastic materials.

Connectivity

Corners numbered first in a counterclockwise direction (right-handed convention). Then the fifth node between first and second; the sixth node between second and third, etc. See Figure 5-45.

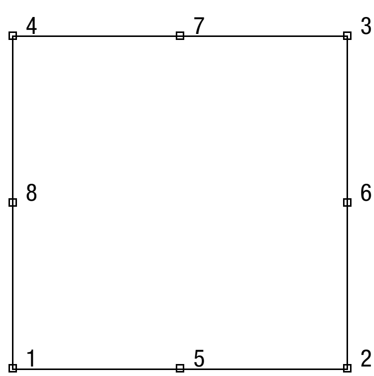


Figure 5-45 Eight-node, Plane Strain Herrmann Element

Geometry

Thickness stored in first data field (EGEOM1). Default thickness is unity. Other data fields are not used.

Coordinates

Two global coordinates, x and y, at each node.

Degrees of Freedom

1 = u = global x-direction displacement
2 = v = global y-direction displacement at corner nodes only

Additional degree of freedom at corner nodes only:

3	$= \sigma_{kk}/E$	= mean pressure variable (isotropic)
		= formula below (orthotropic)
		$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1+\frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2+\frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3\right]\frac{9}{E_1+E_2+E_3}$$
	= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)
	= p / K	(for Herrmann elements using additive decomposition; K here is the effective bulk modulus)

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.
4	Uniform body force in y-direction.
5	Nonuniform body force in the y-direction.
6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
8	Uniform pressure on 2-6-3 face.
9	Nonuniform pressure on 2-6-3 face.
10	Uniform pressure on 3-7-4 face.
11	Nonuniform pressure on 3-7-4 face.
12	Uniform pressure on 4-8-1 face.
13	Nonuniform pressure on 4-8-1 face.
20	Uniform shear force on 1-2-5 face in the 1⇒2⇒5 direction.

Load Type (IBODY)	Description
21	Nonuniform shear force on side 1-5-2.
22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
23	Nonuniform shear force on side 2-6-3.
24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
25	Nonuniform shear force on side 3-7-4.
26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

In coupled soil analyses, add 70 to the IBODY to apply distributed mass flux; a nonuniform mass flux is supplied via the FLUX user subroutine.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Strains are printed in the order listed below:

1	= ε_{xx}, direct
2	= ε_{yy}, direct
3	= ε_{zz}, thickness direction, direct
4	= γ_{xy}, shear

5	$= \sigma_{kk}/E$	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1 + \frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2 + \frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3\right]\frac{9}{E_1+E_2+E_3}$$	
	= -p	= negative pressure (for Mooney, Ogden, or Soil)

The last component is not printed in the output file, but is accessible via the ELEVAR and PLOTV user subroutines.

Output of Stresses

Four stresses corresponding to the first four strains.

Transformation

Only in x-y plane.

Tying

Use the UFORMSN user subroutine.

Output Points

If the CENTROID parameter is used, the output occurs at the centroid of the element, given as point 5 in Figure 5-46.

If the ALL POINTS parameter is used, nine output points are given, as shown in Figure 5-46. This is the usual option for a second-order element.

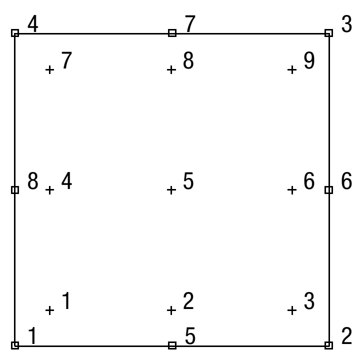


Figure 5-46 Integration Points for Eight-node Element

Special Considerations

The mean pressure degree of freedom must be allowed to be discontinuous across changes in material properties. Use tying for this case.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available. Finite elastic strains can be calculated with the MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 41. See Element 41 for a description of the conventions used for entering the flux and film data for this element.

Element 33 Axisymmetric, Eight-node Distorted Quadrilateral, Herrmann Formulation

Element type 33 is an eight-node, isoparametric, arbitrary quadrilateral written for incompressible axisymmetric applications. This element uses biquadratic interpolation functions to represent the coordinates and displacements. This allows for an accurate representation of the strain fields in elastic analyses. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is represented using bilinear interpolation functions based upon the extra degree of freedom at the corner nodes.

Lower-order elements, such as type 81, are preferred in contact analyses.

The stiffness of this element is formed using nine-point Gaussian integration.

This element is designed to be used for incompressible elasticity only. It can be used for either small strain behavior or large strain behavior using the Mooney or Ogden models.This element can be used in conjunction with other elements such as type 28 when other material behavior, such as plasticity, must be represented.

This element can also be used in coupled soil-pore pressure analyses.

Quick Reference

Type 33

Second-order, isoparametric, distorted quadrilateral. Axisymmetric formulation. Hybrid formulation for incompressible or nearly incompressible materials.

Connectivity

Corners numbered first in a counterclockwise direction (right-handed convention in z-r plane). Then the fifth node between the first and second; the sixth node between the second and third, etc. See Figure 5-47.

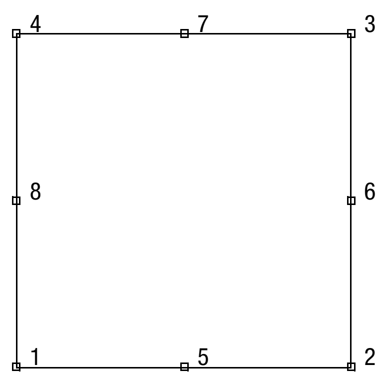


Figure 5-47 Eight-node Axisymmetric Herrmann Element

Geometry

No geometry input for this element.

Coordinates

Two global coordinates, z and r, at each node.

Degrees of Freedom

1 = u = global z-direction displacement (axial)
2 = v = global r-direction displacement (radial)

Additional degree of freedom at corner nodes only:

3 = σ_{kk}/E	= mean pressure variable (isotropic)
= formula below (orthotropic)	
$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1+\frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2+\frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3\right]\frac{9}{E_1+E_2+E_3}$$	
= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)
= p / K	(for Herrmann elements using additive decomposition; K here is the effective bulk modulus)

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.
4	Uniform body force in y-direction.
5	Nonuniform body force in the y-direction.
6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
8	Uniform pressure on 2-6-3 face.
9	Nonuniform pressure on 2-6-3 face.
10	Uniform pressure on 3-7-4 face.
11	Nonuniform pressure on 3-7-4 face.
12	Uniform pressure on 4-8-1 face.
13	Nonuniform pressure on 4-8-1 face.
20	Uniform shear force on 1-2-5 face in the 1⇒2⇒5 direction.

Load Type (IBODY)	Description
21	Nonuniform shear force on side 1-5-2.
22	Uniform shear force on side 2-6-3 in the 2⇒6⇒ 3 direction.
23	Nonuniform shear force on side 2-6-3.
24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
25	Nonuniform shear force on side 3-7-4.
26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

In coupled soil analyses, add 70 to the IBODY to apply distributed mass flux; a nonuniform mass flux is supplied via the FLUX user subroutine.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

The strains are printed in the order defined below:

1	= ε_{zz}, direct	
2	= ε_{rr}, direct	
3	= $\varepsilon_{\theta\theta}$, hoop direct	
4	= γ_{zr}, shear in the section	
5	= σ_{kk}/E	= mean pressure variable (isotropic)

	= formula below (orthotropic)	
	$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1+\frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2+\frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3\right]\frac{9}{E_1+E_2+E_3}$$	
	= -p	= negative pressure (for Mooney, Ogden, or Soil)

The last component is not printed in the output file, but is accessible via the ELEVAR and PLOTV user subroutines.

Output of Stresses

Output for stresses is the same as the first four Output of Strains .

Transformation

Only in z-r plane.

Tying

Use the UFORMSN user subroutine.

Output Points

If the CENTROID parameter is used, the output occurs at the centroid of the element, given as point 5 in Figure 5-48.

If the ALL POINTS parameter is used, nine output points are given as shown in Figure 5-48. This is the usual option for a second-order element.

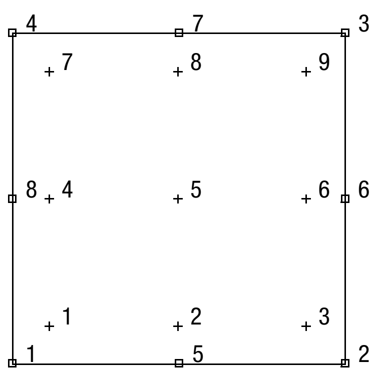


Figure 5-48 Integration Points for Eight-node Element

Special Considerations

The mean pressure degree of freedom must be allowed to be discontinuous across changes in material properties. Use tying for this case.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available. Finite elastic strains can be calculated with the MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 38. See Element 38 for a description of the conventions used for entering the flux and film data for this element.

Element 34 Generalized Plane Strain Distorted Quadrilateral, Herrmann Formulation

This element is an extension of the plane strain isoparametric quadrilateral (element type 32) to the generalized plane strain case. The generalized plane strain condition is obtained by allowing two extra nodes in each element. One of these nodes has the single degree of freedom of change of distance between the top and bottom of the structure at one point (that is, change in thickness at that point) while the other additional node contains the relative rotations θ_{xx} and θ_{yy} of the top surface with respect to the bottom surface about the same point. The generalized plane strain condition is attained by allowing these two nodes to be shared by all elements forming the structure. Note that this is an extension of the usual generalized plane strain condition which is obtained by setting the relative rotations to zero (by kinematic boundary conditions) and from there the element could become a plane strain element if the relative displacement is also made zero. Note that the sharing of the two extra nodes by all elements implies two full nodal rows in the generated plane strain part of the assembled stiffness matrix considerable computational savings are achieved if these two nodes are given the highest node numbers in that part of the structure.These elements cannot be used with the element-by-element iterative solver.

This element uses biquadratic interpolation functions to represent the coordinates and displacements. This allows for an accurate representation of the strain fields in elastic analyses. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is represented using bilinear interpolation functions based upon the extra degree of freedom at the corner nodes.

Lower-order elements, such as type 81, are preferred in contact analyses.

The stiffness of this element is formed using nine-point Gaussian integration.

This element is designed to be used for incompressible elasticity only. It can be used for either small strain behavior or large strain behavior using the Mooney or Ogden models.This element can be used in conjunction with other elements such as type 29 when other material behavior, such as plasticity, must be represented.

This element cannot be used with the CASI iterative solver.

Quick Reference

Type 34

Second order, isoparametric, distorted quadrilateral, generalized plane strain, Hybrid formulation.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention in x-y plane). Then the fifth node between the first and second; the sixth node between the second and third, etc. The ninth and tenth nodes are the generalized plane strain nodes shared with all elements in the mesh.

Geometry

Thickness stored in first data field (`EGEOM1`). Default thickness is unity. Other fields are not used.

Coordinates

Two global coordinates, x and y, at each of the ten nodes. Note that the ninth and tenth nodes can be anywhere in the (x,y) plane.

Degrees of Freedom

At each of the first eight nodes:

1 = u = global x-direction displacement
2 = v = global y-direction displacement

Additional degree of freedom at the first four (corner) nodes:

3	= σ_{kk}/E	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1+\frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2+\frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3\right]\frac{9}{E_1+E_2+E_3}$	
	= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)
	= p / K	(for Herrmann elements using additive decomposition; K here is the effective bulk modulus)

One at the ninth node:

1 = Δ_z = relative z-direction displacement of front and back surfaces. See Figure 5-49.

Two at the tenth node:

1 = $\Delta\theta_x$ = relative rotation of front and back surfaces about global x-axis.
2 = $\Delta\theta_y$ = relative rotation of front and back surfaces about global y-axis. See Figure 5-49.

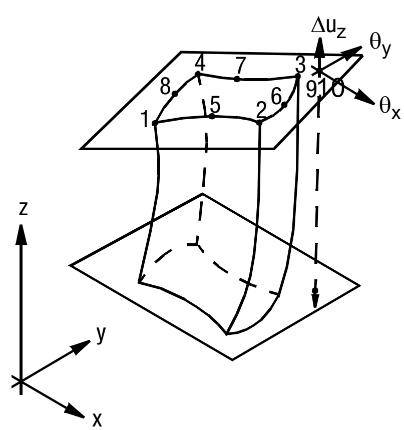


Figure 5-49 Generalized Plane Strain Distorted Quadrilateral

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type (IBODY)	Description
* 0	Uniform pressure on 1-5-2 face.
* 1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.
4	Uniform body force in y-direction.
5	Nonuniform body force in the y-direction.
* 6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
* 7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
* 8	Uniform pressure on 2-6-3 face.
* 9	Nonuniform pressure on 2-6-3 face.
* 10	Uniform pressure on 3-7-4 face.
* 11	Nonuniform pressure on 3-7-4 face.
* 12	Uniform pressure on 4-8-1 face.
* 13	Nonuniform pressure on 4-8-1 face.
* 20	Uniform shear force on 1-2-5 face in the 1⇒2⇒5 direction.
* 21	Nonuniform shear force on side 1-5-2.
* 22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
* 23	Nonuniform shear force on side 2-6-3.
* 24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
* 25	Nonuniform shear force on side 3-7-4.
* 26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
* 27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Load Type (IBODY)	Description
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Load types shown with an asterisk (*) require the magnitude of the load to be entered as force per unit area. To prescribe these loads in force per unit length, add 50 to the load type. This is often useful in design optimization where the thickness changes, but it is desired that the applied force remain the same.For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Strains are printed in the order defined below:

1 = ε_{xx}, direct

2 = ε_{yy}, direct

3 = ε_{zz}, thickness direction, direct

4 = γ_{xy}, shear in the (x-y) plane

No γ_{xz} or γ_{yz} shear

5	= σ_{kk}/E	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1+\frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2+\frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3\right]\frac{9}{E_1+E_2+E_3}$$	
	= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)

The last component is not printed in the output file, but is accessible via the ELEVAR and PLOTV user subroutines.

Output of Stresses

Output for stresses is the same as the first four strains.

Transformation

Only in x-y plane.

Tying

Use the UFORMSN user subroutine.

Output Points

If the CENTROID parameter is used, the output occurs at the centroid of the element, given as point 5 in Figure 5-50.

If the ALL POINTS parameter is used, nine output points are given as shown in Figure 5-50. This is the usual parameter for a second-order element.

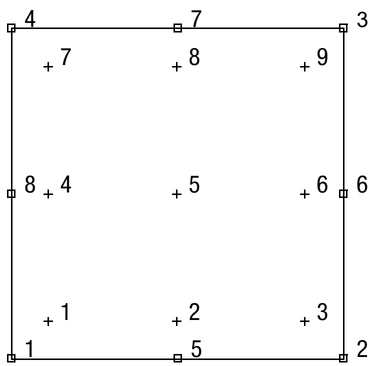


Figure 5-50 Integration Points for Eight-node Element

Special Considerations

The mean pressure degree of freedom must be allowed to be discontinuous across changes in material properties. Use tying for this case.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available. Finite elastic strains can be calculated with the MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 41. See Element 41 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered a design variable for this element type.

Element 35 Three-dimensional 20-node Brick, Herrmann Formulation

Element type 35 is a 20-node, isoparametric, arbitrary hexahedral written for incompressible applications. This element uses triquadratic interpolation functions to represent the coordinates and displacements. This allows for an accurate representation of the strain fields in elastic analyses. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is represented using trilinear interpolation functions based upon the extra degree of freedom at the corner nodes.

Lower-order elements, such as type 84, are preferred in contact analyses.

The stiffness of this element is formed using 27-point Gaussian integration.

This element is designed to be used for incompressible elasticity only. It can be used for either small strain behavior or large strain behavior using the Mooney or Ogden models.This element can be used in conjunction with other elements such as type 21 when other material behavior, such as plasticity, must be represented.

This element can also be used in coupled soil-pore pressure analyses.

Geometry

The geometry of the element is interpolated from the Cartesian coordinates of 20 nodes.

Connectivity

The convention for the ordering of the connectivity array is as follows:

> Nodes 1, 2, 3, 4 are the corners of one face, given in a counterclockwise direction when viewed from inside the element. Nodes 5, 6, 7, 8 are the corners of the opposite face; node 5 shares an edge with 1, node 6 with 2, etc. Nodes 9, 10, 11, 12 are middle of the edges of the 1, 2, 3, 4 face; node 9 between 1 and 2; node 10 between 2 and 3, etc. Similarly, nodes 13, 14, 15, 16 are midpoints on the 5, 6, 7, 8 face; node 13 between 5 and 6, etc. Finally, node 17 is the midpoint of the 1-5 edge; node 18 of the 2-6 edge, etc.

Note that in most normal cases, the elements are generated automatically, so that you need not concern yourself with the node numbering scheme.

Reduction to Wedge or Tetrahedron

The element can be reduced as far as a tetrahedron, simply by repeating node numbers. Element type 130 would be preferred for tetrahedrals.

Integration

The element is integrated numerically using 27 points (Gaussian quadrature). The first plane of such points is closest to the 1, 2, 3, 4 face of the element, with the first point closest to the first node of the element (see Figure 5-51). Two similar planes follow, moving toward the 5, 6, 7, 8 face. Thus, point 14 represents the “centroid” of the element, and is used for stress output, if only one point is flagged.

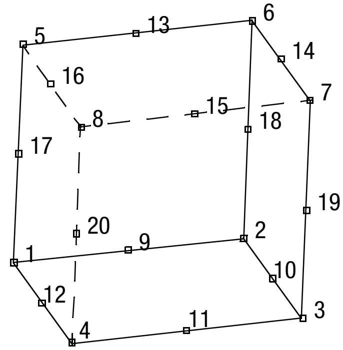


Figure 5-51 Twenty-node – Brick Element

The FORCEM user subroutine is called once per integration point when flagged. The magnitude of load defined by DIST LOADS is ignored and the FORCEM value is used instead.

For nonuniform body force, force values must be provided for the 27 integration points.

For nonuniform surface pressures, values need only be supplied for the nine integration points on the face of application.

Quick Reference

Type 35

Twenty-nodes, isoparametric arbitrary distorted cube. Herrmann formulation. See Marc Volume A: Theory and User Information for details on this theory.

Connectivity

Twenty nodes numbered as described in the connectivity write-up for this element and is shown in Figure 5-51.

Geometry

Not required.

Coordinates

Three global coordinates in the x, y, and z directions.

Degrees of Freedom

Three global degrees of freedom, u, v, and w, at all nodes. Additional degree of freedom at the corner nodes (first 8 nodes) is:

σ_{kk}/E	= mean pressure variable (isotropic)

= formula below (orthotropic)

$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1 + \frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2 + \frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3 \right] \frac{9}{E_1 + E_2 + E_3}$$

-p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)
= p / K	(for Herrmann elements using additive decomposition; K here is the effective bulk modulus)

Distributed Loads

Distributed loads chosen by value of IBODY are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3-4 face.
1	Nonuniform pressure on 1-2-3-4 face; magnitude supplied through the FORCEM user subroutine.
2	Uniform body force in z-direction.
3	Nonuniform body force per unit volume (e.g., centrifugal force); magnitude and direction supplied through the FORCEM user subroutine.
4	Uniform pressure on 6-5-8-7 face.
5	Nonuniform pressure on 6-5-8-7 face.
6	Uniform pressure on 2-1-5-6 face.
7	Nonuniform pressure on 2-1-5-6 face.
8	Uniform pressure on 3-2-6-7 face.
9	Nonuniform pressure on 3-2-6-7 face.
10	Uniform pressure on 4-3-7-8 face.
11	Nonuniform pressure on 4-3-7-8 face.
12	Uniform pressure on 1-4-8-5 face.
13	Nonuniform pressure on 1-4-8-5 face.
20	Uniform pressure on 1-2-3-4 face.
21	Nonuniform load on 1-2-3-4 face; magnitude and direction supplied in the FORCEM user subroutine.
22	Uniform body force in z-direction.
23	Nonuniform body force per unit volume (e.g., centrifugal force); magnitude supplied through the FORCEM user subroutine.
24	Uniform pressure on 6-5-8-7 face.
25	Nonuniform load on 6-5-8-7 face; magnitude and direction supplied in the FORCEM user subroutine.
26	Uniform pressure on 2-1-5-6 face.
27	Nonuniform load on 2156 face; magnitude and direction supplied in the FORCEM user subroutine.
28	Uniform pressure on 3-2-6-7 face.
29	Nonuniform load on 3267 face; magnitude and direction supplied in the FORCEM user subroutine.
30	Uniform pressure on 4-3-7-8 face.
31	Nonuniform load on 4378 face; magnitude and direction supplied in the FORCEM user subroutine.

Load Type	Description
32	Uniform pressure on 1-4-8-5 face.
33	Nonuniform load on 1-4-8-5 face; magnitude and direction supplied in the FORCEM user subroutine.
40	Uniform shear 1-2-3-4 face in 12 direction.
41	Nonuniform shear 1-2-3-4 face in 12 direction.
42	Uniform shear 1-2-3-4 face in 23 direction.
43	Nonuniform shear 1-2-3-4 face in 23 direction.
48	Uniform shear 6-5-8-7 face in 56 direction.
49	Nonuniform shear 6-5-8-7 face in 56 direction.
50	Uniform shear 6-5-8-7 face in 67 direction.
51	Nonuniform shear 6-5-8-7 face in 67 direction.
52	Uniform shear 2-1-5-6 face in 12 direction.
53	Nonuniform shear 2-1-5-6 face in 12 direction.
54	Uniform shear 2-1-5-6 face in 15 direction.
55	Nonuniform shear 2-1-5-6 face in 15 direction.
56	Uniform shear 3-2-6-7 face in 23 direction.
57	Nonuniform shear 3-2-6-7 face in 23 direction.
58	Uniform shear 3-2-6-7 face in 26 direction.
59	Nonuniform shear 2-3-6-7 face in 26 direction.
60	Uniform shear 4-3-7-8 face in 34 direction.
61	Nonuniform shear 4-3-7-8 face in 34 direction.
62	Uniform shear 4-3-7-8 face in 37 direction.
63	Nonuniform shear 4-3-7-8 face in 37 direction.
64	Uniform shear 1-4-8-5 face in 41 direction.
65	Nonuniform shear 1-4-8-5 face in 41 direction.
66	Uniform shear 1-4-8-5 face in 15 direction.
67	Nonuniform shear 1-4-8-5 face in 15 direction.

Load Type	Description
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

In coupled soil analyses, add 70 to the IBODY to apply distributed mass flux; a nonuniform mass flux is supplied via the FLUX user subroutine.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Strain output in global components:

1	$= \varepsilon_{xx}$	
2	$= \varepsilon_{yy}$	
3	$= \varepsilon_{zz}$	
4	$= \varepsilon_{xy}$	
5	$= \varepsilon_{yz}$	
6	$= \varepsilon_{zx}$	
7	$= \sigma_{kk}/E$	= mean pressure variable (isotropic)
	= formula below (orthotropic) $H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1 + \frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\Delta_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2 + \frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3 \right] \frac{9}{E_1 + E_2 + E_3}$	
	$= -p$	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)

The last component is not printed in the output file, but is accessible via the ELEVAR and PLOTV user subroutines.

Output of Stresses

Output for stresses is the same as the first six strains.

Transformation

Three global degrees of freedom can be transformed to local degrees of freedom.

Tying

Use the UFORMSN user subroutine.

Output Points

Centroid or twenty-seven Gaussian integration points (see Figure 5-52).

Note:	Large bandwidth results in long run times; optimize.

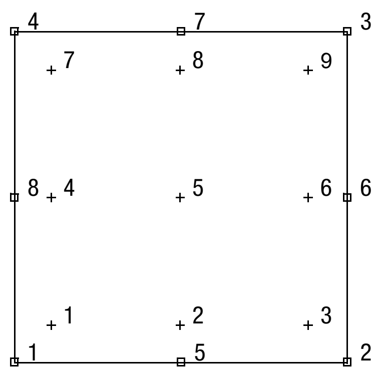


Figure 5-52 Integration Points on First Plane of Brick Element

Special Considerations

The mean pressure degree of freedom must be allowed to be discontinuous across changes in material properties. Use tying for this case.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available. Finite elastic strains can be calculated with the MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 44. See Element 44 for a description of the conventions used for entering the flux and film data for this element.

Element 36

Three-dimensional Link (Heat Transfer Element)

This element is a simple, linear, straight link with constant cross-sectional area. It is the heat transfer equivalent of element type 9.

This element can be used as a convection-radiation link for the simulation of convective and/or radiative boundary conditions (known ambient temperatures) or, for the situation of cross convection and/or cross radiation. In order to use this element as a convection-radiation link, additional input data must be entered using the GEOMETRY option.

The conductivity and specific heat capacity of this element are formed using one-point and two-point Gaussian integration, respectively.

Quick Reference

Type 36

Three-dimensional, two-node, heat transfer link.

Connectivity

Two nodes per element (see Figure 5-53).

Figure 5-53 Three-dimensional Heat Link

Geometry

The cross-sectional area is input in the first data field (EGEOM1); the other fields are not used. If not specified, the cross-sectional area defaults to 1.0.

If the element is used as a convection-radiation link, the following data must be entered:

EGEOM2 = emissivity

EGEOM5 = constant film coefficient

The Stefan-Boltzmann constant and the conversion to absolute temperatures is entered through the PARAMETERS model definition option.

If the element needs to be offset from the user-specified position, the eighth data field (EGEOM8) is set to -1.0. In this case, an extra line containing the offset information is provided (see the GEOMETRY option in Marc Volume C: Program Input). The offset vector at the first corner node is provided in the first three fields and the offset vector at the second corner node is provided in the next three fields of the extra line. The coordinate system used for the offset vectors at the two nodes are indicated by the corresponding flags in the eighth and ninth fields of the extra line. A flag setting of 0 indicates global coordinate system, 2 indicates local shell normal, and 3 indicates local nodal coordinate system. The flag setting of 2 can be used when the beam nodes are also attached to a shell. In this case, the offset magnitude at the first and second nodes are specified in the first and fourth data fields of the extra line, respectively. The automatically calculated shell normals at the nodes are used to construct the offset vectors.

Coordinates

Three coordinates per node in the global x, y, and z direction.

Degrees of Freedom

One degree of freedom per node:

1 = temperature

Fluxes

Distributed fluxes according to value of IBODY are as follows:

Flux Type	Description
0	Uniform flux on first node (per cross-sectional area).*
1	Volumetric flux on entire element (per volume).
2	Nonuniform flux given in the FLUX user subroutine on first node (per cross-sectional area).*
3	Nonuniform volumetric flux given in the FLUX user subroutine on entire element (per volume).
*Flux types 0 and 2 are not supported using table input.	

Films

Same specification as Fluxes .

Tying

Use the UFORMSN user subroutine.

Joule Heating

Capability is available.

Current

Same specification as Fluxes .

Output Points

A single value at the centroid is given.

Element 37

Arbitrary Planar Triangle (Heat Transfer Element)

Element type 37 is a three-node, isoparametric, triangular element written for planar heat transfer applications. This element can also be used for electrostatic and magnetostatic analysis. As this element uses bilinear interpolation functions, the thermal gradients are constant throughout the element.

In general, one needs more of these lower-order elements than the higher-order elements such as type 131. Hence, use a fine mesh.

The conductivity of this element is formed using one-point integration at the centroid. The specific heat capacity matrix of this element is formed using four-point Gaussian integration.

Quick Reference

Type 37

Two-dimensional, arbitrary, three-node, heat transfer triangle.

Connectivity

Node numbering must follow right-handed convention (see Figure 5-54).

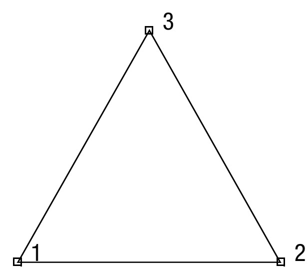


Figure 5-54 Planar Heat Transfer Triangle

Geometry

The thickness of the element can be specified in the first data field (EGEOM1); the other fields are not used. If not specified, unit thickness is assumed.

Coordinates

Two coordinates in the global x and y directions.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)
1 = potential (magnetostatic)

Fluxes

Flux types for distributed fluxes are as follows:

Flux Type	Description
0	Flux on 1-2 face of element (per unit area).
1	Volumetric flux on entire element (per unit volume).
3	Nonuniform flux given in the FLUX user subroutine on 1-2 face (per unit area).
4	Nonuniform volumetric flux given in the FLUX user subroutine on entire element (per unit volume).
6	Uniform flux on 2-3.
7	Nonuniform flux on 2-3.
8	Uniform flux on 3-1.
9	Nonuniform flux on 3-1.

For all nonuniform fluxes, the magnitude is given in the FLUX user subroutine.

All fluxes are positive when adding heat to the element. In addition, point fluxes can be applied at the nodes. Edge fluxes are evaluated using a two-point integration scheme where the integration points have the same location as the nodal points.

Films

Same specification as Fluxes .

Tying

Use the UFORMSN user subroutine.

Joule Heating

Capability is available.

Magnetostatic

Capability is available.

Electrostatic

Capability is available.

Charge

Same specification as Fluxes .

Current

Same specification as Fluxes .

Output Points

A single value at the centroid is given.

Element 38

Arbitrary Axisymmetric Triangle (Heat Transfer Element)

Element type 38 is a three-node, isoparametric, triangular element written for axisymmetric heat transfer applications. This element can also be used for electrostatic and magnetostatic analysis.

As this element uses bilinear interpolation functions, the thermal gradients tend to be constant throughout the element.

In general, one needs more of these lower-order elements than the higher-order elements such as type 132. Hence, use a fine mesh.

The conductivity of this element is formed using one-point integration at the centroid. The specific heat capacity matrix of this element is formed using four-point Gaussian integration.

Quick Reference

Type 38

Arbitrary, three-node, axisymmetric, heat transfer triangle.

Connectivity

Node numbering must follow right-handed convention (see Figure 5-55).

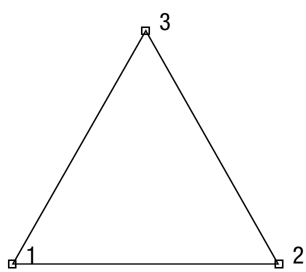


Figure 5-55 Axisymmetric Heat Transfer Triangle

Geometry

No geometry input required.

Coordinates

Two coordinates in the global z and r directions.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)
1 = potential (magnetostatic)

Fluxes

Flux types for distributed fluxes are as follows:

Flux Type	Description
0	Flux on 1-2 face of element (per unit area).
1	Volumetric flux on entire element (per unit volume).
3	Nonuniform flux given in the FLUX user subroutine on 1-2 face (per unit area).
4	Nonuniform volumetric flux given in the FLUX user subroutine on entire element (per unit volume).
6	Uniform flux on 2-3.
7	Nonuniform flux on 2-3.
8	Uniform flux on 3-1.
9	Nonuniform flux on 3-1.

For all nonuniform fluxes, the magnitude is given in the FLUX user subroutine.

All fluxes are positive when adding heat to the element. In addition, point fluxes can be applied at the nodes. Edge fluxes are evaluated using a two-point integration scheme where the integration points have the same location as the nodal points.

Films

Same specification as Fluxes .

Tying

Use the UFORMSN user subroutine.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Charge

Same specification as Fluxes .

Current

Same specification as Fluxes .

Output Points

A single value at the centroid is given.

Element 39

Bilinear Planar Quadrilateral (Heat Transfer Element)

Element type 39 is a four-node, isoparametric, arbitrary quadrilateral which can be used for planar heat transfer applications. This element can also be used for electrostatic or magnetostatic applications.

This element can also be used as a thermal contact or a fluid channel element. The CONRAD GAP and CHANNEL model definition options must be used for thermal contact and fluid channel options, respectively. A description of the thermal contact and fluid channel capabilities is included in Marc Volume A: Theory and User Information . Note that in thermal contact and fluid channel options, the gap face and fluid channel face identifications must be entered for each gap/channel. Face identifications for this element are given in the Face Identifications (Thermal Contact Gap and Fluid Channel Options) .

As this element uses bilinear interpolation functions, the thermal gradients tend to be constant throughout the element.

In general, one needs more of these lower-order elements than the higher-order elements such as types 41 or 69. Hence, use a fine mesh.

The conductivity of this element is formed using four-point Gaussian integration.

Quick Reference

Type 39

Arbitrary heat transfer quadrilateral.

Connectivity

Node numbering must follow the right-hand convention (see Figure 5-56).

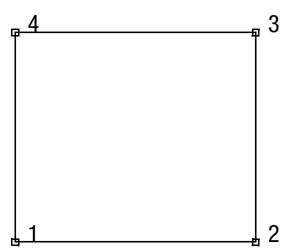


Figure 5-56 Heat Transfer Quadrilateral Element

Geometry

Thickness is input in the first data field (EGEOM1). The other two data fields (EGEOM2 and EGEOM3) are not used. If no thickness is input, unit thickness is assumed.

If a nonzero value is entered in the fourth data field (EGEOM4), the temperatures at the integration points obtained from interpolation of nodal temperatures are constant throughout the element.

Coordinates

Two global coordinates, x and y for planar applications.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)
1 = potential (magnetostatic)

Fluxes

Flux types for distributed fluxes are as follows:

Flux Type	Description
0	Uniform flux per unit area 1-2 face of the element.
1	Uniform flux per unit volume on whole element.
2	Uniform flux per unit volume on whole element.
3	Nonuniform flux per unit area on 1-2 face of the element; magnitude given in the FLUX user subroutine.
4	Nonuniform flux per unit volume on whole element; magnitude given in the FLUX user subroutine.
5	Nonuniform flux per unit volume on whole element; magnitude given in the FLUX user subroutine.
6	Uniform flux per unit area on 2-3 face of the element.
7	Nonuniform flux per unit area on 2-3 face of the element; magnitude given in the FLUX user subroutine.
8	Uniform flux per unit area on 3-4 face of the element.
9	Nonuniform flux per unit area on 3-4 face of the element; magnitude given in the FLUX user subroutine.
10	Uniform flux per unit area on 4-1 face of the element.
11	Nonuniform flux per unit area on 4-1 face of the element; magnitude given in the FLUX user subroutine.

For all nonuniform fluxes, the magnitude is given in the FLUX user subroutine.

All fluxes are positive when adding heat to the element. In addition, point fluxes can be applied at the nodes. Edge fluxes are evaluated using a two-point integration scheme where the integration points have the same location as the nodal points.

Films

Same specification as Fluxes .

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Current

Same specifications as Fluxes .

Charge

Same specifications as Fluxes .

Output Points

Centroid or four Gaussian integration points.

Tying

Use the UFORMSN user subroutine.

Face Identifications (Thermal Contact Gap and Fluid Channel Options)

Gap Face Identification	Radiative/Convective Heat Transfer Takes Place Between Edges
1	(1 - 2) and (3 - 4)
2	(1 - 4) and (2 - 3)

Fluid Channel Face Identification	Flow Direction From Edge to Edge
1	(1 - 2) to (3 - 4)
2	(1 - 4) to (2 - 3)

Element 40 — Axisymmetric Bilinear Quadrilateral Element (Heat Transfer Element)

Element type 40 is a four-node, isoparametric, arbitrary quadrilateral written for axisymmetric heat transfer applications. This element can also be used for electrostatic or magnetostatic applications.

As this element uses bilinear interpolation functions, the thermal gradients tend to be constant throughout the element.

This element can also be used as a thermal contact or a fluid channel element. The CONRAD GAP and CHANNEL model definition options must be used for thermal contact and fluid channel options, respectively. A description of the thermal contact and fluid channel capabilities is included in Marc Volume A: Theory and User Information . Note that in thermal contact and fluid channel options, the gap face and fluid channel face identifications must be entered for each gap/channel. Face identifications for this element are given in the Face Identifications (Thermal Contact Gap and Fluid Channel Options) .

In general, one needs more of these lower-order elements than the higher-order elements such as types 42 or 70. Hence, use a fine mesh.

The conductivity of this element is formed using four-point Gaussian integration.

Quick Reference

Type 40

Arbitrarily distorted axisymmetric heat transfer quadrilateral.

Connectivity

Node numbering must follow right-hand convention (see Figure 5-57).

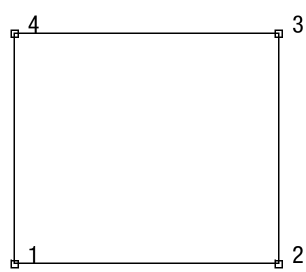


Figure 5-57 Axisymmetric Heat Transfer Quadrilateral

Geometry

If a nonzero value is entered in the fourth data field (EGEOM4), the temperatures at the integration points obtained from interpolation of nodal temperatures are constant throughout the element.

Coordinates

Two global coordinates, z and r.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)
1 = potential (magnetostatic)

Fluxes

Flux types for distributed fluxes are as follows:

Flux Type	Description
0	Uniform flux per unit area 1-2 face of the element.
1	Uniform flux per unit volume on whole element.
2	Uniform flux per unit volume on whole element.
3	Nonuniform flux per unit area on 1-2 face of the element; magnitude given in the FLUX user subroutine.
4	Nonuniform flux per unit volume on whole element; magnitude given in the FLUX user subroutine.
5	Nonuniform flux per unit volume on whole element; magnitude given in the FLUX user subroutine.
6	Uniform flux per unit area on 2-3 face of the element.
7	Nonuniform flux per unit area on 2-3 face of the element; magnitude given in the FLUX user subroutine.
8	Uniform flux per unit area on 3-4 face of the element.
9	Nonuniform flux per unit area on 3-4 face of the element; magnitude given in the FLUX user subroutine.
10	Uniform flux per unit area on 4-1 face of the element.
11	Nonuniform flux per unit area on 4-1 face of the element; magnitude given in the FLUX user subroutine.

For all nonuniform fluxes, the magnitude is given in the FLUX user subroutine.

All fluxes are positive when adding heat to the element. In addition, point fluxes can be applied at the nodes. Edge fluxes are evaluated using a two-point integration scheme where the integration points have the same location as the nodal points.

Films

Same specification as Fluxes .

Tying

Use the UFORMSN user subroutine.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Current

Same specifications as Fluxes .

Charge

Same specifications as Fluxes .

Output Points

Centroid or four Gaussian integration points.

Face Identifications (Thermal Contact Gap and Fluid Channel Options)

Gap Face Identification	Radiative/Convective Heat Transfer Takes Place Between Edges
1	(1 - 2) and (3 - 4)
2	(1 - 4) and (2 - 3)

Fluid Channel Face Identification	Flow Direction From Edge to Edge
1	(1 - 2) to (3 - 4)
2	(1 - 4) to (2 - 3)

View Factors Calculation for Radiation

Capability is available.

Element 41 Eight-node Planar Biquadratic Quadrilateral (Heat Transfer Element)

Element type 41 is an eight-node, isoparametric, arbitrary quadrilateral written for planar heat transfer applications. This element can also be used for electrostatic or magnetostatic applications.

This element uses biquadratic interpolation functions to represent the coordinates and displacements, hence the thermal gradients have a linear variation. This allows for accurate representation of the temperature field.

This element can also be used as a thermal contact or a fluid channel element. The CONRAD GAP and CHANNEL model definition options must be used for thermal contact and fluid channel options, respectively. A description of the thermal contact and fluid channel capabilities is included in Marc Volume A: Theory and User Information . Note that in thermal contact and fluid channel options, the gap face and fluid channel face identifications must be entered for each gap/channel. Face identifications for this element are given in the Face Identifications (Thermal Contact Gap and Fluid Channel Options) .

The conductivity of this element is formed using nine-point Gaussian integration.

Quick Reference

Type 41

Second-order, distorted heat quadrilateral.

Connectivity

Corner nodes 1-4 numbered first in right-handed convention. Nodes 5-8 are midside nodes starting with node 5 in between 1 and 2, and so on (see Figure 5-58).

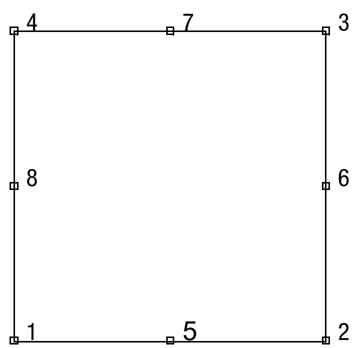


Figure 5-58 Eight-node Planar Heat Transfer Quadrilateral

Geometry

Thickness stored in EGEOM1 field. If not specified, unit thickness is assumed.

Coordinates

Two global coordinates per node – x and y for planar applications.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)
1 = potential (magnetostatic)

Fluxes

Edge Fluxes

Edge fluxes are specified as below. All are per unit surface area. All nonuniform fluxes are specified through the FLUX user subroutine.

Traction Type	Description
0	Uniform flux on 1-5-2 face.
1	Nonuniform flux on 1-5-2 face.
8	Uniform flux on 2-6-3 face.
9	Nonuniform flux on 2-6-3 face.
10	Uniform flux on 3-7-4 face.
11	Nonuniform flux on 3-7-4 face.
12	Uniform flux on 4-8-1 face.
13	Nonuniform flux on 4-8-1 face.

Volumetric Fluxes

Flux type 2 is uniform flux (per unit volume); type 3 is nonuniform flux per unit volume, with magnitude given through the FLUX user subroutine.

Films

Same specification as Fluxes .

Tying

Use the UFORMSN user subroutine.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Current

Same specifications as Fluxes .

Charge

Same specifications as Fluxes .

Output Points

Centroid or nine Gaussian integration points.

Face Identifications (Thermal Contact Gap and Fluid Channel Options)

Gap Face Identification	Radiative/Convective Heat Transfer Takes Place Between Edges
1	(1 - 5 - 2) and (3 - 7 - 4)
2	(4 - 8 - 1) and (2 - 6 - 3)

Fluid Channel Face Identification	Flow Direction From Edge to Edge
1	(1 - 5 - 2) to (3 - 7 - 4)
2	(4 - 8 - 1) to (2 - 6 - 3)

Element 42 Eight-node Axisymmetric Biquadratic Quadrilateral (Heat Transfer Element)

Element type 42 is an eight-node, isoparametric, arbitrary quadrilateral written for axisymmetric heat transfer applications. This element can also be used for electrostatic or magnetostatic applications.

This element uses biquadratic interpolation functions to represent the coordinates and displacements. Hence, the thermal gradients have a linear variation. This allows for accurate representation of the temperature field.

This element can also be used as a thermal contact or a fluid channel element. The CONRAD GAP and CHANNEL model definition options must be used for thermal contact and fluid channel options, respectively. A description of the thermal contact and fluid channel capabilities is included in Marc Volume A: Theory and User Information . Note that in thermal contact and fluid channel options, the gap face and fluid channel face identifications must be entered for each gap/channel. Face identifications for this element are given in the Face Identifications (Thermal Contact Gap and Fluid Channel Options) .

The conductivity of this element is formed using nine-point Gaussian integration.

Quick Reference

Type 42

Second-order, distorted axisymmetric heat quadrilateral.

Connectivity

Corner nodes 1-4 numbered first in right-handed convention. Nodes 5-8 are midside nodes with node 5 located between nodes 1 and 2, etc. (see Figure 5-59).

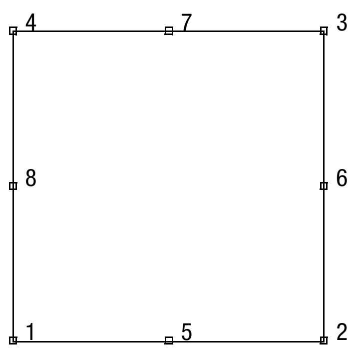


Figure 5-59 Eight-node Axisymmetric Heat Transfer Quadrilateral

Geometry

Not applicable.

Coordinates

Two global coordinates per node, z and r.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)
1 = potential (magnetostatic)

Fluxes

Surface Fluxes

Surfaces fluxes are specified as below. All are per unit surface area. All nonuniform fluxes are specified through the FLUX user subroutine.

Flux Type (IBODY)	Description
0	Uniform flux on 1-5-2 face.
1	Nonuniform flux on 1-5-2 face.
8	Uniform flux on 2-6-3 face.
9	Nonuniform flux on 2-6-3 face.
10	Uniform flux on 3-7-4 face.
11	Nonuniform flux on 3-7-4 face.
12	Uniform flux on 4-8-1 face.
13	Nonuniform flux on 4-8-1 face.

Volumetric Fluxes

Flux type 2 is uniform flux (per unit volume); type 3 is nonuniform flux per unit volume, with magnitude given through the FLUX user subroutine.

Films

Same specification as Fluxes .

Tying

Use the UFORMSN user subroutine.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Current

Same specifications as Fluxes .

Charge

Same specifications as Fluxes .

Output Points

Centroid or nine Gaussian integration points.

Face Identifications (Thermal Contact Gap and Fluid Channel Options)

Gap Face Identification	Radiative/Convective Heat Transfer Takes Place Between Edges
1	(1 - 5 - 2) and (3 - 7 - 4)
2	(4 - 8 - 1) and (2 - 6 - 3)

Fluid Channel Face Identification	Flow Direction From Edge to Edge
1	(1 - 5 - 2) to (3 - 7 - 4)
	(4 - 8 - 1) to (2 - 6 - 3)

View Factors Calculation for Radiation

Capability is available.

Element 43

Three-dimensional Eight-node Brick (Heat Transfer Element)

Element type 43 is an eight-node, isoparametric, arbitrary hexahedral written for three-dimensional heat transfer applications. This element can also be used for electrostatic applications.

As this element uses trilinear interpolation functions, the thermal gradients tend to be constant throughout the element.

This element can also be used as a thermal contact or a fluid channel element. The CONRAD GAP and CHANNEL model definition options must be used for thermal contact and fluid channel options, respectively. A description of the thermal contact and fluid channel capabilities is included in Marc Volume A: Theory and User Information . Note that in thermal contact and fluid channel options, the gap face and fluid channel face identifications must be entered for each gap/channel. Face identifications for this element are given in the Face Identifications (Thermal Contact Gap and Fluid Channel Options) .

In general, one needs more of these lower order elements than the higher order elements such as 44 or 71. Hence, use a fine mesh.

The conductivity of this element is formed using eight-point Gaussian integration.

Quick Reference

Type 43

Eight-node, three-dimensional, first-order isoparametric heat transfer element.

Connectivity

Eight nodes per element.

Nodes 1-4 are the corners on one face, numbered in a counterclockwise direction when viewed from inside the element. Nodes 5-8 are the nodes on the other face, with node 5 opposite node 1, and so on (see Figure 5-60).

Geometry

If a nonzero value is entered in the fourth data field (EGEOM4), the temperatures at the integration points obtained from interpolation of nodal temperatures are constant throughout the element.

Coordinates

Three coordinates in the global x, y, and z directions.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)

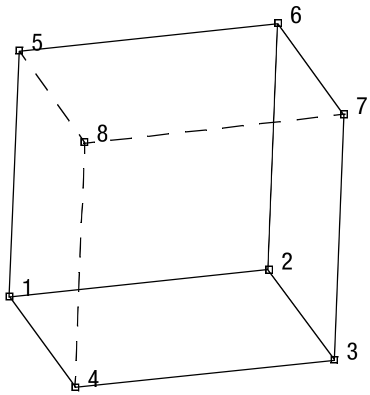


Figure 5-60 Arbitrarily Distorted Heat Transfer Cube

Fluxes

Fluxes are distributed according to the appropriate selection of a value of IBODY. Surface fluxes are assumed positive when directed into the element and are evaluated using a 4-point integration scheme, where the integration points have the same location as the nodal points.

Load Type (IBODY)	Description
0	Uniform flux on 1-2-3-4 face.
1	Nonuniform surface flux (supplied via the FLUX user subroutine) on 1-2-3-4 face.
2	Uniform volumetric flux.
3	Nonuniform volumetric flux (with the FLUX user subroutine).
4	Uniform flux on 5-6-7-8 face.
5	Nonuniform surface flux on 5-6-7-8 face (with the FLUX user subroutine).
6	Uniform flux on 1-2-6-5 face.
7	Nonuniform flux on 1-2-6-5 face (with the FLUX user subroutine).
8	Uniform flux on 2-3-7-6 face.
9	Nonuniform flux on 2-3-7-6 face (with the FLUX user subroutine).
10	Uniform flux on 3-4-8-7 face.
11	Nonuniform flux on 3-4-8-7 face (with the FLUX user subroutine).
12	Uniform flux on 1-4-8-5 face.
13	Nonuniform flux on 1-4-8-5 face (with the FLUX user subroutine).

For `IBODY= 3`, P is the magnitude of volumetric flux at volumetric integration point NN of element N. For `IBODY` odd but not equal to 3, P is the magnitude of surface flux for surface integration point NN of element N.

Films

Same specification as Fluxes .

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Current

Same specification as Fluxes .

Charge

Same specifications as Fluxes .

Output Points

Centroid or eight Gaussian integration points (see Figure 5-61).

Note:	As in all three-dimensional analysis, a large nodal bandwidth results in long computing times. Use the optimizers as much as possible.

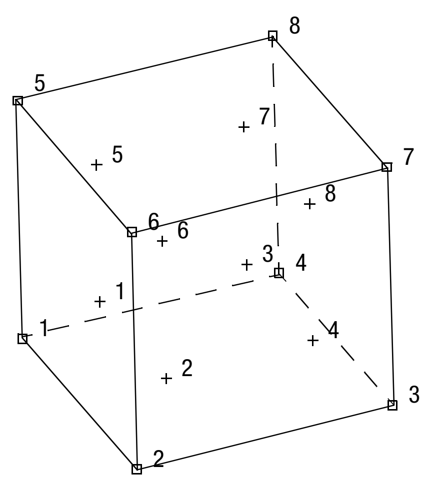


Figure 5-61 Integration Points for Element 43

Face Identifications (Thermal Contact Gap and Fluid Channel Options)

Gap Face Identification	Radiative/Convective Heat Transfer Takes Place Between Faces
1	(1 - 2 - 6 - 5) and (3 - 4 - 8 - 7)
2	(2 - 3 - 7 - 6) and (1 - 4 - 8 - 5)
3	(1 - 2 - 3 - 4) and (5 - 6 - 7 - 8)

Fluid Channel Face Identification	Flow Direction From Edge to Edge
1	(1 - 2 - 6 - 5) to (3 - 4 - 8 - 7)
2	(2 - 3 - 7 - 6) to (1 - 4 - 8 - 5)
3	(1 - 2 - 3 - 4) to (5 - 6 - 7 - 8)

Element 44 Three-dimensional 20-node Brick (Heat Transfer Element)

Element type 44 is a 20-node isoparametric arbitrary hexahedral written for three-dimensional heat transfer applications. This element can also be used for electrostatic applications.

This element uses triquadratic interpolation functions to represent the coordinates and displacements. Hence, the thermal gradients have a linear variation. This allows for accurate representation of the temperature field.

This element can also be used as a thermal contact or a fluid channel element. The CONRAD GAP and CHANNEL model definition options must be used for thermal contact and fluid channel options, respectively. A description of the thermal contact and fluid channel capabilities is included in Marc Volume A: Theory and User Information . Note that in thermal contact and fluid channel options, the gap face and fluid channel face identifications must be entered for each gap/channel. Face identifications for this element are given in the Face Identifications (Thermal Contact Gap and Fluid Channel Options) .

The conductivity of this element is formed using 27-point Gaussian integration.

Connectivity

The convention for the ordering of the connectivity array is as follows:

> Nodes 1, 2, 3, and 4 are the corners of one face, given in a counterclockwise direction when viewed from inside the element. Nodes 5, 6, 7, 8 are the corners of the opposite face; node 5 shares an edge with 1; node 6 shares an edge with 2, etc. Nodes 9, 10, 11, 12 are the middles of the edges of the 1, 2, 3, 4 face; node 9 between 1 and 2; node 10 between 2 and 3, etc. Similarly, nodes 13, 14, 15, 16 are midpoints on the 5, 6, 7, 8 face; node 13 between 5 and 6, etc. Finally, node 17 is the midpoint of the 1-5 edge; node 18 of the 2-6 edge, etc. (see Figure 5-62).

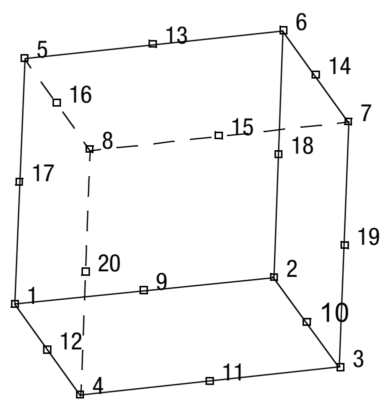


Figure 5-62 Twenty-node Heat Transfer Brick

Note that in most normal cases, the elements are generated automatically so you need not concern yourself with the node numbering scheme.

Integration

The element is integrated numerically using 27 points (Gaussian quadrature). The first plane of such points is closest to the 1, 2, 3, 4 face of the element, with the first point closest to the first node of the element (see Figure 5-63). Two similar planes follow, moving toward the 5, 6, 7, 8 face. Thus, point 14 represents the "centroid" of the element, and is used for temperature output if the CENTROID parameter is used.

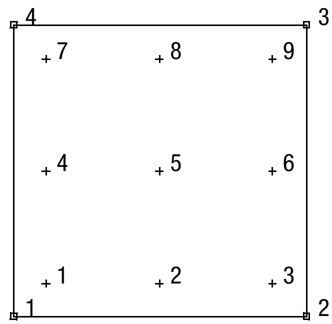


Figure 5-63 Points of Integration in a Sample Integration Plane

Fluxes

Distributed fluxes chosen by value of `IBODY`.

Reduction to Wedge or Tetrahedron

The element can be reduced as far as a tetrahedron, simply by repeating node numbers at the same spatial position. Element type 133 is preferred for tetrahedrals.

Notes:	A large bandwidth results in long run times. Optimize as much as possible.
	The lumped specific heat option gives poor results with this element at early times in transient solutions. If accurate transient analysis is required, you should not use the lumping option with this element.

Quick Reference

Type 44

Twenty-node isoparametric brick (heat transfer element).

Connectivity

Twenty nodes numbered as described in the connectivity write-up for this element, and as shown in Figure 5-62.

Geometry

Not applicable.

Coordinates

Three global coordinates in the x, y, and z directions.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)

Fluxes

Load Type	Description
0	Uniform flux on 1-2-3-4 face.
1	Nonuniform flux on 1-2-3-4 face (with the FLUX user subroutine).
2	Uniform body flux.
3	Nonuniform body flux (with the FLUX user subroutine).
4	Uniform flux on 6-5-8-7 face.
5	Nonuniform flux on 6-5-8-7 face (with the FLUX user subroutine).
6	Uniform flux on 2-1-5-6 face.
7	Nonuniform flux on 2-1-5-6 face (with the FLUX user subroutine).
8	Uniform flux on 3-2-6-7 face.
9	Nonuniform flux on 3-2-6-7 face (with the FLUX user subroutine).
10	Uniform flux on 4-3-7-8 face.
11	Nonuniform flux on 4-3-7-8 face (with the FLUX user subroutine).
12	Uniform flux on 1-4-8-5 face.
13	Nonuniform flux on 1-4-8-5 face (with the FLUX user subroutine).

For `IBODY=3`, the value of P in the FLUX user subroutine is the magnitude of volumetric flux at volumetric integration point NN of element N. For `IBODY` odd but not equal to 3, P is the magnitude of surface flux at surface integration point NN of element N. Surface flux is positive when heat energy is added to the element.

Films

Same specification as Fluxes .

Tying

Use the UFORMSN user subroutine.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Current

Same specifications as Fluxes .

Charge

Same specifications as Fluxes .

Output Points

Centroid or 27 Gaussian integration points.

Face Identifications (Thermal Contact Gap and Fluid Channel Options)

Gap Face Identification	Radiative/Convective Heat Transfer Takes Place Between Faces
1	(1 - 2 - 6 - 5) and (3 - 4 - 8 - 7)
2	(2 - 3 - 7 - 6) and (1 - 4 - 8 - 5)
3	(1 - 2 - 3 - 4) and (5 - 6 - 7 - 8)

Fluid Channel Face Identification	Flow Direction From Edge to Edge
1	(1 - 2 - 6 - 5) to (3 - 4 - 8 - 7)
2	(2 - 3 - 7 - 6) to (1 - 4 - 8 - 5)
3	(1 - 2 - 3 - 4) to (5 - 6 - 7 - 8)

Element 45

Curved Timoshenko Beam in a Plane

This is a three-node planar beam element which allows transverse shear as well as axial straining. It is based on a quadratic displacement assumption on the global displacements and rotation. The strain-displacement relationships are complete except for large curvature change terms (consistent with the other beam and shell elements in Marc). The shear strain is assumed to be quadratic across the thickness. The default cross section is a rectangle. Integration for section properties uses a Simpson rule across the section. Integration for element stiffness and mass uses a two- and three-point Gauss rule along the beam axis. All constitutive relations can be used with this element type.

Quick Reference

Type 45

Curved, planar Timoshenko beam.

Connectivity

Three nodes per element (see Figure 5-64).

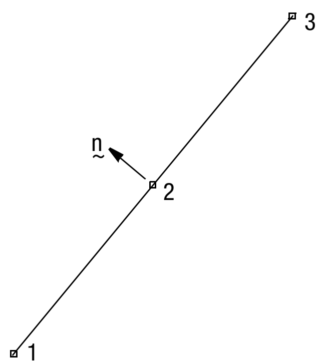


Figure 5-64 Three-node Timoshenko Beam in a Plane

Geometry

Linear thickness variation along the element. Thickness at first node of the element stored in first data field (`EGEOM1`).

Thickness at third node of the element stored in third data field (`EGEOM3`). If `EGEOM3=0`, constant thickness is assumed.

Beam width (normal to the plane of deformation) stored in second data field, `EGEOM2`. The default width is unity.

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

Beam Offsets Pin Codes and Local Beam Orientation

If an offset is applied to the beam geometry or pin codes are applied to the degrees of freedom of the end point of the beams or the beam axis is defined with respect to a local coordinate system, then a code is placed in the 8th field of the 3rd data block of the GEOMETRY option (`EGEOM8`). This code is formed by the negative sum of ioffset, iorien, and ipin where:

ioffset	=	0	-	no offset
		1	-	beam offset
iorien	=	0	-	conventional definition of local beam orientation; beam axis given in 4th through 6th field with respect to global system
		10	-	the local beam orientation entered in the 4th through 6th field is given with respect to the coordinate system of the first beam node. This vector is transformed to the global coordinate system before any further processing.
ipin	=	0	-	no pin codes are used
		10 0	-	pin codes are used.

If ioffset = 1, an additional line is read in the GEOMETRY option (4th data block).

If ipin = 100, an additional line is read in the GEOMETRY option (5th data block)

The offset vector at the first corner node is provided in the first three fields and the offset vector at the second corner node is provided in the next three fields of the extra line. If the interpolation flag (seventh data field) is set to 1, the offset vector at the midside node is obtained by interpolating from the corresponding corner vectors. The coordinate system used for the offset vectors at the two nodes is indicated by the corresponding flags in the eighth and ninth fields of the extra line. A flag setting of 0 indicates global coordinate system, 1 indicates local element coordinate system and 3 indicates local nodal coordinate system. When the flag setting is 1, the local beam coordinate system is constructed by taking the local x axis along the beam, local z axis as [0,0,1], and local y axis as perpendicular to both.

Another way to input a pin code is by using the PIN CODE model definition option. This option is supported by Mentat while the input using the GEOMETRY option is not.

Coordinates

Two coordinates at all nodes:

1 = x
2 = y

Right-handed coordinate set.

Degrees of Freedom

1 = u (global x component of displacement)
2 = v (global y component of displacement)
3 = ϕ_s = rotation of the cross section (right-handed rotation)

Tractions

Distributed loads. Selected with load type are as follows:

Load Type	Description
0	Uniform normal force per beam length as shown in Figure 5-64. Positive pressure is in the negative normal (n) direction.
1	Uniform load (force per beam length) in global x-direction.
2	Uniform load (force per beam length) in global y-direction.
3	Nonuniform load (force per beam length) in global x-direction. (FORCEM user subroutine).
4	Nonuniform load (force per beam length) in global y-direction. (FORCEM user subroutine).
5	Nonuniform normal force per beam length as shown in Figure 5-64 (FORCEM user subroutine). Positive pressure is in the negative normal (n) direction.
11	Fluid drag/buoyancy loading – fluid behavior specified in the FLUID DRAG option.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For nonuniform loads (types 3, 4, 5) the magnitude is supplied at each of the three Gauss points via the FORCEM user subroutine.

Concentrated loads and moments can be applied at the nodes.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strain and Stress

Two values of strains and stresses are stored at each of the integration points through the thickness:

1 = axial
2 = transverse shear

The first point of the section is on the surface up the positive normal (opposite to the positive pressure), the last point is on the opposite surface.

Transformation

Allowable in x-y plane.

Output Points

Centroidal section or the two Gauss points. The first Gauss point is closest to the first node.

Section Stress Integration

Integration through the thickness is performed numerically using Simpson's rule. Use the SHELL SECT parameter to define the number of integration points. This number must be odd.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – output of stress and strain as for total Lagrangian approach. Thickness is updated, but beam width is assumed to be constant.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 65. See Element 65 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness (beam height) and/or the beam width can be considered as design variables.

Element 46

Eight-node Plane Strain Rebar Element

This element is isoparametric, plane strain, 8-node hollow quadrilateral in which you can place single strain members such as reinforcing rods or cords (that is, rebars). The element is then used in conjunction with the 8-node plane strain continuum element (for example, element 27 or 32) to represent cord reinforced composite materials. This technique allows the rebar and the filler to be represented accurately with respect to their stress distribution, so that separate constitutive theories can be used in each (e.g., cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via the REBAR option or via the REBAR user subroutine.

Integration

It is assumed that several "layers" of rebars are presented. The number of such "layers" is input by you via REBAR option or, if the REBAR user subroutine is used, in the second element geometry field. A maximum number of five layers can be used within a rebar element. Each layer is assumed to be similar to a pair of opposite element edges (although the rebar direction is arbitrary), so that the "thickness" direction is from one of the element edges to its opposite one. For instance (see Figure 5-65), if the layer is similar to the 1, 2 and 3, 4 edges of the element, the "thickness" direction is from the 1, 2 edge to 3, 4 edge of the element. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains two integration points. At each such integration point on each layer, you must input, via either the REBAR option or the REBAR user subroutine, the position, equivalent thickness (or, alternatively, spacing and area of cross section), and orientation of the rebars. See Marc Volume C: Program Input for REBAR option or Marc Volume D: User Subroutines and Special Routines for the REBAR user subroutine.

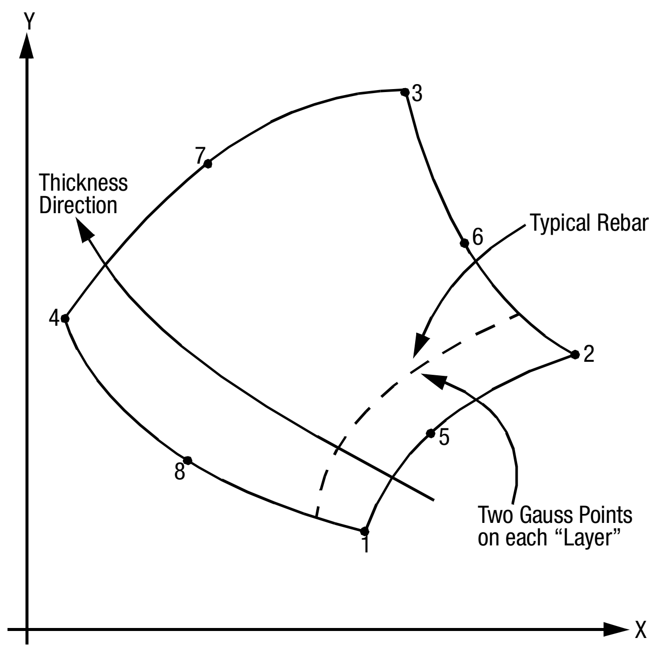


Figure 5-65 Eight-node Rebar Element Conventions

Quick Reference

Type 46

Eight-node, isoparametric rebar element to be used with 8-node plane strain continuum element.

Connectivity

Eight nodes per element. Node numbering of the element is same as that for element 27 or 32.

Geometry

Element thickness (in z-direction) in first field. Default thickness is unity. Note, this should not be confused with the "thickness" concept associated with rebar layers.

The number of rebar layers and the isoparametric direction of the layers are defined using the REBAR model definition option.

Coordinates

Two global coordinates x- and y-directions.

Degrees of Freedom

Displacement output in global components is as follows:

1 - u
2 - v

Tractions

Point loads can be applied at the nodes, but no distributed loads are available. Distributed loads are applied only to corresponding 8-node plane strain elements (for example, element types 27 or 32).

Output of Stress and Strain

One stress and one strain are output at each integration point – the axial rebar value. For the case of large deformations, the stress is the Second Piola-Kirchhoff stress and the strain is the Green strain.

By using post code 471 and 481 (representing Second Piola-Kirchhoff stress in undeformed configuration and Cauchy stress in deformed configuration, respectively), the rebar stress can be written into the post file in the form of a stress tensor defined in the global coordinate directions. Mentat can be used to plot the principal directions of the stress tensor to show the magnitude of rebar stress, rebar orientation, and their changes based on deformation.

Transformation

Any local set (u,v) can be used at any node.

Special Consideration

Either REBAR option or the REBAR user subroutine is needed to input the position, size, and orientation of the rebars.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Element 47

Generalized Plane Strain Rebar Element

This element is similar to element 46, but is written for generalized plane strain conditions. It is a hollow ten-node planar quadrilateral in which you can place single strain members such as reinforcing rods or cords (that is, rebars). The element is then used in conjunction with the corresponding generalized plane strain continuum element (for example, element types 29 or 34) to represent a reinforced composite materials. This technique allows the rebar and the filler to be represented accurately with respect to their stress distribution, so that separate constitutive theories can be used in each (for example, cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via REBAR option or via the REBAR user subroutine.

Integration

It is assumed that several "layers" of rebars are presented. The number of such "layers" is input by you via the REBAR option or, if the REBAR user subroutine is used, in the second element geometry field. A maximum number of five layers can be used within a rebar element. Each layer is assumed to be similar to a pair of opposite element edges (although the rebar direction is arbitrary), so that the "thickness" direction is from one of the element edges to its opposite one. For instance (see Figure 5-66), if the layer is similar to the 1, 2 and 3, 4 edges of the element, the "thickness" direction is from the 1, 2 edge to 3, 4 edge of the element. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains two integration points. At each such integration point on each layer, you must input via either the REBAR option or the REBAR user subroutine the position, equivalent thickness (or, alternatively, spacing and area of cross section), and orientation of the rebars. See Marc Volume C: Program Input for REBAR option or Marc Volume D: User Subroutines and Special Routines for the REBAR user subroutine.

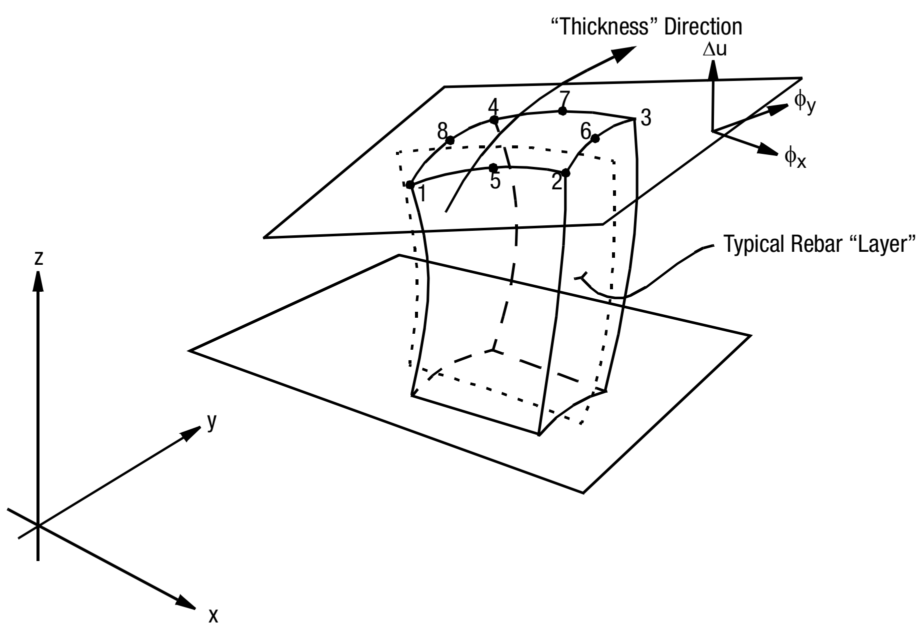


Figure 5-66 Ten-node Generalized Plane Strain Rebar Element Conventions

Quick Reference

Type 47

Ten-node, generalized plane strain rebar element.

Connectivity

Node numbering of the element is same as that for element 29 or 34.

Geometry

Element thickness (in the z-direction) in first field. Default thickness is unity. Note, this should not be confused with "thickness" concept associated with rebar layers.

The number of rebar layers and the isoparametric direction of the layers are defined using the REBAR model definition option.

Coordinates

Two global coordinates in x- and y-directions.

Degrees of Freedom

At the first eight nodes, global x and y displacements.

At node 9, the relative translation of the top surface of the element with respect to the bottom surface.

At node 10, the relative rotations of the top surface of the element with respect to the bottom surface.

Tractions

Point loads can be applied at the nodes, but no distributed loads are available. Distributed loads are applied only to corresponding generalized plane strain elements (e.g., element types 29 or 34).

Output Of Stress and Strain

One stress and one strain are output at each integration point – the axial rebar value. For the case of large deformations, the stress is the Second Piola-Kirchhoff stress and the strain is the Green strain.

By using post code 471 and 481 (representing Second Piola-Kirchhoff stress in undeformed configuration and Cauchy stress in deformed configuration, respectively), the rebar stress can be written into the post file in the form of a stress tensor defined in the global coordinate directions. Mentat can be used to plot the principal directions of the stress tensor to show the magnitude of rebar stress, rebar orientation, and their changes based on deformation.

Transformation

Any local set (u,v) can be used at the first 8 nodes.

Special Considerations

Either REBAR option or the REBAR user subroutine is needed to input the position, size, and orientation of the rebars.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Element 48

Eight-node Axisymmetric Rebar Element

This element is similar to element 46, but is written for axisymmetric conditions. It is a hollow, isoparametric 8-node quadrilateral in which you can place single strain members such as reinforcing rods or cords (that is, rebars). The element is then used in conjunction with the 8-node axisymmetric continuum element (for example, element 28 or 33) to represent cord reinforced composite materials. This technique allows the rebar and the filler to be represented accurately with respected to their stress distribution, so that separate constitutive theories can be used in each (for example, cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via REBAR option or via the REBAR user subroutine.

Integration

It is assumed that several “layers” of rebars are presented. The number of such “layers” is input by you via the REBAR option or, if the REBAR user subroutine is used, in the second element geometry field. A maximum number of five layers can be used within a rebar element. Each layer is assumed to be similar to a pair of opposite element edges (although the rebar direction is arbitrary), so that the “thickness” direction is from one of the element edges to its opposite one. For instance (see Figure 5-67), if the layer is similar to the 1, 2 and 3, 4 edges of the element, the “thickness” direction is from the 1, 2 edge to 3, 4 edge of the element. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains two integration points.

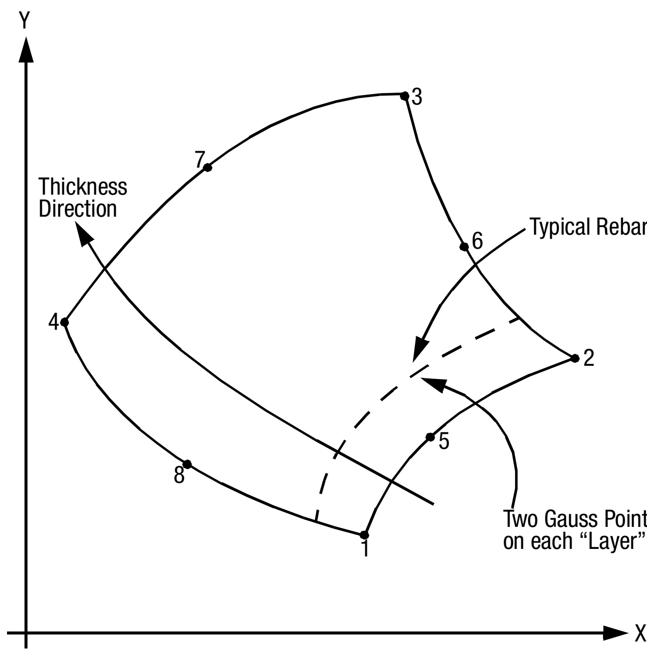


Figure 5-67 Eight-node Rebar Element Conventions

At each such integration point on each layer, you must input, via either the REBAR option or the REBAR user subroutine, the position, equivalent thickness (or, alternatively, spacing and area of cross section), and orientation of the rebars. See Marc Volume C: Program Input for REBAR option or Marc Volume D: User Subroutines and Special Routines for the REBAR user subroutine.

Quick Reference

Type 48

Eight-node, isoparametric rebar element to be used with 8-node axisymmetric continuum element.

Connectivity

Eight nodes per element. Node numbering of the element is same as that for element 28 or 33.

Geometry

The number of rebar layers and the isoparametric direction of the layers are defined using the REBAR model definition option.

Coordinates

Two global coordinates in z- and r-directions.

Degrees of Freedom

Displacement output in global components is as follows:

1 - z
2 - r

Tractions

Point loads can be applied at the nodes but no distributed loads are available. Distributed loads are applied only to corresponding 8-node axisymmetric elements (for example, element types 28 or 33).

Output of Stress and Strain

One stress and one strain are output at each integration point – the axial rebar value. For the case of large deformations, the stress is the Second Piola-Kirchhoff stress and the strain is the Green strain.

By using post code 471 and 481 (representing Second Piola-Kirchhoff stress in undeformed configuration and Cauchy stress in deformed configuration, respectively), the rebar stress can be written into the post file in the form of a stress tensor defined in the global coordinate directions. Mentat can be used to plot the principal directions of the stress tensor to show the magnitude of rebar stress, rebar orientation, and their changes based on deformation.

Transformations

Any local set (u,v) can be used in the (z-r) plane at any node.

Special Considerations

Either the REBAR option or the REBAR user subroutine is needed to input the position, size, and orientation of the rebars.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Element 49

Finite Rotation Linear Thin Shell Element

Element type 49 is a finite rotation, six-node, thin shell element. The in-plane displacements and the in-plane coordinates are linearly interpolated while the out-of-plane displacement and coordinate are quadratically interpolated. This quadratic interpolation provides the possibility to model slightly curved elements for which the influence of the (changes of) curvature on the membrane deformations is taken into account. This influence is especially important in cases where pressure loads have to be carried mainly by membrane forces and in cases of (nearly) inextensional bending. By default, this influence is taken into account. The element can also be used as a flat plate element by entering a nonzero value on the fifth (`EGEOM5`) geometry data field. In that case, the influence of the curvature on the membrane deformations is not taken into account, and the coordinates of the midside nodes are calculated as the average of the corresponding corner nodes. The degrees of freedom consist of three global translational degrees of freedom for the corner nodes and one local rotational degree of freedom at the midside nodes. These rotational degrees of freedom represent the average rotation of the surface normal about the element edges (see Figure 5-68).

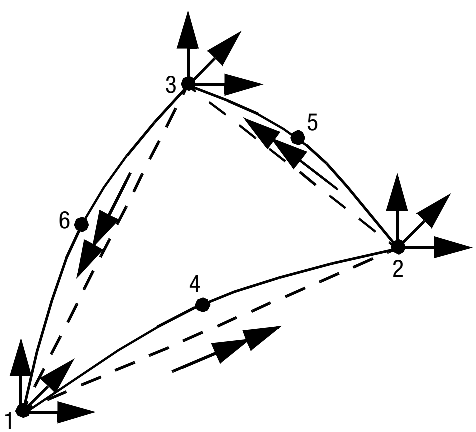


Figure 5-68 Element Type 49 Degrees of Freedom

This element does not suffer from the restriction that the incremental rotations must remain small provided that the LARGE DISP parameter is used.

Element type 49 has only one integration point. Together with the relatively small number of degrees of freedom, this element is very effective from a computational point of view.

All constitutive relations can be used with this element.

Geometric Basis

The first two element base vectors lie in the plane of the three corner nodes. The first one (V_1) points from node 1 to node 2; the second one (V_2) is perpendicular to V_1 and lies on the same side as node 3. The third one (V_3) is determined such that V_1, V_2, and V_3 form a right-hand system. With respect to the set of base vectors (V_1, V_2, V_3), the generalized as well as the layer stresses and strains are given for the Gaussian integration point which coordinates readily follow from the average of the corner node coordinates (see Figure 5-69). The triangle determined by the corner nodes is called the basic triangle.

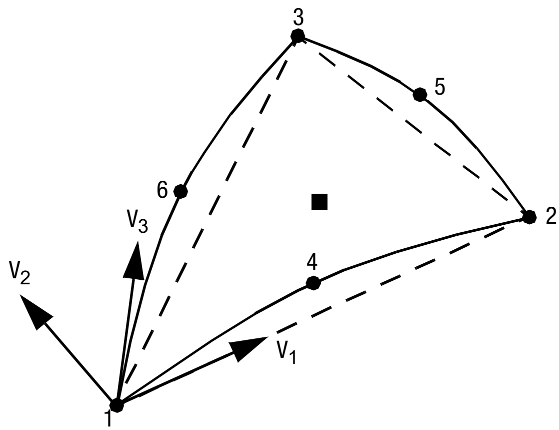


Figure 5-69 Element Type 49 Local Base Vectors and Integration Point Position

Degrees of Freedom

The nodal degrees of freedom are as follows:

At the three corner nodes:	u, v, w Cartesian displacement components.
At the three midside nodes:	ϕ, rotation of the surface normal about the element edge. The positive rotation vector points from the corner with the lower (external) node number to the node with the higher (external) node number.

Using these degrees of freedom, modeling of intersecting plates can be done without special tying types.

Quick Reference

Type 49

Linear, six-node shell element.

Connectivity

Six nodes. Corners given first, proceeding continuously around the element.

Then the midside nodes are given as follows:

4 = Between corners 1 and 2
5 = Between corners 2 and 3
6 = Between corners 3 and 1

Geometry

Linear thickness variation can be specified in the plane of the element. Internally, the average thickness is used. Thicknesses at first, second, and third node are stored for each element in the first (`EGEOM1`), second (`EGEOM2`), and third (`EGEOM3`) geometry data field, respectively. If `EGEOM2-EGEOM3` are zero, then a constant thickness (`EGEOM1`) is assumed for the element.

Alternatively, the NODAL THICKNESS model definition option can be used for the input of the element thickness.

If a nonzero value is entered on the fifth (EGEOM5) GEOMETRY data field, the element is considered to be flat.

Coordinates

(x, y, z) global Cartesian coordinates are given. If the coordinates of the midside nodes are not given, they are calculated as the average of the coordinates of the corresponding corner nodes.

Degrees of Freedom

At the three corner nodes:

1 = u = global Cartesian displacement in x-direction
2 = v = global Cartesian displacement in y-direction
3 = w = global Cartesian displacement in z-direction

At the three midside nodes:

1 = ϕ = rotation of surface normal about the edge. Positive rotation vector points from the corner with the lower (external) node number to the corner with the higher (external) node number.

Distributed Loading

Types of distributed loading are as follows:

Load Type	Description
1	Uniform gravity load per surface area in -z direction.
2	Uniform pressure; positive magnitude in $-V_3$ direction.
3	Uniform gravity load per surface area in -z direction; magnitude given in the FORCEM user subroutine.
4	Nonuniform pressure; magnitude given in the FORCEM user subroutine, positive magnitude given in $-V_3$ direction.
5	Nonuniform pressure; magnitude and direction given in the FORCEM user subroutine.
11	Uniform edge load in the plane of the basic triangle on the 1-2 edge and perpendicular to the edge.
12	Uniform edge load in the plane of the basic triangle on the 1-2 edge and tangential to the edge.
13	Nonuniform edge load in the plane of the basic triangle on the 1-2 edge and perpendicular to the edge; magnitude given in the FORCEM user subroutine.
14	Nonuniform edge load in the plane of the basic triangle on the 1-2 edge and tangential to the edge; magnitude given in the FORCEM user subroutine.
15	Nonuniform edge load in the plane of the basic triangle on the 1-2 edge; magnitude and direction given in the FORCEM user subroutine.
16	Uniform load on the 1-2 edge of the shell. Perpendicular to the shell; i.e., $-v_3$ direction
21	Uniform edge load in the plane of the basic triangle on the 2-3 edge and perpendicular to the edge.
22	Uniform edge load in the plane of the basic triangle on the 2-3 edge and tangential to the edge.

Load Type	Description
23	Nonuniform edge load in the plane of the basic triangle on the 2-3 edge and perpendicular to the edge; magnitude given in the FORCEM user subroutine.
24	Nonuniform edge load in the plane of the basic triangle on the 2-3 edge and tangential to the edge; magnitude given in the FORCEM user subroutine.
25	Nonuniform edge load in the plane of the basic triangle on the 2-3 edge; magnitude and direction given in the FORCEM user subroutine.
26	Uniform load on the 2-3 edge of the shell. Perpendicular to the shell; i.e., $-v_3$ direction
31	Uniform edge load in the plane of the basic triangle on the 3-1 edge and perpendicular to the edge.
32	Uniform edge load in the plane of the basic triangle on the 3-1 edge and tangential to the edge.
33	Nonuniform edge load in the plane of the basic triangle on the 3-1 edge and perpendicular to the edge; magnitude given in the FORCEM user subroutine.
34	Nonuniform edge load in the plane of the basic triangle on the 3-1 edge and tangential to the edge; magnitude given in the FORCEM user subroutine.
35	Nonuniform edge load in the plane of the basic triangle on the 3-1 edge; magnitude and direction given in the FORCEM user subroutine.
36	Uniform load on the 3-1 edge of the shell. Perpendicular to the shell; i.e., $-v_3$ direction
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity load in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.

All edge loads require the input as force per unit length.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output Of Strains

Generalized strain components are as follows:

middle surface stretches ε_{11}, ε_{22}, ε_{12}

middle surface curvatures κ_{11}, κ_{22}, κ_{12}

in local (V_1, V_2, V_3) system.

Output Of Stress

Generalized stress components are as follows:

Tangential stress resultants σ_{11}, σ_{22}, σ_{12}

Tangential stress couples μ_{11}, μ_{22}, μ_{12}

all in local (V_1, V_2, V_3) system.

Stress components:

σ_{11}, σ_{22}, σ_{12} in local (V_1, V_2, V_3) system at equally spaced layers through the thickness. First layer is on positive V_3 direction surface.

Transformation

Displacement components at corner nodes can be transformed to local directions.

Tying

Use the UFORMSN user subroutine.

Output Points

Output occurs at the centroid of the element.

Section Stress Integration

Integration through the thickness is performed numerically using Simpson's rule. Use the SHELL SECT parameter to define the number of integration points. This number must be odd.

Beam Stiffeners

For small rotational increments, element type 49 is compatible with beam element types 76 and 77.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – output for stresses and strains as for total Lagrangian approach. Notice that if only UPDATE is used, the subsequent increments are based upon linear strain-displacement relations. If both UPDATE and LARGE DISP are used, the full nonlinear strain-displacement relations are used for each increment.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 50. See Element 50 for a description of the convention used for entering the flux and film data for this element.

Element 50

Three-node Linear Heat Transfer Shell Element

This is a three-node heat transfer shell element with temperatures as nodal degrees of freedom. A linear interpolation is used for the temperatures in the plane of the shell and either a linear or a quadratic temperature distribution is assumed in the shell thickness direction (see the HEAT parameter).

In the plane of the shell, a one-point Gaussian integration is used for the evaluation of the conductivity matrix and a three-point Gaussian integration for the evaluation of the heat capacity matrix. In the thickness direction of the shell, Simpson's rule is used where the number of point can be given by the SHELL SECT parameter (the default number is 11).

This element is compatible with element types 49 and 138 in a thermal-stress analysis and can be used in conjunction with three-dimensional heat transfer brick elements through a tying for heat transfer analyses.

Geometric Basis

The element is defined geometrically by the (x, y, and z) coordinates of the three corner nodes. The thickness is specified using either the GEOMETRY or NODAL THICKNESS option. The first two element base vectors lie in the plane of the three corner nodes. The first one (V_1) points from node 1 to node 2; the second one (V_2) stands perpendicular to V_1 and lies on the same side as node 3. The third one (V_3) is determined such that V_1, V_2 and V_3 form a right-hand system (see Figure 5-70).

In addition, this element can be used for an electrostatic problem. A description of this option can be found in Marc Volume A: Theory and User Information.

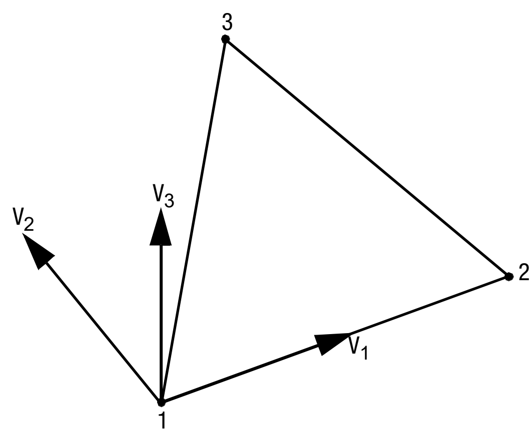


Figure 5-70 Element Type 50, Local Base Vectors

Quick Reference

Type 50

Three-node linear heat transfer shell element.

Connectivity

Three nodes per element.

Geometry

Linear thickness variation is allowed in the plane of the element. Internally, the average thickness is used. Thicknesses at the first, second, and third node are stored for each element in the first (EGEOM1), second (EGEOM2), and third (EGEOM3) geometry data field, respectively.

Alternatively, the NODAL THICKNESS model definition option can be used for the input of the element thickness.

If the element needs to be offset from the user-specified position, the eighth data field (EGEOM8) is set to -2.0. In this case, an extra line containing the offset information is provided (see the GEOMETRY option in Marc Volume C: Program Input). The offset magnitudes along the element normal for the three corner nodes are provided in the first, second, and third data fields of the extra line. A uniform offset for all nodes can be set by providing the offset magnitude in the first data field and then setting the constant offset flag (sixth data field of the extra line) to 1.

Coordinates

Three coordinates per node in the global x, y, and z direction.

Degrees of Freedom

The definition of the degrees of freedom depends on the temperature distribution through the thickness and is defined on the HEAT parameter.

The number of degrees of freedom per node is N.

Linear distribution through the thickness (N=2):

1	=	Top Surface Temperature
2	=	Bottom Surface Temperature

Quadratic distribution through the thickness (N=3):

1	=	Top Surface Temperature
2	=	Bottom Surface Temperature
3	=	Mid Surface Temperature

Linear distribution through every layer in Composites with M layers (N=M+1):

1	=	Top Surface Temperature
2	=	Temperature at Interface between Layer 1 and Layer 2
3	=	Temperature at Interface between Layer 2 and Layer 3 etc
N = M+1	=	Bottom Surface Temperature

Quadratic distribution through every layer in Composites with M layers (N=2*M+1):

1	=	Top Surface Temperature
2	=	Temperature at Interface between Layer 1 and Layer 2
3	=	Mid Temperature of Layer 1
4	=	Temperature at Interface between Layer 2 and Layer 3
5	=	Mid Temperature of Layer 2
...		
N-1 = 2*M	=	Bottom Surface Temperature
N = 2*M+1	=	Mid Temperature of Layer N

Piecewise Quadratic distribution through thickness in Non-Composites with M layers (SHELL SECT,M) (N=M+1):

1	=	Top Surface Temperature (thickness position 0.)
X	=	Temperature at thickness position (X-1)/M
N = M+1	=	Bottom Surface Temperature (thickness position 1.

Fluxes

Two types of fluxes:

Volumetric Fluxes

Load Type	Description
1	Uniform flux per unit volume on whole element.
3	Uniform flux per unit volume on whole element; magnitude of flux id defined in the FLUX user subroutine.

Surface Fluxes

Load Type	Description
5	Uniform flux per unit surface area on top surface.
6	Uniform flux per unit surface area on top surface; magnitude of flux is defined in the FLUX user subroutine.
2	Uniform flux per unit surface area on bottom surface.
4	Uniform flux per unit surface area on bottom surface; magnitude of flux is defined in the FLUX user subroutine.

Surface fluxes are evaluated using a 3-point integration scheme, where the integration points have the same location as the nodal points. Point fluxes can also be applied at nodal degrees of freedom.

Films

Same specification as Fluxes .

Tying

Standard tying types 85 and 86 with three-dimensional heat transfer brick elements.

Shell Section – Integration through the thickness

Integration through the shell thickness is performed numerically using Simpson's rule. Use the SHELL SECT parameter to specify the number of integration points. This number must be odd. The default is 11 points.

Output Points

Temperatures are printed out at the centroid of the element through the thickness of the shell. The first point in thickness direction is on the surface of the positive normal.

Joule Heating

Capability is not available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Charge

Same specification as Fluxes .

Element 51

Cable Element

This element is the sag cable element (Figure 5-71). The assumptions for this element are small strain, large displacement and constant strain through the element. This element allows linear elastic behavior only. This element cannot be used with the CONTACT option.

Notes:	All distributed loads are formed on the basis of the current geometry.
	Whenever this element is included in the structure, the distributed load magnitude given in the FORCEM user subroutine must be the total magnitude to be reached at the end of the current increment and not the incremental magnitude.
	Wind load magnitude is based on the unit projected distance and not projected cable length. If there is a big difference between cable length and the distance between the two nodes, it is recommended to subdivide the element along the cable.

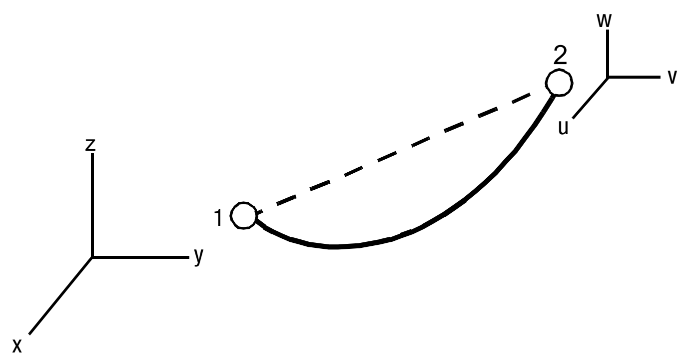


Figure 5-71 Three-dimensional Cable

Quick Reference

Type 51

Three-dimensional, two-node, sag cable.

Connectivity

Two node per element.

Geometry

The cross-section area is entered in the first data field (`EGEOM1`). The cable length is entered in the second data field (`EGEOM2`). The initial length can also be defined as a ratio of the original length on the fourth data field (`EGEOM4`). This is useful when you have a long cable of multiple elements. If the cable length is unknown and the initial stress is known, then enter a zero in the second and fourth data fields and enter the initial stress in the third data field (EGEOM3). If the cable length is equal to the distance between the two nodes, only the first data (cross-sectional area) is required.

Coordinates

Three coordinates per node in the global X, Y, and Z direction.

Degrees of Freedom

Global displacement degree of freedom:

1 = u displacement
2 = v displacement
3 = w displacement

Tractions

Distributed loads according to the value of IBODY are as follows:

Load Type	Description
0	Uniform gravity load (force per unit cable length) in the arbitrary direction.
1	Wind load (force per unit projected distance to the normal plane with respect to wind vector).
2	Arbitrary load (force per unit length); use the FORCEM user subroutine.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Uniaxial in the cable member.

Output of Stresses

Uniaxial in the cable member.

Transformation

The three global degrees of freedom for any node can be transformed to local degrees of freedom.

Tying

Use the UFORMSN user subroutine.

Output Point

Constant value through the element.

Coupled Analysis

There is no heat transfer equivalent of this element; so a coupled thermal-mechanical analysis is not possible.

Note:	This element can neither be used with table driven boundary conditions nor with the AUTO INCREMENT load stepping procedure.

Element 52

Elastic or Inelastic Beam

This is a straight, Euler-Bernoulli beam in space with linear elastic material response as its standard material response, but it also allows nonlinear elastic and inelastic material response. The large displacement formulation only affects the axial strain; large curvature changes are neglected. Linear interpolation is used along the axis of the beam (constant axial force) with cubic displacement normal to the beam axis (linear variation in curvature).

This element can be used to model linear or nonlinear elastic response by direct entry of the cross-section properties. Nonlinear elastic response can be modeled when the material behavior is given in the UBEAM user subroutine (see Marc Volume D: User Subroutines and Special Routines). This element can also be used to model inelastic and nonlinear elastic material response when employing numerical integration over the cross section. Standard cross sections and arbitrary cross sections are entered through the BEAM SECT parameter if the element is to use numerical cross-section integration. Inelastic material response includes plasticity models, creep models, and shape memory models, but excludes powder models, soil models, concrete cracking models, and rigid plastic flow models. Elastic material response includes isotropic elasticity models and NLELAST nonlinear elasticity models, but excludes finite strain elasticity models like Mooney, Ogden, Gent, Arruda-Boyce, Foam and orthotropic or anisotropic elasticity models. With numerical integration, the HYPELA2 user subroutine can be used to model arbitrary nonlinear material response (see Note). Arbitrary sections can be used in a pre-integrated way. In that case, only linear elasticity and nonlinear elasticity through the UBEAM user subroutine are available.

Geometric Basis

The element uses a local (x,y,z) set for section properties. Local x and y are the principal axes of the cross section. Local z is along the beam axis (Figure 5-72). Using fields 4, 5, and 6 of the third data block in the GEOMETRY option, a vector in the plane of the local x-axis and the beam axis must be specified. If no vector is defined here, the local coordinate system can alternatively be defined by the global (x,y,z) coordinates at the two nodes and by (x_1, x_2, x_3), a point in space which locates the local x-axis of the cross section. This axis lies in the plane defined by the beam nodes and this point, pointing from the beam axis toward the point. The local x-axis is normal to the beam axis.

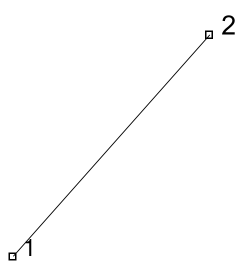


Figure 5-72 Elastic Beam Element

The local z-axis goes from node 1 to node 2, and the local y-axis forms a right-handed set with local x and z.

Quick Reference

Type 52

Elastic straight beam. Linear interpolation axially, cubic normal displacement interpolation.

Connectivity

Two nodes. Local z-axis from first to second node.

Geometry

First geometry data field	– A	– area
Second geometry data field	– I_{xx}	– moment of inertia of section about local x-axis
Third geometry data field	– I_{yy}	– moment of inertia of section about local y-axis

The bending stiffnesses of the section are calculated as EI_{xx} and EI_{yy}. The torsional stiffness of the section is calculated as $\frac{E}{2(1+\upsilon)}(I_{xx} + I_{yy})$. Here E and υ are Young's modulus and Poisson's ratio, calculated as functions of temperature.

If a zero is entered in the first geometry field, Marc uses the beam section data corresponding to the section number given in the second geometry field. (Sections are defined using the BEAM SECT parameter.) This allows specification of the torsional stiffness factor K unequal to $I_{xx} + I_{yy}$ or the specification of arbitrary sections using numerical section integration. See Solid Sections in Chapter 1 for more details.

`EGEOM4-EGEOM6`:	Components of a vector in the plane of the local x-axis and the beam axis. The local x-axis lies on the same side as the specified vector.

If beam-to-beam contact is switched on (see CONTACT option), the radius used when the element comes in contact with other beam or truss elements must be entered in the 7th data field (`EGEOM7`).

For segment-segment beam contact, various flags that control the creation of patches are entered in the seventh data file (`EGEOM7`).

```
EGEOM7 = real(100 * NSEG + 10 * IPTCH + IESCAP)
```

where

`NSEG` = number of segments for solid circular and solid elliptical sections (default = 32)		
`IPTCH` =	Not applicable.	
`IESCAP` = end cap/side cap flag		
	0	No end or side cap (default).
	1	End cap only.
	2	Side cap only.
	3	Average side only.
	4	End and side cap.
	5	End cap and average side.

Beam Offsets Pin Codes and Local Beam Orientation

If an offset is applied to the beam geometry or pin codes are applied to the degrees of freedom of the end point of the beams or the beam axis is defined with respect to a local coordinate system, then a code is placed in the 8th field of the 3rd data block of the GEOMETRY option (`EGEOM8`). This code is formed by the negative sum of ioffset, iorien, and ipin where:

ioffset	=	0	-	no offset
		1	-	beam offset
iorien	=	0	-	conventional definition of local beam orientation; beam axis given in 4th through 6th field with respect to global system
		10	-	the local beam orientation entered in the 4th through 6th field is given with respect to the coordinate system of the first beam node. This vector is transformed to the global coordinate system before any further processing.
ipin	=	0	-	no pin codes are used
		10 0	-	pin codes are used.

If ioffset = 1, an additional line is read in the GEOMETRY option (4th data block).

If ipin = 100, an additional line is read in the GEOMETRY option (5th data block)

The offset vector at the first corner node is provided in the first three fields and the offset vector at the second corner node is provided in the next three fields of the extra line. The coordinate system used for the offset vectors at the two nodes is indicated by the corresponding flags in the eighth and ninth fields of the extra line. A flag setting of 0 indicates global coordinate system, 1 indicates local element coordinate system, 2 indicates local shell normal, and 3 indicates local nodal coordinate system. When the flag setting is 1, the local beam coordinate system is constructed by taking the local z axis along the beam, the local x axis as the user-specified vector (obtained from `EGEOM4-EGEOM6`) and the local y axis as perpendicular to both. The flag setting of 2 can be used when the beam nodes are also attached to a shell. In this case, the offset magnitude at the first and second nodes are specified in the first and fourth data fields of the extra line respectively. The automatically calculated shell normals at the nodes are used to construct the offset vectors.

Coordinates

First three coordinates - (x, y, z) global Cartesian coordinates.

Fourth, fifth, and sixth coordinates at each node – global Cartesian coordinates of a point in space which locates the local x-axis of the cross section: this axis lies in the plane defined by the beam nodes and this point, pointing from the beam towards this point. The local x-axis is normal to the beam axis. The fourth, fifth and sixth coordinates are only used if the local x-axis direction is not specified in the GEOMETRY option.

Degrees of Freedom

1 = u_x = global Cartesian x-direction displacement
2 = u_y = global Cartesian y-direction displacement
3 = u_z = global Cartesian z-direction displacement
4 = ϕ_x = rotation about global x-direction
5 = ϕ_y = rotation about global y-direction
6 = ϕ_z = rotation about global z-direction

Tractions

The four types of distributed loading are as follows:

Load Type	Description
1	Uniform load per unit length in global x-direction.
2	Uniform load per unit length in global y-direction.
3	Uniform load per unit length in global z-direction.
4	Nonuniform load per unit length; magnitude and direction supplied via the FORCEM user subroutine.
11	Fluid drag/buoyancy loading – fluid behavior specified in the FLUID DRAG option.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Point loads and moments can be applied at the nodes.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Generalized strain components are as follows:

Axial stretch ε
Curvature about local x-axis of cross section K_{xx}
Curvature about local y-axis of cross section K_{yy}
Twist about local z-axis of cross section K_{zz}

Output of Stresses

Generalized stresses:

Axial force
Bending moment about x-axis of cross section
Bending moment about y-axis of cross section
Torque about beam axis

Layer stresses:

Layer stresses in the cross section are only available if the element uses numerical cross-section integration (and the section is not pre-integrated) and are only printed if explicitly requested or if plasticity is present.

1 = σ_{zz}

2 = τ_{zx}

3 = τ_{zy}

Transformation

Displacements and rotations can be transformed to local directions.

Tying

For interacting beams use tying type 100 for fully moment carrying joint, tying type 52 for pinned joint.

Output Points

Centroidal section or three Gauss integration sections.

For all beam elements, the default printout gives section forces and moments.

Updated Lagrange Procedure and Finite Strain Plasticity

The Updated Lagrange procedure is available for this element, and should be used if the element is subjected to large rotations. Since the cross-sectional dimensions of the element are not updated, the element should not be used for finite strain applications.

Note:	Nonlinear elasticity without numerical cross-section integration can be implemented with the HYPOELASTIC option and the UBEAM user subroutine. In this case, the user enters a relationship between the generalized stresses and generalized strains. Arbitrary nonlinear material behavior with numerical cross-section integration can be implemented with the HYPOELASTIC option and the HYPELA2 user subroutine. However, the deformation gradient, the rotation tensor, and the stretch ratios are not available in the subroutine. It is also possible to use the NLELAST option with this element.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 99. See Element 99 for a description of the convention used for entering the flux and film data for this element.

Design Variables

The cross-sectional area (A) and moments of inertia (I_{xx}, I_{yy}) can be considered as design variables when the cross-section does not employ numerical section integration.

Element 53 — Plane Stress, Eight-node Distorted Quadrilateral with Reduced Integration

Element type 53 is an eight-node, isoparametric, arbitrary quadrilateral written for plane stress applications using reduced integration. This element uses biquadratic interpolation functions to represent the coordinates and displacements. Hence, the strains have a linear variation. This allows for accurate representation of the strain fields in elastic analyses.

Lower-order elements, such as type 3, are preferred in contact analyses.

The stiffness of this element is formed using four-point Gaussian integration. This is a reduced integration element – which may exhibit hourglass modes. This element should be used with caution. The mass matrix of this element is formed using nine-point Gaussian integration.

All constitutive models can be used with this element.

Quick Reference

Type 53

Second-order, isoparametric, distorted quadrilateral with reduced integration. Plane stress.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention). Then fifth node between first and second; the sixth node between second and third, etc. See Figure 5-73.

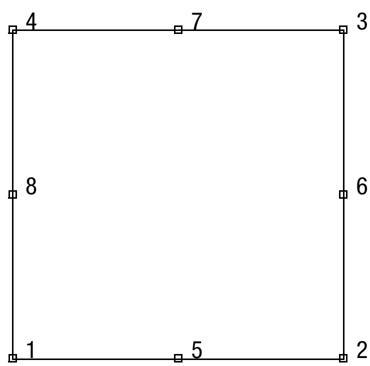


Figure 5-73 Nodes of Eight-node, 2-D Element

Geometry

Thickness stored in first data field (EGEOM1). Default thickness is unity. Other fields not used.

Coordinates

Two global coordinates, x and y, at each node.

Degrees of Freedom

Two at each node:

1 = u = global x-direction displacement.
2 = v = global y-direction displacement.

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type (IBODY)	Description
* 0	Uniform pressure on 1-5-2 face.
* 1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.
4	Uniform body force in y-direction.
5	Nonuniform body force in the y-direction.
* 6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
* 7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
* 8	Uniform pressure on 2-6-3 face.
* 9	Nonuniform pressure on 2-6-3 face.
* 10	Uniform pressure on 3-7-4 face.
* 11	Nonuniform pressure on 3-7-4 face.
* 12	Uniform pressure on 4-8-1 face.
* 13	Nonuniform pressure on 4-8-1 face.
* 20	Uniform shear force on 1-2-5 face in the 1⇒2⇒5 direction.
* 21	Nonuniform shear force on side 1-5-2.
* 22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
* 23	Nonuniform shear force on side 2-6-3.
* 24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
* 25	Nonuniform shear force on side 3-7-4.
* 26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
* 27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Load Type (IBODY)	Description
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine. Load types shown with an asterisk (*) require the magnitude of the load to be entered as force per unit area. To prescribe these loads in force per unit length, add 50 to the load type. This is often useful in design optimization where the thickness changes, but it is desired that the applied force remain the same.

A nine-point integration scheme is used for the integration of body forces (see Figure 5-74).

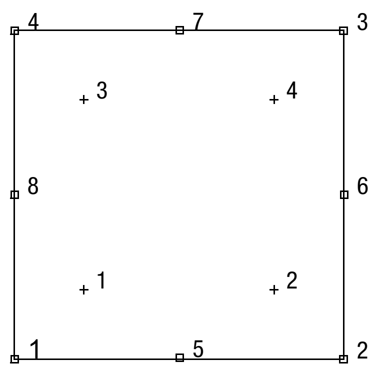


Figure 5-74 Integration Points of Eight-node, 2-D Element with Reduced Integration

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid of element or four integration points (see Figure 5-74 and Output Points below) in the following order:

1 = ε_{xx}, direct

2 = ε_{yy}, direct

3 = γ_{xy}, shear

Note:	Although $\varepsilon_{zz} = \frac{-\nu}{E}(\sigma_{xx} + \sigma_{yy})$, it is not printed and is posted as for isotropic materials. For Mooney or Ogden (TL formulation) Marc post code 49 provides the thickness strain for plane stress elements. See Marc Volume A: Theory and User Information , Chapter 12 Output Results, Element Information for von Mises intensity calculation for strain.

Output of Stresses

Output of stresses is the same as Output of Strains .

Transformation

Only in x-y plane.

Tying

Use the UFORMSN user subroutine.

Output Points

If the CENTROID parameter is used, the output occurs at the centroid of the element.

If the ALL POINTS parameter is used, four output points are given, as shown in Figure 5-74. This is the usual option for a second-order element with reduced integration.

Note:	Because this is a reduced element, it is possible to excite so-called “hourglass” or “breathing” modes. This mode, shown in Figure 5-75, makes no contribution to the strain energy of the element.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinates directions. Thickness is updated.

Note:	Distortion of element during analysis can cause bad results. Element type 3 is to be preferred.

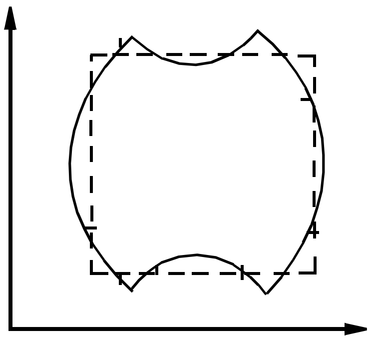


Figure 5-75 Breathing Mode of Reduced Integration

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 69. See Element 69 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered as a design variable for this element.

Element 54 Plane Strain, Eight-node Distorted Quadrilateral with Reduced Integration

Element type 54 is an eight-node, isoparametric, arbitrary quadrilateral written for plane strain applications using reduced integration. This element uses biquadratic interpolation functions to represent the coordinates and displacements. Hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses.

Lower-order elements, such as type 11, are preferred in contact analyses.

The stiffness of this element is formed using four-point Gaussian integration. This is a reduced integration element – which may exhibit hourglass modes. This element should be used with caution. The mass matrix of this element is formed using nine-point Gaussian integration.

This element can be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 58 instead. Element type 58 is also preferable for small strain incompressible elasticity.

Quick Reference

Type 54

Second-order, isoparametric, distorted quadrilateral with reduced integration. Plane strain.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention). Then the fifth node between first and second; the sixth node between second and third, etc. See Figure 5-76.

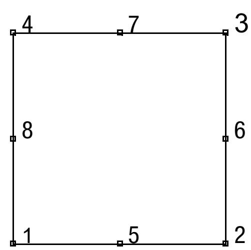


Figure 5-76 Eight-node, Plane Strain Element

Geometry

Thickness stored in first data field (`EGEOM1`). Default thickness is unity. Other fields are not used.

Coordinates

Two global coordinates, x and y, at each node.

Degrees Of Freedom

Two at each node:

1 = u = global x-direction displacement
2 = v = global y-direction displacement

Tractions

Surface Forces. Pressure and shear surface forces available for this element are as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.
4	Uniform body force in y-direction.
5	Nonuniform body force in the y-direction.
6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
8	Uniform pressure on 2-6-3 face.
9	Nonuniform pressure on 2-6-3 face.
10	Uniform pressure on 3-7-4 face.
11	Nonuniform pressure on 3-7-4 face.
12	Uniform pressure on 4-8-1 face.
13	Nonuniform pressure on 4-8-1 face.
20	Uniform shear force on 1-2-5 face in the 1⇒2⇒5 direction.
21	Nonuniform shear force on side 1-5-2.
22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
23	Nonuniform shear force on side 2-6-3.
24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
25	Nonuniform shear force on side 3-7-4.
26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Load Type (IBODY)	Description
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

A nine-point integration scheme is used for the integration of body forces (see Figure 5-77).

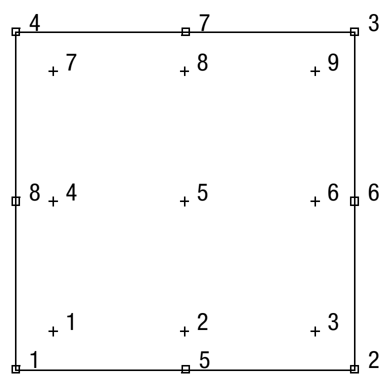


Figure 5-77 Integration Points for Body Forces Calculation

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid or element integration points (see Figure 5-78 and Output Points on the following page) in the following order:

1 = ε_{xx}, direct

2 = ε_{yy}, direct

3 = ε_{zz}, thickness direction, direct

4 = γ_{xy}, shear

Output of Stresses

Output of stresses is the same as Output of Strains .

Transformation

Only in the x-y plane.

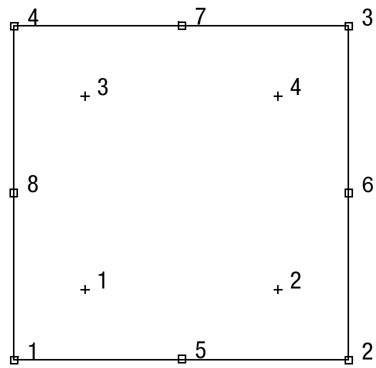


Figure 5-78 Integration Points for Reduced Integration Planar Element

Tying

Use the UFORMSN user subroutine.

Output Points

If the CENTROID parameter is used, the output occurs at the centroid of the element.

If the ALL POINTS parameter is used, four output points are given, as shown in Figure 5-78. This is the usual option for a second-order element with reduced integration.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinate directions.

Note:	Distortion or element during analysis can cause bad results. Element type 6 or 11 is preferred.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 69. See Element 69 for a description of the conventions used for entering the flux and film data for this element.

Element 55 Axisymmetric, Eight-node Distorted Quadrilateral with Reduced Integration

Element type 55 is an eight-node, isoparametric, arbitrary quadrilateral written for axisymmetric applications using reduced integration. This element uses biquadratic interpolation functions to represent the coordinates and displacements, hence the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses.

Lower-order elements, such as type 10, are preferred in contact analyses.

The stiffness of this element is formed using four-point Gaussian integration. This is a reduced integration element – which may exhibit hourglass modes. This element should be used with caution. The mass matrix of this element is formed using nine-point Gaussian integration.

This element can be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 59 instead. Element type 59 is also preferable for small strain incompressible elasticity.

Quick Reference

Type 55

Second-order, isoparametric, distorted quadrilateral with reduced integration. Axisymmetric formulation.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention in z-r plane). Then the fifth node between first and second; the sixth between second and third, etc. (see Figure 5-79).

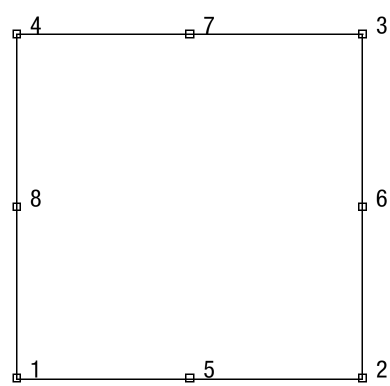


Figure 5-79 Nodes of Eight-node Axisymmetric Element with Reduced Integration

Geometry

No geometry in input for this element.

Coordinates

Two global coordinates, z and r, at each node.

Degrees of Freedom

Two at each node:

1 = u = global z-direction displacement (axial)
2 = v = global r-direction displacement (radial)

Tractions

Surface Forces. Pressure and shear forces available for this element are as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.
4	Uniform body force in y-direction.
5	Nonuniform body force in the y-direction.
6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
8	Uniform pressure on 2-6-3 face.
9	Nonuniform pressure on 2-6-3 face.
10	Uniform pressure on 3-7-4 face.
11	Nonuniform pressure on 3-7-4 face.
12	Uniform pressure on 4-8-1 face.
13	Nonuniform pressure on 4-8-1 face.
20	Uniform shear force on 1-2-5 face in the 1⇒2⇒5 direction.
21	Nonuniform shear force on side 1-5-2.
22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
23	Nonuniform shear force on side 2-6-3.
24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
25	Nonuniform shear force on side 3-7-4.
26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
27	Nonuniform shear force on side 4-8-1.

Load Type (IBODY)	Description
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

A nine-point integration scheme is used for the integration of body forces (see Figure 5-80).

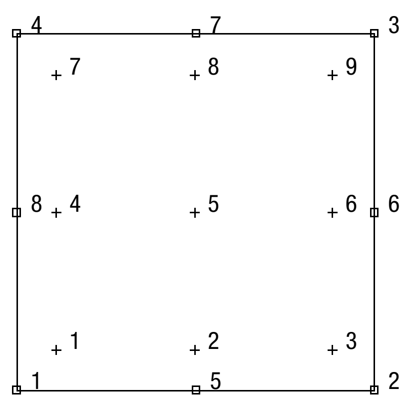


Figure 5-80 Integration Points for Body Force Calculation

Concentrated nodal loads must be the value of the load integrated around the circumference.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid or element integration points (see Output Points on the following page and Figure 5-81) in the following order:

1 = ε_{zz}, direct

2 = ε_{rr}, direct

3 = $\varepsilon_{\theta\theta}$, hoop direct

4 = γ_{zr}, shear in the section

Output of Stresses

Output of stresses is the same as Output of Strains .

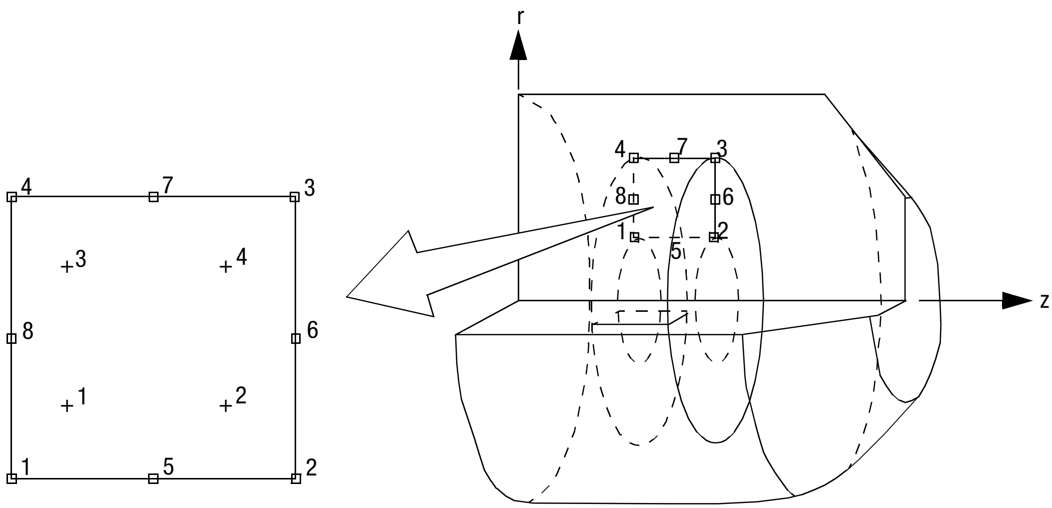


Figure 5-81 Integration Points of EIght-node, Axisymmetric Element with Reduced Integration

Transformation

Only in z-r plane.

Tying

Use the UFORMSN user subroutine.

Output Points

If the CENTROID parameter is used, the output occurs at the centroid of the element.

If the ALL POINTS parameter is used, four output points are given, as shown in Figure 5-81. This is the usual option for a second-order element with reduced integration.

Updating Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinate directions.

Note: Distortion or element during analysis can cause bad results. Element type 2 or 10 is preferred.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 70. See Element 70 for a description of the conventions used for entering the flux and film data for this element.

Element 56 Generalized Plane Strain, Distorted Quadrilateral with Reduced Integration

This element is an extension of the plane strain isoparametric quadrilateral (element type 54) to the generalized plane strain case. The generalized plane strain condition is obtained by allowing two extra nodes in each element. One of these nodes has the single degree of freedom of change of distance between the top and bottom of the structure at one point (that is, change in thickness at that point) while the other additional node contains the relative rotations θ_{xx} and θ_{yy} of the top surface with respect to the bottom surface about the same point. The generalized plane strain condition is attained by allowing these two nodes to be shared by all elements forming the structure. Note that this is an extension of the usual generalized plane strain condition which is obtained by setting the relative rotations to zero (by kinematic boundary conditions) and from there the element could become a plane strain element if the relative displacement is also made zero. Note that the sharing of the two extra nodes by all elements implies two full nodal rows in the generated plane strain part of the assembled stiffness matrix considerable computational savings are achieved if these two nodes are given the highest node numbers in that part of the structure.

This element uses biquadratic interpolation functions to represent the coordinates and displacements. Hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses.

Lower-order elements, such as type 19, are preferred in contact analyses.

The stiffness of this element is formed using four-point Gaussian integration. This is a reduced integration element – which may exhibit hourglass modes. This element should be used with caution. The mass matrix of this element is formed using nine-point Gaussian integration.

This element can be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 60 instead. Element type 60 is also preferable for small strain incompressible elasticity.

This element cannot be used with the CASI iterative solver.

Quick Reference

Type 56

Second-order, isoparametric, distorted quadrilateral with reduced integration. Generalized plane strain formulation.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention in x-y plane). Then the fifth node between first and second; the sixth node between second and third, etc. The ninth and tenth nodes are the generalized plane strain nodes shared with all elements in the mesh.

Geometry

Thickness stored in first data field (`EGEOM1`). Default thickness is unity. Other fields not used.

Coordinates

Two global coordinates, x and y, at each of the ten nodes. Note that the ninth and tenth nodes can be anywhere in the (x, y) plane.

Degrees of Freedom

Two at each of the first eight nodes:

1 = u = global x-direction displacement
2 = v = global y-direction displacement

One at the ninth node:

1 = Δz = relative z-direction displacement of front and back surfaces. See Figure 5-82.

Two at the tenth node:

1 = $\Delta\theta_x$ = Relative rotation of front and back surfaces about global x-axis.

2 = $\Delta\theta_y$ = Relative rotation of front and back surfaces about global y-axis. See Figure 5-82.

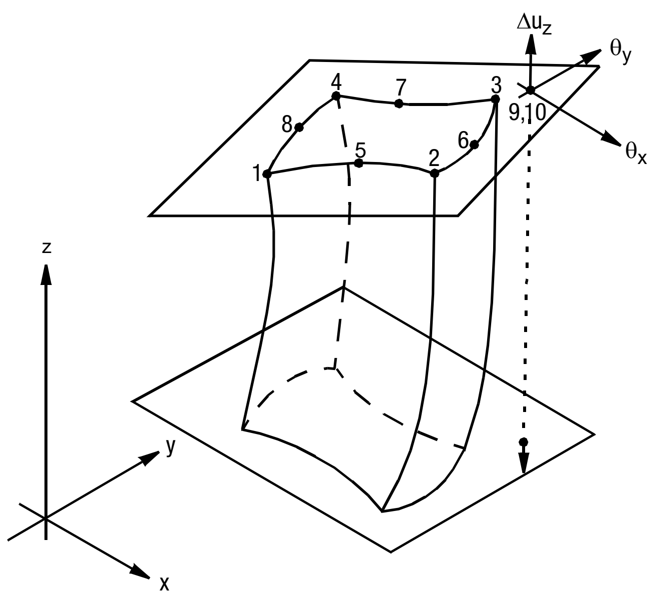


Figure 5-82 Generalized Plane Strain Distorted Quadrilateral with Reduced Integration

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type (IBODY)	Description
* 0	Uniform pressure on 1-5-2 face.
* 1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.
4	Uniform body force in y-direction.

Load Type (IBODY)	Description
5	Nonuniform body force in the y-direction.
* 6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
* 7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
* 8	Uniform pressure on 2-6-3 face.
* 9	Nonuniform pressure on 2-6-3 face.
* 10	Uniform pressure on 3-7-4 face.
* 11	Nonuniform pressure on 3-7-4 face.
* 12	Uniform pressure on 4-8-1 face.
* 13	Nonuniform pressure on 4-8-1 face.
* 20	Uniform shear force on 1-2-5 face in the 1⇒2⇒5 direction.
* 21	Nonuniform shear force on side 1-5-2.
* 22	Uniform shear force on side 2-6-3 in the 2⇒6⇒ 3 direction.
* 23	Nonuniform shear force on side 2-6-3.
* 24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
* 25	Nonuniform shear force on side 3-7-4.
* 26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
* 27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine. Load types shown with an asterisk (*) require the magnitude of the load to be entered as force per unit area. To prescribe these loads in force per unit length, add 50 to the load type. This is often useful in design optimization where the thickness changes, but it is desired that the applied force remain the same.

A nine-point integration scheme is used for the integration of body forces (Figure 5-83).

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid or element integration points (see Output Points on the following page and Figure 5-83) in the following order:

1 = ε_{xx}, direct

2 = ε_{yy}, direct

3 = ε_{zz}, thickness direction direct

4 = γ_{xy}, shear in the (x-y) plane

No γ_{xz}, γ_{yz} or shear – relative rotations of front and back surfaces.

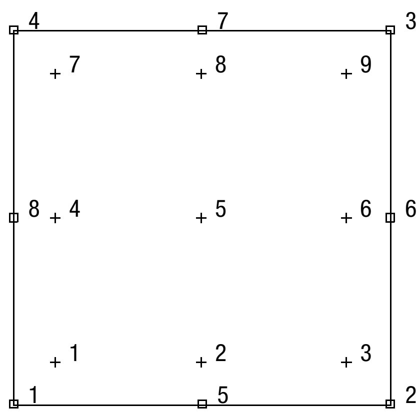


Figure 5-83 Integration Points for Body Force Calculation

Output of Stresses

Output of stresses is the same as Output of Strains .

Transformation

Only in x-y plane.

Tying

Use the UFORMSN user subroutine.

Output Points

If the CENTROID parameter is used, the output occurs at the centroid of the element.

If the ALL POINTS parameter is used, four output points are given as shown in Figure 5-84. This is the usual option for a second-order element with reduced integration.

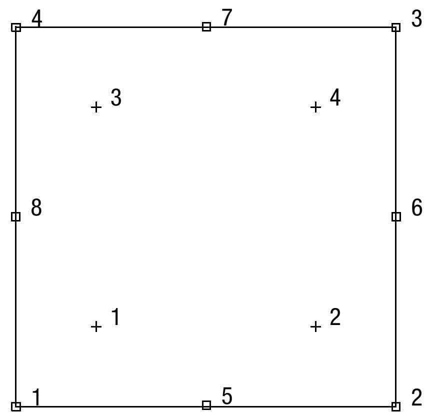


Figure 5-84 Integration Points for Stiffness Matrix

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinates. Thickness is updated.

Note:	Distortion or element during analysis can cause bad results. Element type 19 is preferred.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 69. See Element 69 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered as a design variable for this element.

Element 57

Three-dimensional 20-node Brick with Reduced Integration

Element type 57 is a 20-node, isoparametric, arbitrary hexahedral using reduced integration.This element uses triquadratic interpolation functions to represent the coordinates and displacements. Hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses.

Lower-order elements, such as type 7, are preferred in contact analyses.

The stiffness of this element is formed using eight-point Gaussian integration. This is a reduced integration element – which may exhibit hourglass modes. This element should be used with caution. The mass matrix of this element is formed using twenty-seven-point Gaussian integration.

This element can be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 61 instead. Element type 61 is also preferable for small strain incompressible elasticity.

Note:	Reduction to Wedge or Tetrahedron – By simply repeating node numbers on the same spatial position, the element can be reduced as far as a tetrahedron. Element type 127 is preferred for tetrahedrals

Geometry

The geometry of the element is interpolated from the Cartesian coordinates of 20 nodes.

Connectivity

The convention for the ordering of the connectivity array is as follows:

> Nodes 1, 2, 3, 4 are the corners of one face, given in counterclockwise order when viewed from inside the element. Nodes 5, 6, 7, 8 are corners of the opposite face. Node 5 shares an edge with 1; node 6 shares an edge with 2, etc. Nodes 9, 11, and 12 are the middle of the edges of the 1, 2, 3, 4 face; node 9 between 1 and 2; node 10 between 2 and 3, etc. Similarly nodes 13, 14, 15, 16 are midpoints on the 5, 6, 7, 8 face; node 13 between 5 and 6, etc. Finally, node 17 is the midpoint of the 1-5 edge; node 18 of the 2-6 edge, etc.

Note that in most normal cases, the elements are automatically generated, so that you need not concern yourself with the node numbering scheme.

Integration

The element is integrated numerically using eight points (Gaussian quadrature). The first plane of such points is closest to the 1, 2, 3, 4 face of the element, with the first point closest to the first node of the element (see Figure 5-85). A similar plane follows, moving toward the 5, 6, 7, 8 face. If the CENTROID parameter is used, the output occurs at the centroid of the element. If the ALL POINTS parameter is used, the integration points are used for stress output. That is the usual option for a second-order element with reduced integration.

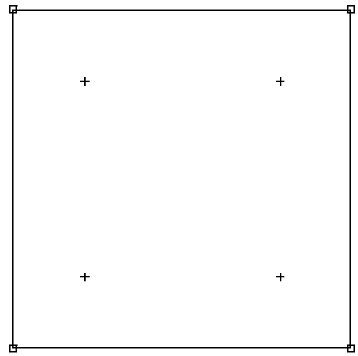


Figure 5-85 Element 57 Integration Plane

The FORCEM user subroutine is called once per integration point when flagged. The magnitude of load defined by DIST LOADS is ignored and the FORCEM value is used instead. For nonuniform body force, force values must be provided for 27 integration points, as specified in Figure 5-86 since the reduced integration scheme is not used for the body forces. For nonuniform surface pressures, values need only be supplied for the nine integration points on the face of application.

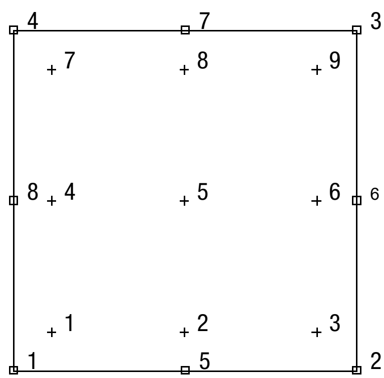


Figure 5-86 Integration Points for Distributed Loads

Nodal (concentrated) loads can also be supplied.

Quick Reference

Type 57

Twenty-nodes, isoparametric arbitrary distorted cube.

Connectivity

Twenty nodes numbered as described in the connectivity write-up for this element and is shown in Figure 5-87.

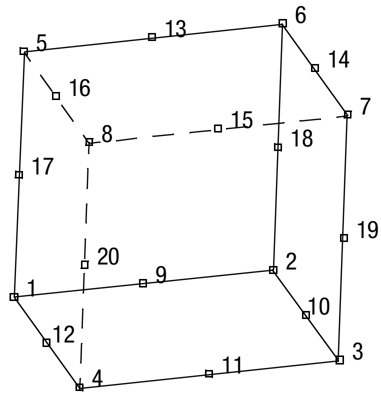


Figure 5-87 Form of Element 57

Geometry

Generally not required. The first field contains the transition thickness if the automatic brick to shell transition constraints are to be used (see Figure 5-88).

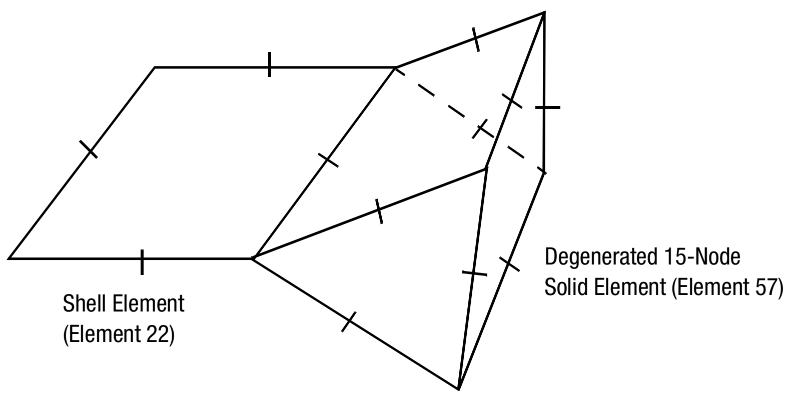


Figure 5-88 Shell-to-solid Automatic Constraint

Coordinates

Three global coordinates in the x, y and z directions.

Degrees of Freedom

Three global degrees of freedom: u, v and w.

Distributed Loads

Distributed loads chosen by value of IBODY are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3-4 face.
1	Nonuniform pressure on 1-2-3-4 face; magnitude supplied through the FORCEM user subroutine.
2	Uniform body force per unit volume in -z direction.
3	Nonuniform body force per unit volume (e.g., centrifugal force); magnitude and direction supplied through the FORCEM user subroutine.
4	Uniform pressure on 6-5-8-7 face.
5	Nonuniform pressure on 6-5-8-7 face.
6	Uniform pressure on 2-1-5-6 face.
7	Nonuniform pressure on 2-1-5-6 face.
8	Uniform pressure on 3-2-6-7 face.
9	Nonuniform pressure on 3-2-6-7 face.
10	Uniform pressure on 4-3-7-8 face.
11	Nonuniform pressure on 4-3-7-8 face.
12	Uniform pressure on 1-4-8-5 face.
13	Nonuniform pressure on 1-4-8-5 face.
20	Uniform pressure on 1-2-3-4 face.
21	Nonuniform load on 1-2-3-4 face; magnitude and direction supplied in the FORCEM user subroutine.
22	Uniform body force per unit volume in -z direction.
23	Nonuniform body force per unit volume (e.g., centrifugal force; magnitude supplied through the FORCEM user subroutine).
24	Uniform pressure on 6-5-8-7 face.
25	Nonuniform load on 6-5-8-7 face; magnitude and direction supplied in the FORCEM user subroutine.
26	Uniform pressure on 2-1-5-6 face.
27	Nonuniform load on 2-1-5-6 face; magnitude and direction supplied in the FORCEM user subroutine.
28	Uniform pressure on 3-2-6-7 face.
29	Nonuniform load on 3-2-6-7 face; magnitude and direction supplied in the FORCEM user subroutine.
30	Uniform pressure on 4-3-7-8 face.

Load Type	Description
31	Nonuniform load on 4-3-7-8 face; magnitude and direction supplied in the FORCEM user subroutine.
32	Uniform pressure on 1-4-8-5 face.
33	Nonuniform load on 1-4-8-5 face; magnitude and direction supplied in the FORCEM user subroutine.
40	Uniform shear 1-2-3-4 face in 12 direction.
41	Nonuniform shear 1-2-3-4 face in 12 direction.
42	Uniform shear 1-2-3-4 face in 23 direction.
43	Nonuniform shear 1-2-3-4 face in 23 direction.
48	Uniform shear 6-5-8-7 face in 56 direction.
49	Nonuniform shear 6-5-8-7 face in 56 direction.
50	Uniform shear 6-5-8-7 face in 67 direction.
51	Nonuniform shear 6-5-8-7 face in 67 direction.
52	Uniform shear 2-1-5-6 face in 12 direction.
53	Nonuniform shear 2-1-5-6 face in 12 direction.
54	Uniform shear 2-1-5-6 face in 15 direction.
55	Nonuniform shear 2-1-5-6 face in 15 direction.
56	Uniform shear 3-2-6-7 face in 23 direction.
57	Nonuniform shear 3-2-6-7 face in 23 direction.
58	Uniform shear 3-2-6-7 face in 26 direction.
59	Nonuniform shear 2-3-6-7 face in 26 direction.
60	Uniform shear 4-3-7-8 face in 34 direction.
61	Nonuniform shear 4-3-7-8 face in 34 direction.
62	Uniform shear 4-3-7-8 face in 37 direction.
63	Nonuniform shear 4-3-7-8 face in 37 direction.
64	Uniform shear 1-4-8-5 face in 41 direction.
65	Nonuniform shear 1-4-8-5 face in 41 direction.
66	Uniform shear 1-4-8-5 face in 15 direction.
67	Nonuniform shear 1-4-8-5 face in 15 direction.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Load Type	Description
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

1 ε_{xx}
2 ε_{yy}
3 ε_{zz}
4 ε_{xy}
5 ε_{yz}
6 ε_{zx}

Output of Stresses

Same as for Output of Strains .

Transformation

Three global degrees of freedom can be transformed to local degrees of freedom.

Tying

Use the UFORMSN user subroutine. An automatic constraint is available for brick to shell transition meshes. (See Geometry .)

Output Points

Centroid or eight Gaussian integration points (see Figure 5-89).

Note:	A large bandwidth results in long run times. Optimize as much as possible.

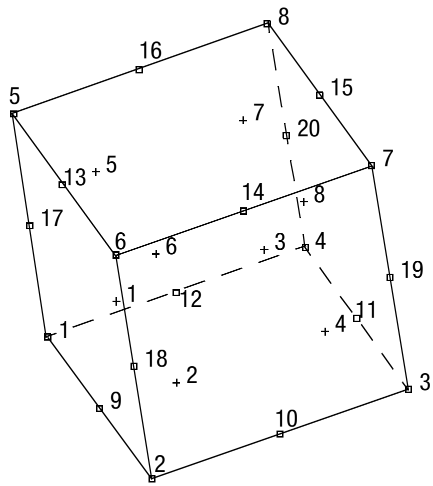


Figure 5-89 Integration Points for 20-node Brick with Reduced Integration

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinates.

Note:	Distortion of element during analysis can cause bad results. Element type 7 is preferred.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 71. See Element 71 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux generated by dissipated plastic work is specified with type 101.

Element 58

Plane Strain Eight-node Distorted Quadrilateral with Reduced Integration Herrmann Formulation

Element type 58 is an eight-node, isoparametric, arbitrary quadrilateral written for incompressible plane strain applications using reduced integration. This element uses biquadratic interpolation functions to represent the coordinates and displacements. Hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is represented using bilinear interpolation functions based upon the extra degree of freedom at the corner nodes.

Lower-order elements, such as type 80, are preferred in contact analyses.

The stiffness of this element is formed using four-point Gaussian integration. This is a reduced integration element – which may exhibit hourglass modes. This element should be used with caution. The mass matrix of this element is formed using nine-point Gaussian integration.

This element is designed to be used for incompressible elasticity only. It can be used for either small strain behavior or large strain behavior using the Mooney or Ogden models.This element can be used in conjunction with other elements such as type 11 when other material behavior, such as plasticity, must be represented.

Quick Reference

Type 58

Second-order, isoparametric, distorted quadrilateral with reduced integration. Plane strain. Hybrid formulation for incompressible and nearly incompressible elastic materials.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention). Then the fifth node between first and second; the sixth node between second and third, etc. See Figure 5-90.

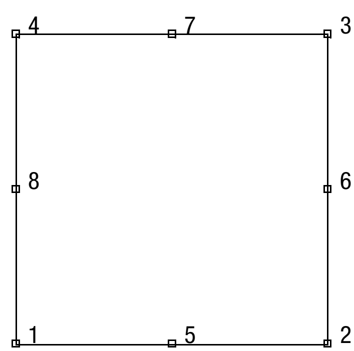


Figure 5-90 Eight-node, Planar Element

Geometry

Thickness stored in first data field (`EGEOM1`). Default thickness is unity. Other fields are not used.

Coordinates

Two global coordinates, x and y, at each node.

Degrees of Freedom

1 = u = global x-direction displacement

2 = v = global y-direction displacement; additional degree of freedom at corner nodes only

Additional degree of freedom at corner nodes only.

3	= σ_{kk}/E	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1+\frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2+\frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3\right]\frac{9}{E_1+E_2+E_3}$$	
	= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)
	= p / K	(for Herrmann elements using additive decomposition; K here is the effective bulk modulus)

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.
4	Uniform body force in y-direction.
5	Nonuniform body force in the y-direction.
6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
8	Uniform pressure on 2-6-3 face.
9	Nonuniform pressure on 2-6-3 face.
10	Uniform pressure on 3-7-4 face.
11	Nonuniform pressure on 3-7-4 face.
12	Uniform pressure on 4-8-1 face.
13	Nonuniform pressure on 4-8-1 face.
20	Uniform shear force on 1-2-5 face in the 1⇒2⇒5 direction.

Load Type	Description
21	Nonuniform shear force on side 1-5-2.
22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
23	Nonuniform shear force on side 2-6-3.
24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
25	Nonuniform shear force on side 3-7-4.
26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

Note that a nine-point scheme is used for the integration or body forces.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid or element integration points (see Figure 5-91 and Output Points) in the following order:

1	= ε_{xx}, direct
2	= ε_{yy}, direct
3	= ε_{zz}, thickness direction, direct
4	= γ_{xy} shear

5	$= \sigma_{kk}/E$	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1+\frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2+\frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3\right]\frac{9}{E_1+E_2+E_3}$$	
	= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)

The last component is not printed in the output file, but is accessible via the ELEVAR and PLOTV user subroutines.

Output of Stresses

Four stresses corresponding to first four Output of Strains .

Transformation

Only in x-y plane.

Tying

Use the UFORMSN user subroutine.

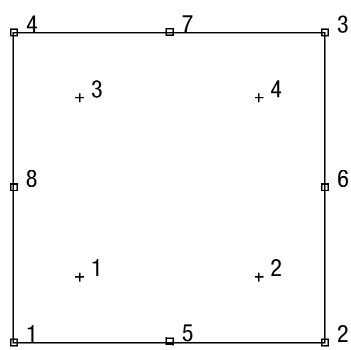


Figure 5-91 Integration Points for Eight-node Reduced Integration Element

Output Points

If the CENTROID parameter is used, the output occurs at the centroid of the element.

If the ALL POINTS parameter is used, four output points are given, as shown in Figure 5-91. This is the usual option for a second-order element with reduced integration.

Special Considerations

The mean pressure degree of freedom must be allowed to be discontinuous across changes in material properties. Use tying for this case.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available. Finite elastic strains can be calculated with the MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 69. See Element 69 for a description of the conventions used for entering the flux and film data for this element.

Element 59 Axisymmetric, Eight-node Distorted Quadrilateral with Reduced Integration, Herrmann Formulation

Element type 59 is an eight-node, isoparametric, arbitrary quadrilateral written for incompressible axisymmetric applications using reduced integration. This element uses biquadratic interpolation functions to represent the coordinates and displacements. Hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is represented using bilinear interpolation functions based upon the extra degree of freedom at the corner nodes.

Lower-order elements, such as type 82, are preferred in contact analyses.

The stiffness of this element is formed using four-point Gaussian integration. This is a reduced integration element – which may exhibit hourglass modes. This element should be used with caution. The mass matrix of this element is formed using nine-point Gaussian integration.

This element is designed to be used for incompressible elasticity only. It can be used for either small strain behavior or large strain behavior using the Mooney or Ogden models.This element can be used in conjunction with other elements such as type 55 when other material behavior, such as plasticity, must be represented.

Quick Reference

Type 59

Second-order, isoparametric, distorted quadrilateral with reduced integration axisymmetric formulation. Hybrid formulation for incompressible or nearly incompressible materials.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention in z-r plane). Then the fifth node between first and second; the sixth node between second and third, etc. See Figure 5-92.

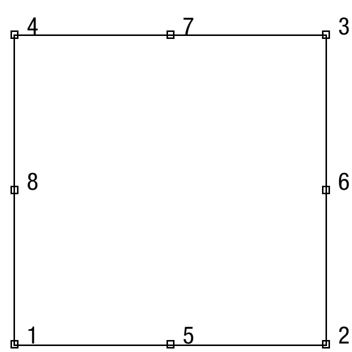


Figure 5-92 Eight-node Axisymmetric Element

Geometry

No geometry input is necessary for this element.

Coordinates

Two global coordinates, z and r, at each node.

Degrees of Freedom

1 = u = global z-direction displacement (axial)
2 = v = global r-direction displacement (radial)

Additional degree of freedom at each corner node:

3 = σ_{kk}/E	= mean pressure variable (isotropic)
= formula below (orthotropic)	

$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}} \sigma_1 + \frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}} \sigma_2 + \frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}} \sigma_3 \right] \frac{9}{E_1 + E_2 + E_3}$$

= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)
= p / K	(for Herrmann elements using additive decomposition; K here is the effective bulk modulus)

Tractions

Surface Forces. Pressure and shear surface forces available for this element are as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.
4	Uniform body force in y-direction.
5	Nonuniform body force in the y-direction.
6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
8	Uniform pressure on 2-6-3 face.
9	Nonuniform pressure on 2-6-3 face.
10	Uniform pressure on 3-7-4 face.
11	Nonuniform pressure on 3-7-4 face.
12	Uniform pressure on 4-8-1 face.
13	Nonuniform pressure on 4-8-1 face.
20	Uniform shear force on 1-2-5 face in the 1⇒2⇒5 direction.

Load Type (IBODY)	Description
21	Nonuniform shear force on side 1-5-2.
22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
23	Nonuniform shear force on side 2-6-3.
24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
25	Nonuniform shear force on side 3-7-4.
26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

Note that a nine-point scheme is used for integration of body forces.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid of element integration points (see Figure 5-93 and Output Points) in the following order:

1	= ε_{zz}, direct	
2	= ε_{rr}, direct	
3	= $\varepsilon_{\theta\theta}$, hoop direct	
4	= γ_{zr} shear in the section	
5	= σ_{kk}/E	= mean pressure variable (isotropic)

= formula below (orthotropic)

$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1 + \frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2 + \frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3 \right] \frac{9}{E_1 + E_2 + E_3}$$

The last component is not printed in the output file, but is accessible via the ELEVAR and PLOTV user subroutines.

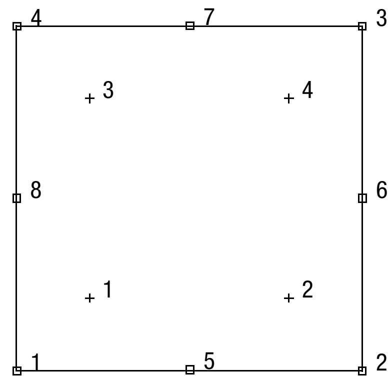


Figure 5-93 Integration Points for Eight-node Reduced Integration Element

Output of Stresses

Output of stresses is the same as first four Output of Strains .

Transformation

Only in z-r plane.

Tying

Use the UFORMSN user subroutine.

Output Points

If the CENTROID parameter is used, the output occurs at the centroid of the element.

If the ALL POINTS parameter is used, four output points are given as shown in Figure 5-93. This is the usual option for a second-order element with reduced integration.

Special Considerations

The mean pressure degree of freedom must be allowed to be discontinuous across changes in material properties. Use tying for this case.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available. Finite elastic strains can be calculated with the MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 70. See Element 70 for a description of the conventions used for entering the flux and film data for this element.

Element 60

Generalized Plane Strain Distorted Quadrilateral with Reduced Integration, Herrmann Formulation

This element is an extension of the plane strain isoparametric quadrilateral (element type 58) to the generalized plane strain case. The generalized plane strain condition is obtained by allowing two extra nodes in each element. One of these nodes has the single degree of freedom of change of distance between the top and bottom of the structure at one point (that is, change in thickness at that point) while the other additional node contains the relative rotations θ_{xx} and θ_{yy} of the top surface with respect to the bottom surface about the same point. The generalized plane strain condition is attained by allowing these two nodes to be shared by all elements forming the structure. Note that this is an extension of the usual generalized plane strain condition which is obtained by setting the relative rotations to zero (by kinematic boundary conditions) and from there the element could become a plane strain element if the relative displacement is also made zero. Note that the sharing of the two extra nodes by all elements implies two full nodal rows in the generated plane strain part of the assembled stiffness matrix considerable computational savings are achieved if these two nodes are given the highest node numbers in that part of the structure.

This element uses biquadratic interpolation functions to represent the coordinates and displacements. Hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is represented using bilinear interpolation functions based upon the extra degree of freedom at the corner nodes.

Lower-order elements, such as type 81, are preferred in contact analyses.

The stiffness of this element is formed using four-point Gaussian integration. This is a reduced integration element – which may exhibit hourglass modes. This element should be used with caution. The mass matrix of this element is formed using nine-point Gaussian integration.

This element is designed to be used for incompressible elasticity only. It can be used for either small strain behavior or large strain behavior using the Mooney or Ogden models.This element can be used in conjunction with other elements such as type 56 when other material behavior, such as plasticity, must be represented.

This element cannot be used with the CASI iterative solver.

Quick Reference

Type 60

Second-order, isoparametric, distorted quadrilateral with reduced integration, generalized plane strain, hybrid formulation. See Marc Volume A: Theory and User Information for generalized plane strain theory.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention in x-y plane). Then the fifth node between first and second; the sixth node between second and third, etc. The ninth and tenth nodes are the generalized plane strain nodes shared with all elements in the mesh.

Geometry

Thickness stored in first data field (EGEOM1). Default thickness is unity. Other fields are not used.

Coordinates

At all ten nodes, two global coordinates, x and y. Note that the ninth and tenth nodes can be anywhere in the (xi) plane.

Degrees Of Freedom

At each of the first eight nodes:

1 = u = global x-direction displacement
2 = v = global y-direction displacement

Additional degree of freedom at the first four (corner) nodes:

3 = σ_{kk}/E	= mean pressure variable (isotropic)
= formula below (orthotropic)	
	$H = \left[\dfrac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1 + \dfrac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2 + \dfrac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3 \right] \dfrac{9}{E_1 + E_2 + E_3}$
= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)
= p / K	(for Herrmann elements using additive decomposition; K here is the effective bulk modulus)

One at the ninth node:

1 = Δz = relative z-direction displacement of front and back surfaces (see Figure 5-94).

Two at the tenth node:

1 = $\Delta\theta_x$ = relative rotation of front and back surfaces about global x-axis.
2 = $\Delta\theta_y$ = relative rotation of front and back surfaces about global y-axis (see Figure 5-94).

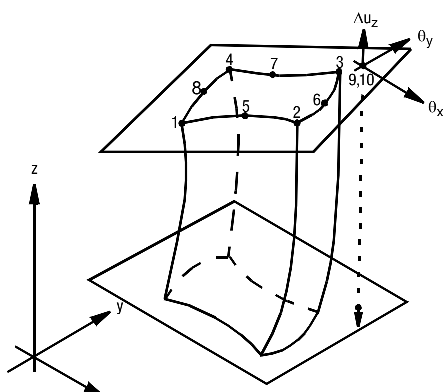


Figure 5-94 Generalized Plane Strain Distorted Quadrilateral

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type	Description
* 0	Uniform pressure on 1-5-2 face.
* 1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.
4	Uniform body force in y-direction.
5	Nonuniform body force in the y-direction.
* 6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
* 7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
* 8	Uniform pressure on 2-6-3 face.
* 9	Nonuniform pressure on 2-6-3 face.
* 10	Uniform pressure on 3-7-4 face.
* 11	Nonuniform pressure on 3-7-4 face.
* 12	Uniform pressure on 4-8-1 face.
* 13	Nonuniform pressure on 4-8-1 face.
* 20	Uniform shear force on 1-2-5 face in the 1⇒2⇒5 direction.
* 21	Nonuniform shear force on side 1-5-2.
* 22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
* 23	Nonuniform shear force on side 2-6-3.
* 24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
* 25	Nonuniform shear force on side 3-7-4.
* 26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
* 27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Load Type	Description
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine. Load types shown with an asterisk (*) require the magnitude of the load to be entered as force per unit area. To prescribe these loads in force per unit length, add 50 to the load type. This is often useful in design optimization where the thickness changes, but it is desired that the applied force remain the same.

Note that a nine-point scheme is used for the integration or body forces (see Figure 5-95).

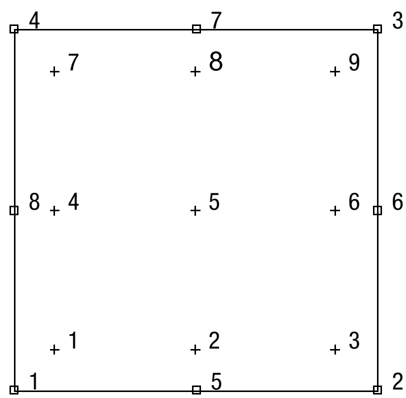


Figure 5-95 Integration Points for Body Force Calculations

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid or element integration points (see Figure 5-96 and Output Points) in the following order:

1 = ε_{xx}, direct

2 = ε_{yy}, direct

3 = ε_{zz}, thickness direction, direct

4 = γ_{xy} shear in the (x-y) plane

No γ_{xz} or γ_{yz} shear.

5	$= \sigma_{kk}/E$	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}} \sigma_1 + \frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}} \sigma_2 + \frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}} \sigma_3 \right] \frac{9}{E_1 + E_2 + E_3}$$	
	= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)

The last component is not printed in the output file, but is accessible via the ELEVAR and PLOTV user subroutines.

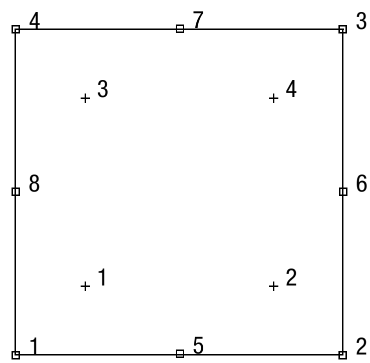


Figure 5-96 Integration Points for Eight-node Reduced Integration Element

Output of Stresses

Output of stresses is the same as first four Output of Strains .

Transformation

Only in x-y plane.

Tying

Use the UFORMSN user subroutine.

Output Points

If the CENTROID parameter is used, the output occurs at the centroid of the element.

If the ALL POINTS parameter is used, four output points are given as shown in Figure 5-96. This is the usual option for a second-order element.

Special Considerations

The mean pressure degree of freedom must be allowed to be discontinuous across changes in material properties. Use tying for this case.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available. Finite elastic strains can be calculated with the MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 69. See Element 69 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness (beam height) and/or the beam width can be considered as design variables.

Element 61

Three-dimensional, 20-node Brick with Reduced Integration, Herrmann Formulation

Element type 61 is a 20-node, isoparametric, arbitrary hexahedral written for incompressible applications using reduced integration. This element uses triquadratic interpolation functions to represent the coordinates and displacements. This allows for an accurate representation of the strain fields in elastic analyses. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is represented using trilinear interpolation functions based upon the extra degree of freedom at the corner nodes.

Lower-order elements, such as type 84, are preferred in contact analyses.

The stiffness of this element is formed using eight-point Gaussian integration. This is a reduced integration element – which may exhibit hourglass modes. This element should be used with caution. The mass matrix of this element is formed using twenty-seven-point Gaussian integration.

This element is designed to be used for incompressible elasticity only. It can be used for either small strain behavior or large strain behavior using the Mooney or Ogden models.This element can be used in conjunction with other elements such as type 57 when other material behavior, such as plasticity, must be represented.

Geometry

The geometry of the element is interpolated from the Cartesian coordinates of 20 nodes.

Connectivity

The convention for the ordering of the connectivity array is as follows:

> Nodes 1, 2, 3, 4 are the corners of one face, given in counterclockwise order when viewed from inside the element. Nodes 5, 6, 7, 8 are the corners of the opposite face; node 5 shares an edge with 1, 6 with 2, etc. Nodes 9, 10, 11, 12 are the middle of the edges of the 1, 2, 3, 4 face; node between 1 and 2, 10 between 2 and 3, etc. Similarly, nodes 13, 14, 15, 16 are midpoints on the 5, 6, 7, 8 face; node 13 between 5 and 6, etc. Finally, node 17 is the midpoint of the 1-5 edge; node 18 of the 2-6 edge, etc.

Note that in most normal cases, the elements are generated automatically, so that you need not concern yourself with the node numbering scheme.

Reduction to Wedge or Tetrahedron

The element can be reduced as far as a tetrahedron, simply be repeating node numbers. Element type 130 would be preferred for tetrahedrals.

Integration

The element is integrated numerically using eight points (Gaussian quadrature). The first plane of such points is closest to the 1, 2, 3, 4 face of the element, with the first point closest to the first node of the element (see Figure 5-97). A similar plane follows, moving toward the 5, 6, 7, 8 face. The “centroid” of the element is used for stress output if the CENTROID parameter is flagged.

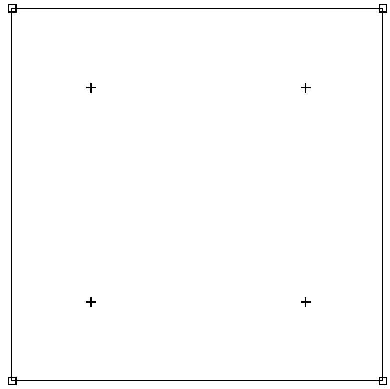


Figure 5-97 Plane of Integration Points for Reduced Integration Brick Element

The FORCEM user subroutine is called once per integration point when flagged. The magnitude of load defined by DIST LOADS is ignored and the FORCEM value is used instead.

Note that for integration of body forces, a 27-point integration scheme is used. Hence, for nonuniform body force, values must be provided for 27 points. Similarly, for nonuniform surface pressures, values need be supplied for nine integration points on the face of application.

Nodal (concentrated) loads can also be supplied.

Quick Reference

Type 61

Twenty-nodes, isoparametric arbitrary distorted cube with reduced integration. Herrmann formulation.

Connectivity

Twenty nodes numbered as described in the connectivity write-up for this element and is shown in Figure 5-98.

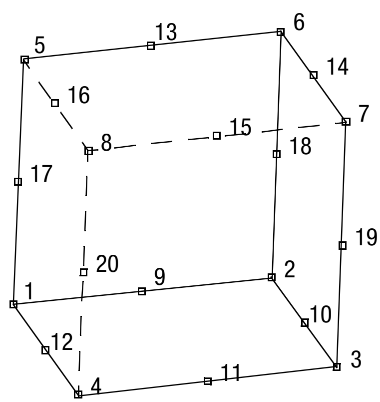


Figure 5-98 Twenty-node Brick Element

Geometry

Not required.

Coordinates

Three global coordinates in the x, y, and z directions.

Degrees of Freedom

Three global degrees of freedom, u, v, and w, at all nodes. Additional degrees of freedom at corner nodes (first 8 nodes) for Herrmann or Mooney is as follows:

$= \sigma_{kk}/E$	= mean pressure variable (isotropic)
= formula below (orthotropic)	
	$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1 + \frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2 + \frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3 \right] \frac{9}{E_1 + E_2 + E_3}$$
= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)
= p / K	(for Herrmann elements using additive decomposition; K here is the effective bulk modulus)

Distributed Loads

Distributed loads chosen by value of `IBODY` are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3-4 face.
1	Nonuniform pressure on 1-2-3-4 face; magnitude supplied through the FORCEM user subroutine.
2	Uniform body force per unit volume in -z direction.
3	Nonuniform body force per unit volume (e.g., centrifugal force); magnitude and direction supplied through the FORCEM user subroutine.
4	Uniform pressure on 6-5-8-7 face.
5	Nonuniform pressure on 6-5-8-7 face.
6	Uniform pressure on 2-1-5-6 face.
7	Nonuniform pressure on 2-1-5-6 face.
8	Uniform pressure on 3-2-6-7 face.
9	Nonuniform pressure on 3-2-6-7 face.
10	Uniform pressure on 4-3-7-8 face.
11	Nonuniform pressure on 4-3-7-8 face.
12	Uniform pressure on 1-4-8-5 face.

Load Type	Description
13	Nonuniform pressure on 1-4-8-5 face.
20	Uniform pressure on 1-2-3-4 face.
21	Nonuniform load on 1-2-3-4 face; magnitude and direction supplied in user subroutine FORCEM.
22	Uniform body force per unit volume in -z direction.
23	Nonuniform body force per unit volume (e.g., centrifugal force; magnitude supplied through the FORCEM user subroutine).
24	Uniform pressure on 6-5-8-7 face.
25	Nonuniform load on 6-5-8-7 face; magnitude and direction supplied in the FORCEM user subroutine.
26	Uniform pressure on 2-1-5-6 face.
27	Nonuniform load on 2-1-5-6 face; magnitude and direction supplied in the FORCEM user subroutine.
28	Uniform pressure on 3-2-6-7 face.
29	Nonuniform load on 3-2-6-7 face; magnitude and direction supplied in the FORCEM user subroutine.
30	Uniform pressure on 4-3-7-8 face.
31	Nonuniform load on 4-3-7-8 face; magnitude and direction supplied in the FORCEM user subroutine.
32	Uniform pressure on 1-4-8-5 face.
33	Nonuniform load on 1-4-8-5 face; magnitude and direction supplied in the FORCEM user subroutine.
40	Uniform shear 1-2-3-4 face in 12 direction.
41	Nonuniform shear 1-2-3-4 face in 12 direction.
42	Uniform shear 1-2-3-4 face in 23 direction.
43	Nonuniform shear 1-2-3-4 face in 23 direction.
48	Uniform shear 6-5-8-7 face in 56 direction.
49	Nonuniform shear 6-5-8-7 face in 56 direction.
50	Uniform shear 6-5-8-7 face in 67 direction.
51	Nonuniform shear 6-5-8-7 face in 67 direction.
52	Uniform shear 2-1-5-6 face in 12 direction.
53	Nonuniform shear 2-1-5-6 face in 12 direction.
54	Uniform shear 2-1-5-6 face in 15 direction.
55	Nonuniform shear 2-1-5-6 face in 15 direction.
56	Uniform shear 3-2-6-7 face in 23 direction.
57	Nonuniform shear 3-2-6-7 face in 23 direction.
58	Uniform shear 3-2-6-7 face in 26 direction.
59	Nonuniform shear 2-3-6-7 face in 26 direction.
60	Uniform shear 4-3-7-8 face in 34 direction.

Load Type	Description
61	Nonuniform shear 4-3-7-8 face in 34 direction.
62	Uniform shear 4-3-7-8 face in 37 direction.
63	Nonuniform shear 4-3-7-8 face in 37 direction.
64	Uniform shear 1-4-8-5 face in 41 direction.
65	Nonuniform shear 1-4-8-5 face in 41 direction.
66	Uniform shear 1-4-8-5 face in 15 direction.
67	Nonuniform shear 1-4-8-5 face in 15 direction.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Stress-strain output in global components:

1	$= \varepsilon_{xx}$	
2	$= \varepsilon_{yy}$	
3	$= \varepsilon_{zz}$	
4	$= \gamma_{xz}$	
5	$= \gamma_{yz}$	
6	$= \gamma_{zx}$	

7	= σ_{kk}/E	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1+\frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2+\frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3\right]\frac{9}{E_1+E_2+E_3}$	
	= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)

The last component is not printed in the output file, but is accessible via the ELEVAR and PLOTV user subroutines.

Output of Stresses

Output for stresses is the same as the first six Output of Strains .

Transformation

Three global degrees of freedom can be transformed to local degrees of freedom.

Tying

Use the UFORMSN user subroutine.

Output Points

Centroid or eight Gaussian integration points (see Figure 5-97).

Note:	A large bandwidth results in long run times – optimize.

Special Considerations

The mean pressure degree of freedom must be allowed to be discontinuous across changes in material properties. Use tying for this case.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available. Finite elastic strains can be calculated with the MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 71. See Element 71 for a description of the conventions used for entering the flux and film data for this element.

Element 62 Axisymmetric, Eight-node Quadrilateral for Arbitrary Loading (Fourier)

Element type 62 is an eight-node, isoparametric, arbitrary quadrilateral written for axisymmetric geometries with arbitrary loading. This allows for variation in the response in the circumferential direction by decomposing the behavior using Fourier series.

This element uses biquadratic interpolation functions to represent the coordinates and displacements. Hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses.

The stiffness of this element is formed using nine-point Gaussian integration.

This Fourier element can only be used for linear elastic analyses. No contact is permitted with this element.

Quick Reference

Type 62

Second-order, isoparametric, distorted quadrilateral for arbitrary loading of axisymmetric solids, formulated by means of the Fourier expansion technique.

Connectivity

Corner numbered first in counterclockwise order (right-handed convention). Then the fifth node between first and second; the sixth node between third, etc. See Figure 5-99.

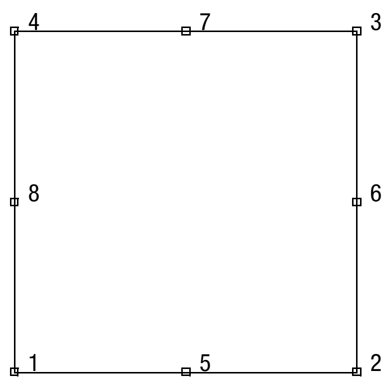


Figure 5-99 Eight-node Fourier Element

Geometry

No geometry input for this element.

Coordinates

Two global coordinates, z and r, at each node.

Degrees of Freedom

Three at each node:

1 = u = global z-direction (axial)
2 = v = global r-direction (radial)
3 = θ = global θ-direction (circumferential displacement)

Tractions

Surface Forces. Pressure and shear surface forces available for this element are listed below:

Load Type (IBODY)	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
6	Uniform shear force in direction 1⇒5⇒2 on 1-5-2 face.
7	Nonuniform shear force in direction 1⇒5⇒ 2 on 1-5-2 face.
8	Uniform pressure on 2-6-3.
9	Nonuniform pressure on 2-6-3.
10	Uniform pressure on 3-7-4.
11	Nonuniform pressure on 3-7-4.
12	Uniform pressure on 4-8-1 face.
13	Nonuniform pressure on 4-8-1 face.
14	Uniform shear in θ-direction (torsion) on 1-5-2 face.
15	Nonuniform shear in θ-direction (torsion) on 1-5-2 face.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

Body forces (per unit volume). Load type 2 is uniform body in the z-direction (axial), load type 3 is nonuniform body force in the z-direction. Load type 4 is uniform body force in the r-direction (radial), load type 5 is nonuniform body force in the r-direction. For load types 3 and 5, the FORCEM user subroutine must supply the force magnitude.

Concentrated nodal loads must be the value of the load integrated around the circumference.

For varying load magnitudes in the θ-direction, each distributed load or concentrated force can be associated with a different Fourier expansion. If no Fourier series is specified for a given loading, it is assumed to be constant around the circumference.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid or element integration points (see Figure 5-100 and Output Points) in the following order.

1 = ε_{zz}, direct
2 = ε_{rr}, direct
3 = $\varepsilon_{\theta\theta}$, direct
4 = γ_{zr}, in-plane shear
5 = $\gamma_{r\theta}$, out-of-plane shear
6 = $\gamma_{\theta z}$, out-of-plane shear

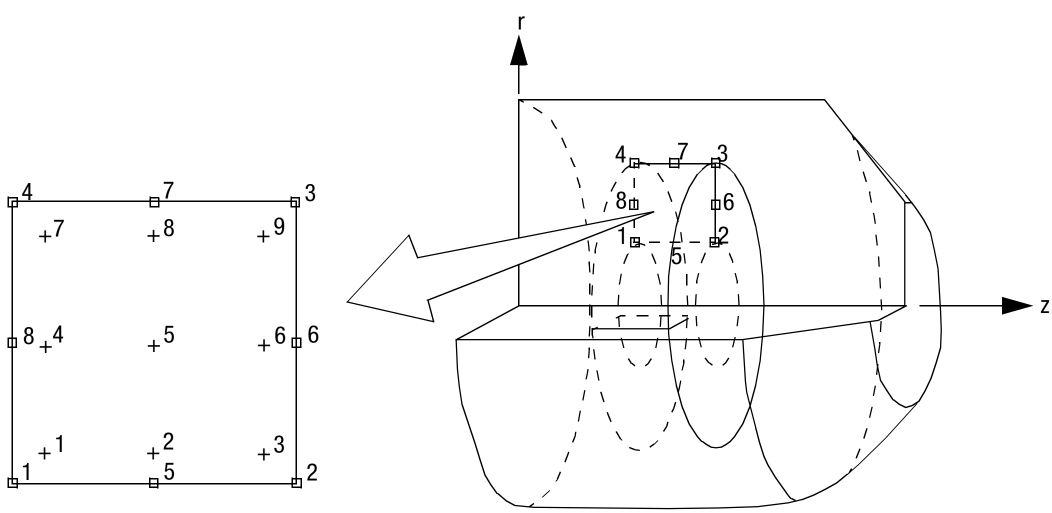


Figure 5-100 Integration Points of Eight-node 2-D Element

Output of Stresses

Same as for Output of Strains .

Transformations

Only in z-r plane.

Tying

Use the UFORMSN user subroutine.

Output Points

If the CENTROID parameter is used, the output occurs at the centroid of the element.

If the ALL POINTS parameter is used, the output is given for all nine integration points.

Element 63

Axisymmetric, Eight-node Distorted Quadrilateral for Arbitrary Loading, Herrmann Formulation (Fourier)

Element type 63 is an eight-node, isoparametric, arbitrary quadrilateral written for incompressible axisymmetric geometries with arbitrary loading. This allows for variation in the response in the circumferential direction by decomposing the behavior using Fourier series.

This element uses biquadratic interpolation functions to represent the coordinates and displacements. Hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is represented using bilinear interpolation functions based upon the extra degree of freedom at the corner nodes.

The stiffness of this element is formed using nine-point Gaussian integration.

This Fourier element can only be used for linear elastic analyses. No contact is permitted with this element.

Quick Reference

Type 63

Second-order, isoparametric quadrilateral for arbitrary loading of axisymmetric solids, formulated by means of the Fourier expansion technique. Hybrid formulation for incompressible or nearly incompressible materials.

Connectivity

Corners numbered first in counterclockwise order. Then the fifth node between first and second; the sixth node between second and third, etc. See Figure 5-101 and Figure 5-102.

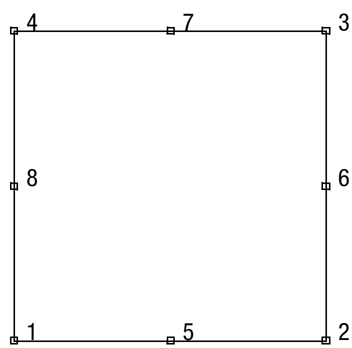


Figure 5-101 Eight-node Fourier Herrmann Element

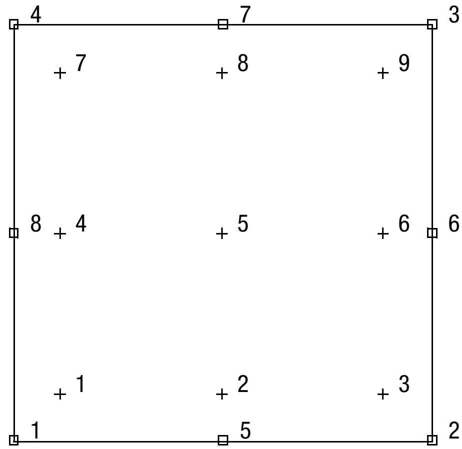


Figure 5-102 Integration Points for Eight-node Fourier Element

Geometry

No geometry input for this element.

Coordinates

Two global coordinates, z and r, at each of the eight nodes.

Degrees of Freedom

1 = u = global z-direction (axial)
2 = v = global r-direction (radial)
3 = θ = global θ-direction (circumferential displacement)

Additional degree of freedom at each corner node.

4 = σ_{kk}/E	= mean pressure variable (isotropic)
= formula below (orthotropic)	

$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1 + \frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2 + \frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3 \right] \frac{9}{E_1+E_2+E_3}$$

= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)
= p / K	(for Herrmann elements using additive decomposition; K here is the effective bulk modulus)

Tractions

Surface Forces. Pressure and shear forces are available for this element as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
6	Uniform shear force in direction 1⇒5⇒2 on 1-5-2 face.
7	Nonuniform shear force in direction 1⇒5⇒2 on 1-5-2 face.
8	Uniform pressure on 2-6-3.
9	Nonuniform pressure on 2-6-3.
10	Uniform pressure on 3-7-4.
11	Nonuniform pressure on 3-7-4.
12	Uniform pressure on 4-8-1 face.
13	Nonuniform pressure on 4-8-1 face.
14	Uniform shear in θ-direction (torsion) on 1-5-2 face.
15	Nonuniform shear in θ-direction (torsion) on 1-5-2 face.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

Body forces (per unit volume). Load type 2 is uniform body in the z-direction (axial), load type 3 is nonuniform body force in the z-direction. Load type 4 is uniform body force in the r-direction (radial), load type 5 is nonuniform body force in the r-direction. For load types 3 and 5, the FORCEM user subroutine must supply the force magnitude.

Concentrated nodal loads must be the value of the load integrated around the circumference.

For varying load magnitudes in the θ-direction, each distributed load or concentrated force can be associated with a different Fourier expansion. If no Fourier series is specified for a given loading, it is assumed to be constant around the circumference.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid or element integration points (see Figure 5-102 and Output Points) in the following order:

1	$= \varepsilon_{zz}$	direct
2	$= \varepsilon_{rr}$	direct
3	$= \varepsilon_{\theta\theta}$	direct
4	$= \gamma_{zr}$	in-plane shear
5	$= \gamma_{r\theta}$	out-of-plane shear
6	$= \gamma_{\theta z}$	out-of-plane shear
7	$= \sigma_{kk}/2G(1 + \upsilon)$	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	$H = \left[\dfrac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1+\dfrac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\Delta_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2+\dfrac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3\right]\dfrac{9}{E_1+E_2+E_3}$	
	= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)

The last component is not printed in the output file, but is accessible via the ELEVAR and PLOTV user subroutines.

Output of Stresses

Same as for Output of Strains .

Transformation

Only in z-r plane.

Tying

Use the UFORMSN user subroutine.

Output Points

If the CENTROID parameter is used, the output occurs at the centroid of the element, shown as point 5 in Figure 5-102.

If the ALL POINTS parameter is used, the output is given for all nine integration points.

Element 64

Isoparametric, Three-node Truss

This element is a quadratic, three-node truss with constant cross section. The strain-displacement relations are written for large strain, large displacement analysis. Three-point Gaussian integration is used along the element. The degrees of freedom are the u, v, and w displacements at the three nodes of the element.

This element is very useful as reinforcement element in conjunction with the two and three-dimensional second order isoparametric elements in Marc. Possible applications include the use as a string in membrane or as discrete reinforcement string in composite materials. All constitutive relations can be used with this element.

Quick Reference

Type 64

Three-dimensional, three-node, isoparametric truss.

Connectivity

Three nodes per element (see Figure 5-103).

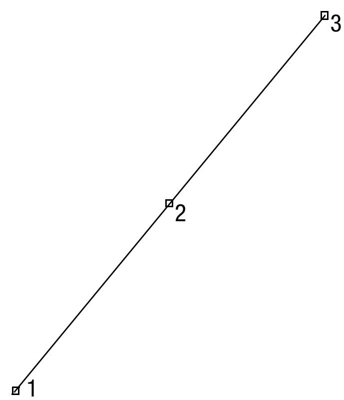


Figure 5-103 Isoparametric Truss Element

Geometry

The cross-sectional area is input in the first data field (EGEOM1). The other two data fields are not used. If not specified, the cross-sectional area defaults to 1.0.

If beam-to-beam contact is switched on (see CONTACT option), the radius used when the element comes in contact with other beam or truss elements must be entered in the 7th data field (EGEOM7).

Coordinates

Three coordinates per node in the global x-, y-, and z-direction.

Degrees of Freedom

1 = u displacement
2 = v displacement
3 = w displacement

Tractions

Distributed loads according to the value of `IBODY` are as follows:

Load Type	Description
0	Uniform load in the direction of the global x-axis per unit volume.
1	Uniform load in the direction of the global y-axis per unit volume.
2	Uniform load in the direction of the global z-axis per unit volume.
11	Fluid drag/buoyancy loading – fluid behavior specified in the FLUID DRAG option.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Uniaxial in the truss member.

Output of Stresses

Uniaxial in the truss member.

Transformation

The three global degrees of freedom for any node can be transformed to local degrees of freedom.

Tying

Use the UFORMSN user subroutine.

Output Points

Centroid or three Gaussian integration points along the truss if the ALL POINTS parameter is used. First point is closest to first node given; second point is centroid; third point is closest to third node of truss. See Figure 5-104.

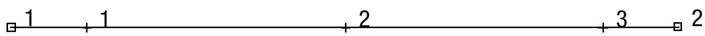


Figure 5-104 Integration Points for Element Type 64

Updated Lagrange and Finite Strain Plasticity

Capability is available; area is updated.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 65. See Element 65 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The cross-sectional area can be considered as a design variable.

Element 65

Heat Transfer Element, Three-node Link

This element is a quadratic heat link with constant cross-sectional area. It is the heat-transfer equivalent of element type 64.

Quick Reference

Type 65

Three-dimensional, three-node, heat transfer link.

Connectivity

Three nodes per element (see Figure 5-105).

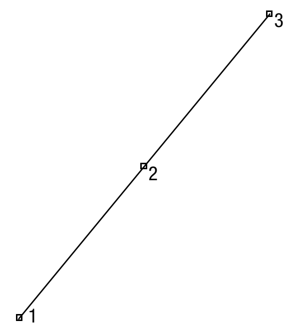


Figure 5-105 Three-node Heat Transfer Element

Geometry

The cross-sectional area is input in the first data field (`EGEOM1`); the other fields are not used. If not specified, the cross-sectional area defaults to 1.0.

If the element needs to be offset from the user-specified position, the eighth data field (EGEOM8) is set to -1.0. In this case, an extra line containing the offset information is provided (see the GEOMETRY option in Marc Volume C: Program Input). The offset vector at the first corner node is provided in the first three fields and the offset vector at the second corner node is provided in the next three fields of the extra line. If the interpolation flag (seventh data field) is set to 1, then the offset vector at the midside node is obtained by interpolating from the corresponding corner vectors. The coordinate system used for the offset vectors at the two nodes are indicated by the corresponding flags in the eighth and ninth fields of the extra line. A flag setting of 0 indicates global coordinate system, 2 indicates local shell normal, and 3 indicates local nodal coordinate system. The flag setting of 2 can be used when the link nodes are also attached to a shell. In this case, the offset magnitude at the first and second nodes are specified in the first and fourth data fields of the extra line respectively and the automatically calculated shell normals at the nodes are used to construct the offset vectors.

Coordinates

Three coordinates per node in the global x-, y-, and z-direction.

Degrees of Freedom

One degree of freedom per node:

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)

Fluxes

Distributed fluxes according to value of `IBODY` are as follows:

Flux Type	Description
0	Uniform flux on first node (per cross-sectional area)*
1	Uniform flux on last node (per cross-sectional area)*
2	Volumetric flux on entire element (per volume)
3	Nonuniform flux on entire element (per volume)
*Flux types 0 and 1 are not supported using table input.	

Films

Same specifications as Fluxes .

Tying

Use the UFORMSN user subroutine.

Joule Heating

Capability is available.

Electrostatic

Capability is not available.

Magnetostatic

Capability is not available.

Current

Same specification as Fluxes .

Output Points

Centroid or three Gaussian integration points along the truss if the ALL POINTS parameter is used. First point is closest to first node given; second point is centroid; third point is closest to third node. See Figure 5-106.

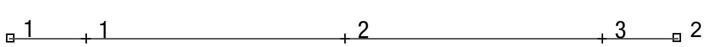


Figure 5-106 Integration Points for Element Type 65

Element 66 Eight-node Axisymmetric Herrmann Quadrilateral with Twist

The modified axisymmetric (includes a twist mode of deformation), eight-node distorted quadrilateral – suitable for materially linear, elastic, and incompressible or nearly incompressible deformation (Herrmann formulation) as well as nonlinear elastic incompressible Mooney-Rivlin or Ogden behavior and/or some other higher order forms of hyperelastic deformation.

If the analysis involves large rotations about the symmetry axis, it is recommended to use a full 3-D model.

Quick Reference

Type 66

Second-order, isoparametric, distorted quadrilateral. Axisymmetric formulation modified to include a twist mode of deformation. Hybrid (Herrmann) formulation of incompressible or nearly incompressible materials.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention in z-r plane), then the fifth node between first and second; the sixth node between second and third, etc. See Figure 5-107.

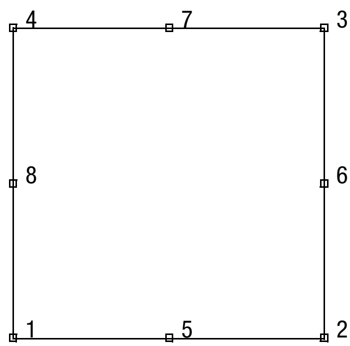


Figure 5-107 Eight-node Axisymmetric Herrmann with Twist

Geometry

No geometry input for this element.

Coordinates

Two global coordinates, z and r, at each node.

Degrees of Freedom

1	$= u_z$	global z-direction displacement (axial).
2	$= u_R$	global R-direction displacement (radial). See Figure 5-108.
3	$= u_\theta$	global θ-direction displacement (tangential) in radians. See Figure 5-108.

4	$= \sigma_{kk}/E$	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1 + \frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2 + \frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3 \right] \frac{9}{E_1+E_2+E_3}$$	
	= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)hydrostatic pressure variable – only at the corner nodes.
	= p / K	(for Herrmann elements using additive decomposition; K here is the effective bulk modulus)

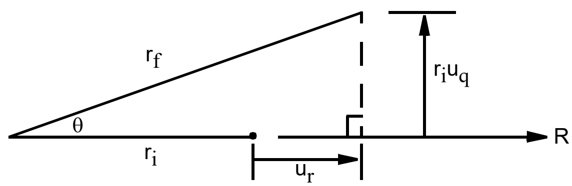


Figure 5-108 Radial and Tangent Displacements

Tractions

Surface Forces. Pressure and shear (in the z-r plane) forces available for this element are as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force per unit volume in the z-direction (axial).
3	Nonuniform body force per unit volume in the z-direction (axial).
4	Uniform body force per unit volume in the r-direction (radial).
5	Nonuniform body force per unit volume in the r-direction (radial).
6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
8	Uniform pressure on 2-6-3 face.
9	Nonuniform pressure on 2-6-3 face.
10	Uniform pressure on 3-7-4 face.
11	Nonuniform pressure on 3-7-4 face.
12	Uniform pressure on 4-8-1 face.
13	Nonuniform pressure on 4-8-1 face.

Load Type (IBODY)	Description
20	Uniform shear force on 1-2-5 face in the 1⇒2⇒5 direction.
21	Nonuniform shear force on side 1-5-2.
22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
23	Nonuniform shear force on side 2-6-3.
24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
25	Nonuniform shear force on side 3-7-4.
26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

Note that a nine-point scheme is used for the integration or body forces.

Note that “loads” associated with the third degree of freedom, can only be specified, at this time, as concentrated nodal values. These “loads” actually correspond to a torque in the θ - Z plane. Also note that in specifying any concentrated nodal load, the value to be used should be that obtained by integration around the entire circumference defined by the radius of the nodal point.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

The total components of strain are printed out at the centroid or element integration points (see Figure 5-109 and Output Points) in the following order:

1	$= E_{ZZ}$,	direct axial
2	$= E_{RR}$,	direct radial
3	$= E_{\theta\theta}$,	direct hoop
4	$= 2E_{ZR}$	(= γ_{ZR}, the engineering definition of strain for small deformations), shear in the section
5	$= 2E_{R\theta}$	(= $\gamma_{R\theta}$, for small deformations), warping of radial lines
6	$= 2E_{\theta Z}$	(= $\gamma_{\theta Z}$, for small deformations), twist deformation
7	$= \sigma_{kk}/E$	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	$H = \left[\dfrac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1+\dfrac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\Delta_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2+\dfrac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3\right]\dfrac{9}{E_1+E_2+E_3}$	
	$= -p$	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)

where E_{ij} are the physical components of the Green's tensor referred to the initial cylindrical reference system (that is, Lagrangian strain).

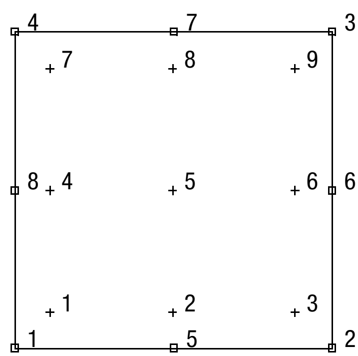


Figure 5-109 Integration Points for Element 66

Output Of Stresses

The stress components that are conjugate to the Green's strain components (listed above), are also printed. These are the physical components of the symmetric second Piola-Kirchhoff stress, S_{ij}. In addition, the physical components of the Cauchy stress tensor are also printed.

Transformation

Only in z-r plane.

Tying

Use the UFORMSN user subroutine.

Output Points

If the CENTROID parameter is used, the output occurs at the centroid of the element, given as point 5 in Figure 5-109. If the ALL POINTS parameter is used, nine output points are given, as shown in Figure 5-109. This is the usual option for a second-order element, particularly when material and/or geometric nonlinearities are present.

Special Considerations

The mean pressure degree of freedom must be allowed to be discontinuous across changes in material properties. Use tying for this case. See description of Marc element type 67 for a compressible (that is, conventional isoparametric axisymmetric formulation) element that is compatible (that is, includes the twist deformation) with this Herrmann type element.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available. Finite elastic strains can be calculated with the MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 42. See Element 42 for a description of the conventions used for entering the flux and film data for this element.

Element 67 Eight-node Axisymmetric Quadrilateral with Twist

Element type 67 is an eight-node, isoparametric, arbitrary quadrilateral written for axisymmetric applications with torsional strains. This element uses biquadratic interpolation functions to represent the coordinates and displacements. Hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses.

Lower-order elements, such as type 20, are preferred in contact analyses.

The stiffness of this element is formed using nine-point Gaussian integration.

This element can be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 66 instead. Element type 66 is also preferable for small strain incompressible elasticity.

Notice that there is no friction contribution in the torsional direction when the CONTACT option is used.

If the analysis involves large rotations about the symmetry axis, it is recommended to use a full 3-D model.

Quick Reference

Type 67

Second-order, isoparametric, distorted quadrilateral. Axisymmetric formulation modified to include a twist mode of deformation. Conventional counterpart to incompressible element type 66 described above.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention in z-r plane). Then the fifth node between first and second; the sixth node between second and third, etc. See Figure 5-110.

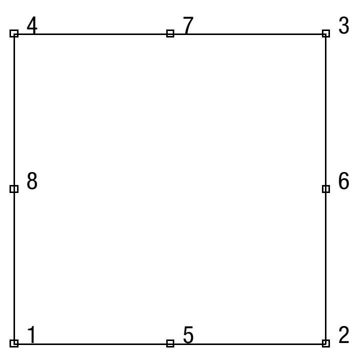


Figure 5-110 Eight-node Axisymmetric with Twist

Geometry

No geometry input for this element.

Coordinates

Two global coordinates, z and r, at each node.

Degrees of Freedom

1 = u_z,	global z-direction displacement (axial).
2 = u_R,	global R-direction displacement (radial). See Figure 5-111.
3 = u_θ,	global θ-direction displacement (tangential) in radians. See Figure 5-111.

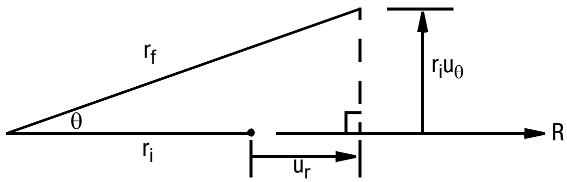


Figure 5-111 Radial and Tangent Displacements

Tractions

Surface forces. Pressure and shear (in the r-z plane) forces are available for this element as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force per unit volume in the z-direction (axial).
3	Nonuniform body force per unit volume in the z-direction (axial).
4	Uniform body force per unit volume in the r-direction (radial).
5	Nonuniform body force per unit volume in the r-direction. (radial)
6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
8	Uniform pressure on 2-6-3 face.
9	Nonuniform pressure on 2-6-3 face.
10	Uniform pressure on 3-7-4 face.
11	Nonuniform pressure on 3-7-4 face.
12	Uniform pressure on 4-8-1 face.
13	Nonuniform pressure on 4-8-1 face.
20	Uniform shear force on 1-2-5 face in the 1⇒2⇒5 direction.
21	Nonuniform shear force on side 1-5-2.
22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
23	Nonuniform shear force on side 2-6-3.
24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.

Load Type (IBODY)	Description
25	Nonuniform shear force on side 3-7-4.
26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

Note that “loads” associated with the third degree of freedom, θ, can only be specified, at this time, as concentrated nodal values. These “loads” actually correspond to a torque in the θ - z plane. Also note that in specifying any concentrated nodal load, the value to be used should be that obtained by integration around the entire circumference defined by the radius of the nodal point.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

The total components of strain are printed out at the centroid or element integration points (see Figure 5-112 and Output Points) in the following order:

1 = E_{ZZ},	direct axial
2 = E_{RR},	direct radial
3 = $E_{\theta\theta}$,	direct hoop
4 = $2E_{ZR}$	(= γ_{ZR}, the engineering definition of strain for small deformations), shear in the section
5 = $2E_{R\theta}$	(= $\gamma_{R\theta}$ for small deformations), warping of radial lines
6 = $2E_{\theta Z}$	(= $\gamma_{\theta Z}$, for small deformations), twist deformation

where E_{ij} are the physical components of the Green's strain tensor referred to the initial cylindrical reference system (that is, Lagrangian strain).

Output of Stresses

The stress components, that are conjugate to the Green's strain components (listed above), are also printed. These are the physical components of the symmetric second Piola-Kirchhoff stress, S_{ij}.

Transformation

Only in z-r plane.

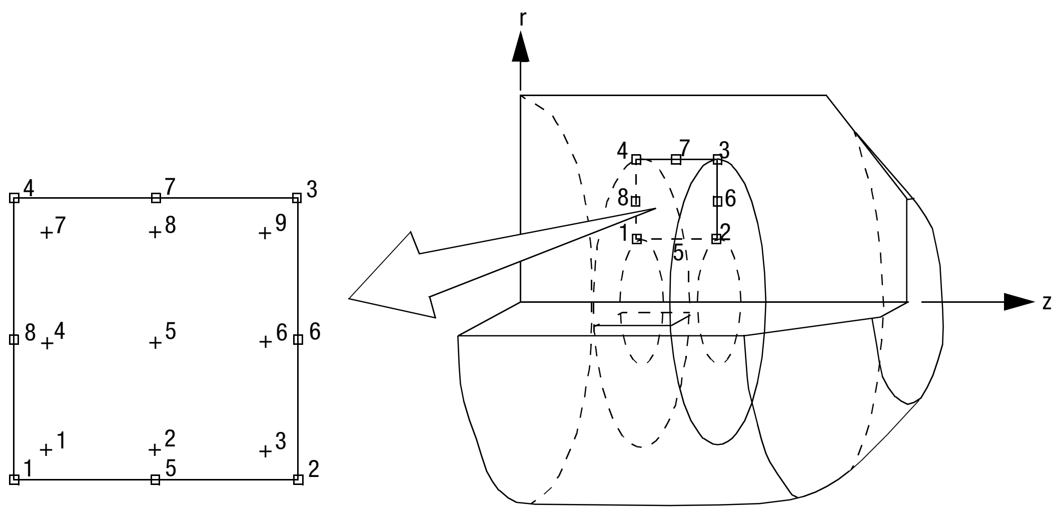


Figure 5-112 Integration Points for Element 67

Tying

Use the UFORMSN user subroutine.

Output Points

If the CENTROID parameter is used, the output occurs at the centroid of the element, given as point 5 in Figure 5-112. If the ALL POINTS parameter is used, nine output points are given, as shown in Figure 5-112. This is the usual option for a second-order element, particularly when material and/or geometric nonlinearities are present.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 42. See Element 42 for a description of the conventions used for entering the flux and film data for this element.

Element 68

Elastic, Four-node Shear Panel

This is a linear-elastic shear panel of arbitrary shape. A shear panel is an idealized model of an elastic sheet. If there are stiffeners present, the panel resists the shearing forces and the stiffeners resist normal and bending forces. The generalization to an arbitrary shape is due to S. J. Garvey[1]. Due to the simplifications involved, the response of the element is restricted to linear materials and large displacement effects are neglected. The stiffness matrix is found in closed form.

Geometric Basis

The element is formulated in a local plane defined by the two diagonals. If they do not intersect, the plane is located midway between the diagonals.

Displacements

The displacements at each node are:

u, v, w global Cartesian components

Connectivity Specification

The element has four nodes. They can be listed in clockwise or counterclockwise order.

Quick Reference

Type 68

Linear-elastic shear panel.

Connectivity

Four nodes per element (see Figure 5-113).

Geometry

Thickness in first data field (EGEOM1).

Coordinates

1 = x
2 = y
3 = z

1 Garvey, S. J., “The Quadrilateral Shear Panel”, *Aircraft Engineering*, p. 134, May 1951.

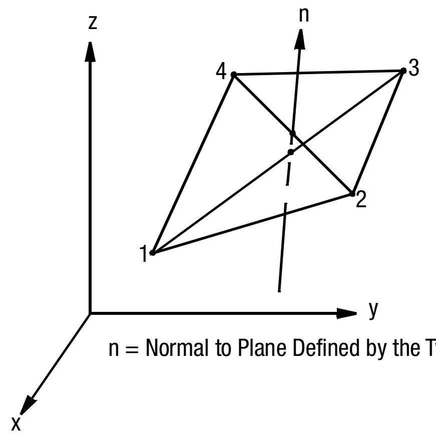


Figure 5-113 Shear Panel

Degrees of Freedom

1 = u
2 = v
3 = w

Tractions

Only concentrated forces at nodal points.

Output of Stresses

Shear stress at all four nodal points. Average shear stresses and maximum shear stresses are printed.

$$\tau_{avg} = 1/4(\tau_1 + \tau_2 + \tau_3 + \tau_4)$$

$$\tau_{max} = max\ (|\tau_1|, |\tau_2|, |\tau_3|, |\tau_4|)$$

Transformation

The degrees of freedom can be transformed to local directions.

Tying

Use tying (UFORMSN user subroutine) needed to formulate constraints between (u, v, w) degrees of freedom of beams and shear panels.

Updated Lagrangian Procedure and Finite Strain Plasticity

Capability is not available.

Coupled Analysis

There is no heat transfer equivalent of this element; so a coupled thermal-mechanical analysis is not possible.

Element 69

Eight-node Planar Biquadratic Quadrilateral with Reduced Integration (Heat Transfer Element)

Element type 69 is an eight-node, isoparametric, arbitrary quadrilateral written for planar heat transfer applications using reduced integration. This element can also be used for electrostatic or magnetostatic applications.

This element uses biquadratic interpolation functions to represent the coordinates and displacements. Hence, the thermal gradients have a linear variation. This allows for accurate representation of the temperature field.

This element can also be used as a thermal contact or a fluid channel element. The CONRAD GAP and CHANNEL model definition options must be used for thermal contact and fluid channel options, respectively. A description of the thermal contact and fluid channel capabilities is included in Marc Volume A: Theory and User Information . Note that in thermal contact and fluid channel options, the gap face and fluid channel face identifications must be entered for each gap/channel. Face identifications for this element are given in the Face Identifications (Thermal Contact Gap and Fluid Channel Options) .

The conductivity of this element is formed using four-point Gaussian integration. This is a reduced integration element – which may exhibit hourglass modes. This element should be used with caution. The specific heat capacity matrix of this element is formed using nine-point Gaussian integration.

Quick Reference

Type 69

Second-order, distorted heat quadrilateral with reduced integration.

Connectivity

Corner nodes 1-4 numbered first in right-handed convention. Nodes 5-8 are midside nodes starting with node 5 in between 1 and 2, and so on (see Figure 5-114).

Geometry

Thickness stored in `EGEOM1` field. If not specified, unit thickness is assumed.

Coordinates

Two global coordinates per node – x and y.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)
1 = potential (magnetostatic)

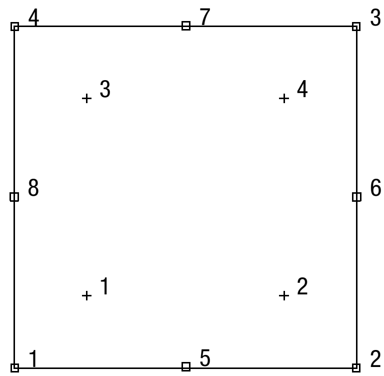


Figure 5-114 Eight-Node Planar Heat Quadrilateral with Reduced Integration

Fluxes

Surface Fluxes

Surface fluxes are specified below. All are per unit surface area. All nonuniform fluxes are specified through the FLUX user subroutine.

Flux Type (IBODY)	Description
0	Uniform flux on 1-5-2 face.
1	Nonuniform flux on 1-5-2 face.
8	Uniform flux on 2-6-3 face.
9	Nonuniform flux on 2-6-3 face.
10	Uniform flux on 3-7-4 face.
11	Nonuniform flux on 3-7-4 face.
12	Uniform flux on 4-8-1 face.
13	Nonuniform flux on 4-8-1 face.

Volumetric Fluxes

Flux type 2 is uniform flux (per unit volume); type 3 is nonuniform flux per unit volume, with magnitude given through the FLUX user subroutine.

Films

Same specification as Fluxes .

Tying

Use the UFORMSN user subroutine.

Output Points

Centroid or four Gaussian integration points.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Current

Same specifications as Fluxes .

Charge

Same specifications as Fluxes .

Face Identifications (Thermal Contact Gap and Fluid Channel Options)

Gap Face Identification	Radiative/Convective Heat Transfer Takes Place Between Edges
1	(1 - 5 - 2) and (3 - 7 - 4)
2	(4 - 8 - 1) and (2 - 6 - 3)

Fluid Channel Face Identification	Flow Direction From Edge to Edge
1	(1 - 5 - 2) to (3 - 7 - 4)
2	(4 - 8 - 1) to (2 - 6 - 3)

Element 70

Eight-node Axisymmetric Biquadrilateral with Reduced Integration (Heat Transfer Element)

Element type 70 is an eight-node, isoparametric, arbitrary quadrilateral written for axisymmetric heat transfer applications using reduced integration. This element can also be used for electrostatic or magnetostatic applications.

This element uses biquadratic interpolation functions to represent the coordinates and displacements. Hence, the thermal gradients have a linear variation. This allows for accurate representation of the temperature field.

This element can also be used as a thermal contact or a fluid channel element. The CONRAD GAP and CHANNEL model definition options must be used for thermal contact and fluid channel options, respectively. A description of the thermal contact and fluid channel capabilities is included in Marc Volume A: Theory and User Information . Note that in thermal contact and fluid channel options, the gap face and fluid channel face identifications must be entered for each gap/channel. Face identifications for this element are given in the Face Identifications (Thermal Contact Gap and Fluid Channel Options) .

The conductivity of this element is formed using four-point Gaussian integration. This is a reduced integration element – which may exhibit hourglass modes. This element should be used with caution. The specific heat capacity matrix of this element is formed using nine-point Gaussian integration.

Quick Reference

Type 70

Second-order, distorted axisymmetric heat quadrilateral with reduced integration.

Connectivity

Corner nodes 1-4 numbered first in right-handed convention. Nodes 5-8 are midside nodes with node 5 located between 1 and 2, etc. (see Figure 5-115).

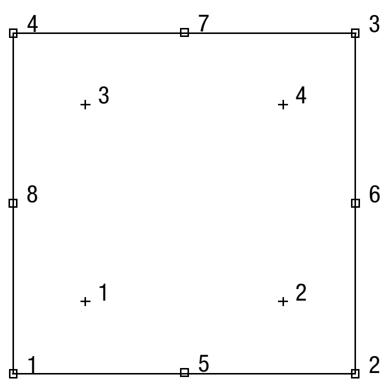


Figure 5-115 Eight-node Axisymmetric Heat Quadrilateral with Reduced Integration

Geometry

Not applicable.

Coordinates

Two global coordinates per node, z and r.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)
1 = potential (magnetostatic)

Fluxes

Surface Fluxes

Surface fluxes are specified below. Surface flux magnitudes are input per unit surface area. The magnitude of nonuniform surface fluxes must be specified through the FLUX user subroutine.

Flux Type	Description
0	Uniform flux on 1-5-2 face.
1	Nonuniform flux on 1-5-2 face.
8	Uniform flux on 2-6-3 face.
9	Nonuniform flux on 2-6-3 face.
10	Uniform flux on 3-7-4 face.
11	Nonuniform flux on 3-7-4 face.
12	Uniform flux on 4-8-1 face.
13	Nonuniform flux on 4-8-1 face.

Volumetric Fluxes

Flux type 2 is uniform flux (per unit volume), type 3 is nonuniform flux per unit volume, with magnitude given through the FLUX user subroutine.

Films

Same specifications as Fluxes .

Tying

Use the UFORMSN user subroutine.

Output Points

Centroid or four Gaussian integration points.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Current

Same specifications as Fluxes .

Charge

Same specifications as Fluxes .

Face Identifications (Thermal Contact Gap and Fluid Channel Options)

Gap Face Identification	Radiative/Convective Heat Transfer Takes Place Between Edges
1	(1 - 5 - 2) and (3 - 7 - 4)
2	(4 - 8 - 1) and (2 - 6 - 3)

Fluid Channel Face Identification	Flow Direction From Edge to Edge
1	(1 - 5 - 2) to (3 - 7 - 4)
2	(4 - 8 - 1) to (2 - 6 - 3)

View Factors Calculation For Radiation

Capability is available.

Element 71

Three-dimensional 20-node Brick with Reduced Integration (Heat Transfer Element)

Element type 71 is a 20-node, isoparametric, arbitrary hexahedral written for three-dimensional heat transfer applications using reduced integration. This element can also be used for electrostatic applications.

This element uses triquadratic interpolation functions to represent the coordinates and displacements. Hence, the thermal gradients have a linear variation. This allows for accurate representation of the temperature field.

This element can also be used as a thermal contact or a fluid channel element. The CONRAD GAP and CHANNEL model definition options must be used for thermal contact and fluid channel options, respectively. A description of the thermal contact and fluid channel capabilities is included in Marc Volume A: Theory and User Information . Note that in thermal contact and fluid channel options, the gap face and fluid channel face identifications must be entered for each gap/channel. Face identifications for this element are given in the Face Identifications (Thermal Contact Gap and Fluid Channel Options) .

The conductivity of this element is formed using eight-point Gaussian integration. This is a reduced integration element – which may exhibit hourglass modes. This element should be used with caution. The specific heat capacity matrix of this element is formed using twenty-seven-point Gaussian integration.

Connectivity

The convention for the ordering of the connectivity array is as follows:

> Nodes 1, 2, 3, and 4 are corners of one face, given in a counterclockwise direction when viewed from inside the element. Nodes 5, 6, 7, and 8 are the corners of the opposite face; node 5 shares an edge with 1; node 6 with 2, etc. Nodes 9, 10, 11, and 12 are the middles of the edges of the 1, 2, 3, 4 face; node 9 between 1 and 2; node 10 between 2 and 3, etc. Similarly, nodes 13, 14, 15, and 16 are midpoints on the 5, 6, 7, and 8 face; node 13 between 5 and 6, etc. Finally, node 17 is the midpoint of the 1-5 edge; node 18 of the 2-6 edge, etc. (see Figure 5-116).

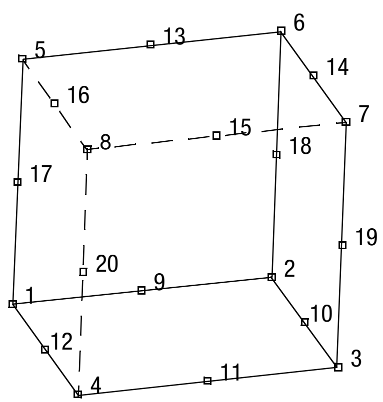


Figure 5-116 Form of Element 71

Integration

The element is integrated numerically using eight points (Gaussian quadrature). The first plane of such points is closest to the 1, 2, 3, 4 face of the element, with the first point closest to the first node of the element (see Figure 5-117). One similar plane follows, closest to the 5, 6, 7, 8 face.

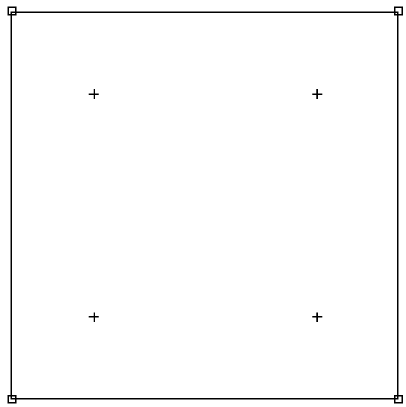


Figure 5-117 Points of Integration in a Sample Integration Plane

Reduction to Wedge or Tetrahedron

The element can be reduced as far as a tetrahedron, simply by repeating node numbers on the same spatial node position. Element type 122 is preferred for tetrahedrals.

Notes:	A large bandwidth results in long run times. Optimize as much as possible.
	The lumped specific heat option gives poor results with this element at early times in transient solutions. If accurate transient analysis is required, you should not use the lumping option with this element.

Quick Reference

Type 71

Twenty-node isoparametric brick (heat transfer element) with reduced integration.

Connectivity

Twenty-nodes numbered as described in the connectivity write-up for this element, and as shown in Figure 5-116.

Geometry

Not applicable.

Coordinates

Three global coordinates in x-, y-, z-directions.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)

Fluxes

Distributed fluxes given by a type specification are as follows:

Load Type	Description
0	Uniform flux on 1-2-3-4 face.
1	Nonuniform flux on 1-2-3-4 face (FLUX user subroutine).
2	Uniform body flux.
3	Nonuniform body flux (FLUX user subroutine).
4	Uniform flux on 6-5-8-7 face.
5	Nonuniform flux on 6-5-8-7 face (FLUX user subroutine).
6	Uniform flux on 2-1-5-6 face.
7	Nonuniform flux on 2-1-5-6 face (FLUX user subroutine).
8	Uniform flux on 3-2-6-7 face.
9	Nonuniform flux on 3-2-6-7 face (FLUX user subroutine).
10	Uniform flux on 4-3-7-8 face.
11	Nonuniform flux on 4-3-7-8 face (FLUX user subroutine).
12	Uniform flux on 1-4-8-5 face.
13	Nonuniform flux on 1-4-8-5 face (FLUX user subroutine).

For type=3, the value of P in the FLUX user subroutine is the magnitude of volumetric flux at volumetric integration point NN of element N. For type odd but not equal to 3, P is the magnitude of surface flux at surface integration point NN of element N. Surface flux is positive when heat energy is added to the element.

Films

Same specifications as Fluxes .

Tying

Use the UFORMSN user subroutine.

Output Points

Centroid or eight Gaussian integration points.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Current

Same specifications as Fluxes .

Charge

Same specifications as Fluxes .

Face Identifications (Thermal Contact Gap and Fluid Channel Options)

Gap Face Identification	Radiative/Convective Heat Transfer Takes Place Between Faces
1	(1 - 2 - 6 - 5) and (3 - 4 - 8 - 7)
2	(2 - 3 - 7 - 6) and (1 - 4 - 8 - 5)
3	(1 - 2 - 3 - 4) and (5 - 6 - 7 - 8)

Fluid Channel Face Identification	Flow Direction From Edge to Edge
1	(1 - 2 - 6 - 5) to (3 - 4 - 8 - 7)
2	(2 - 3 - 7 - 6) to (1 - 4 - 8 - 5)
3	(1 - 2 - 3 - 4) to (5 - 6 - 7 - 8)

Element 72

Bilinear Constrained Shell Element

This is an eight-node, thin-shell element with zero-order degrees of freedom. Bilinear interpolation is used both for global displacements and coordinates. Global rotations are interpolated quadratically from the rotation vectors at the centroid and at the midside nodes. At these midside nodes, constraints are imposed on the rotations by relating the rotation normal to the boundary as well as the rotation about the surface normal to the local displacements. In addition, all three rotation components are related to the local displacements at the centroid. In this way a very efficient and simple element is obtained. The element can be used in curved shell analysis but also for the analysis of complicated plate structures. For the latter case, this element is easier to use than the usual plate elements, since the absence of local higher order degrees of freedom allows for direct connections between folded plates without tying requirements along the folds.

Due to its simple formulation when compared to the standard higher-order shell elements, it is less expensive, and therefore, very attractive in nonlinear analyses. The element is fairly insensitive to distortion, particularly if the corner nodes lie in the same plane. In that case, all constant bending modes are represented exactly. The element can be degenerated to a triangle by collapsing one of the sides. The midside node on that side then becomes a dummy node and can be given the same number as the end nodes of that side. Thus, the element degenerates to a constant shear – constant bending plane triangle. All constitutive relations can be used with this element.

Geometric Basis

The element is defined geometrically by the (x, y, z) coordinates of the four corner nodes. Although coordinates can be specified at the midside nodes, they are ignored and the edges treated as straight lines.

Due to the bilinear interpolation, the surface forms a hyperbolic paraboloid which is allowed to degenerate to a plate. The element thickness is specified in the GEOMETRY option. The stress-strain output is given in local orthogonal surface directions (V_1, V_2, and V_3) which, for the centroid, are defined in the following way (see Figure 5-118).

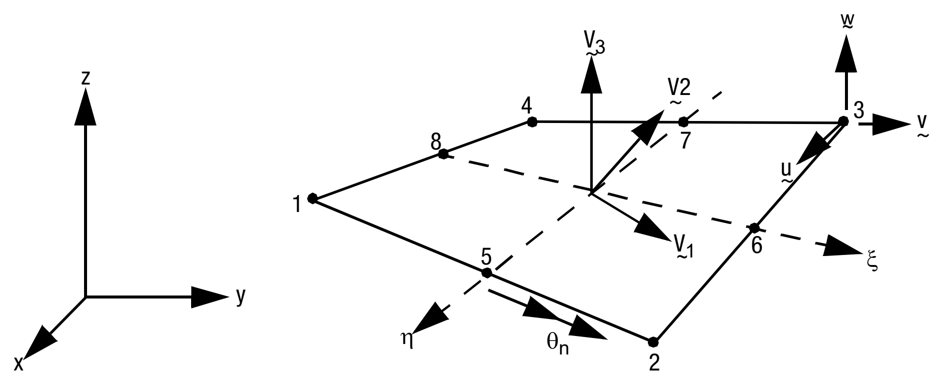


Figure 5-118 Bilinear Constrained Shell Element 72

First, the vectors tangent to the curves with constant isoparametric coordinates ξ and η are normalized:

$$t_1 = \frac{\partial x}{\partial \xi} \Big/ \left|\frac{\partial x}{\partial \xi}\right|, \quad t_2 = \frac{\partial x}{\partial \eta} \Big/ \left|\frac{\partial x}{\partial \eta}\right|$$

Now a new basis is being defined as:

$$s = t_1 + t_2, \quad d = t_1 - t_2$$

After normalizing these vectors by:

$$\bar{s} = s / \sqrt{2}\,|s|, \quad \bar{d} = d / \sqrt{2}\,|d|$$

The local orthogonal directions are then obtained as:

$$V_1 = \bar{s} + \bar{d}, V_2 = \bar{s} - \bar{d}, \text{and } V_3 = V_1 \times V_2$$

In this way, the vectors $\frac{\partial x}{\partial \xi}, \frac{\partial x}{\partial \eta}$ and V_1, V_2 have the same bisecting plane.

The local directions for the Gaussian integrations points are found by projection of the centroid directions. Hence, if the element is flat, the directions at the Gauss points are identical to those at the centroid.

Displacements

The nodal displacement variables are as follows:

At the four corner nodes:	u, v, w Cartesian displacement components.
At the four midside nodes:	ϕ_n, rotation of the element edge about itself. The positive rotation vector points from the corner with the lower (external) node number to the corner with the higher (external) node number.

Due to the ease in modeling intersecting plates (pin joint or moment carrying joint), no special tying types have been developed for this element type.

Connectivity Specification

The four corner nodes of the element are input first, proceeding continuously around the element edges. Then the node between corners 1 and 2 is given, followed by the midside nodes between corners 2 and 3, 3 and 4, and 4 and 1.

In case the element is degenerated to a triangle by collapsing one of the sides, the midside node on the collapsed side no longer has any stiffness associated with it. The midside node is given the same node number as the two corner nodes. Hence, the connectivity of the triangle in Figure 5-119 can, for instance, be specified as:

1, 72, 1, 2, 3, 3, 4, 5, 3, 6,

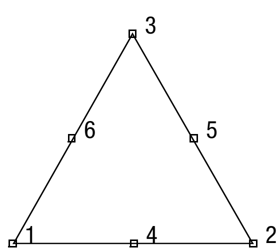


Figure 5-119 Collapsed Shell Element Type 72

Quick Reference

Type 72

Bilinear, constrained eight-node shell element.

Connectivity

Eight nodes: corners nodes given first, proceeding continually around the element.

Then the midside nodes are given as:

5 = between corners 1 and 2
6 = between corners 2 and 3
7 = between corners 3 and 4
8 = between corners 4 and 1

The element can be collapsed to a triangle. The midside node on the collapsed edge has no associated stiffness.

Geometry

Bilinear thickness variation is allowed in the plane of the element. Thicknesses at first, second, third and fourth nodes of the element are stored for each element in the first (`EGEOM1`), second (`EGEOM2`), third (`EGEOM3`) and fourth (`EGEOM4`), geometry data fields, respectively. If `EGEOM2` = `EGEOM3` = `EGEOM4` = 0, then a constant thickness (`EGEOM1`) is assumed for the element.

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

Coordinates

(x, y, z) global Cartesian coordinates are given. Note that since the element is assumed to have straight edges, the coordinates associated with the midside nodes are not required by Marc.

Degrees of Freedom

At four corner nodes:

1 = u = global Cartesian x-direction displacement
2 = v = global Cartesian y-direction displacement
3 = w = global Cartesian z-direction displacement

At midside nodes:

1 = ϕ_n =	rotation of edge about itself. The positive rotation vector points from the corner with the lower (external) node number to the corner with the higher external node number.

Tractions

Types of distributed loading are as follows:

Load Type	Direction
1	Uniform gravity load per surface area in -z-direction.
2	Uniform pressure with positive magnitude in -V_3-direction.

Load Type	Direction
3	Nonuniform gravity load per surface area in -z-direction.
4	Nonuniform pressure with positive magnitude in -V_3-direction.
5	Nonuniform load per surface area in arbitrary direction.
11	Uniform edge load in the plane of the surface on the 1-2 edge and perpendicular to the edge.
12	Nonuniform edge load magnitude given in the FORCEM user subroutine in the plane of the surface on the 1-2 edge.
13	Nonuniform edge load magnitude and direction given in the FORCEM user subroutine on the 1-2 edge.
14	Uniform load on the 1-2 edge of the shell, tangent to, and in the direction of the 1-2 edge
15	Uniform load on the 1-2 edge of the shell. Perpendicular to the shell; i.e., -v_3 direction
21	Uniform edge load in the plane of the surface on the 2-3 edge and perpendicular to the edge.
22	Nonuniform edge load magnitude given in the FORCEM user subroutine in the plane of the surface on the 2-3 edge.
23	Nonuniform edge load magnitude and direction given in the FORCEM user subroutine on the 2-3 edge.
24	Uniform load on the 2-3 edge of the shell, tangent to and in the direction of the 2-3 edge
25	Uniform load on the 2-3 edge of the shell. Perpendicular to the shell; i.e., -v_3 direction
31	Uniform edge load in the plane of the surface on the 3-4 edge and perpendicular to the edge.
32	Nonuniform edge load magnitude given in the FORCEM user subroutine in the plane of the surface on the 3-4 edge.
33	Nonuniform edge load magnitude and direction given in the FORCEM user subroutine on the 3-4 edge.
34	Uniform load on the 3-4 edge of the shell, tangent to and in the direction of the 3-4 edge
35	Uniform load on the 3-4 edge of the shell. Perpendicular to the shell; i.e., -v_3 direction
41	Uniform edge load in the plane of the surface on the 4-1 edge and perpendicular to the edge.
42	Nonuniform edge load magnitude given in the FORCEM user subroutine in the plane of the surface on the 4-1 edge.
43	Nonuniform edge load magnitude and direction given in the FORCEM user subroutine on the 4-1 edge.
44	Uniform load on the 4-1 edge of the shell, tangent to and in the direction of the 4-1 edge
45	Uniform load on the 4-1 edge of the shell. Perpendicular to the shell; i.e., -v_3 direction
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.

Load Type	Direction
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

All edge loads require the input as force per unit length.

Point loads and moments can also be applied at the nodes.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Generalized strain components are:

Middle surface stretches ε_{11} ε_{22} ε_{12}
Middle surface curvatures κ_{11} κ_{22} κ_{12}

in local (V_1, V_2, V_3) system.

Output of Stresses

σ_{11}, σ_{22}, σ_{12} in local V_1, V_2, V_3, system given at equally spaced layers through the thickness. First layer is on positive V_3 direction surface.

Transformation

Displacement components at corner nodes can be transformed to local direction.

The TRANSFORMATION option should not be invoked on the midside nodes.

Tying

Use the UFORMSN user subroutine.

Output Points

If the CENTROID parameter is used, output occurs at the centroid of the element, given as point 5 in Figure 5-120. If the ALL POINTS parameter is used, the output is given in the first four points as shown in Figure 5-120.

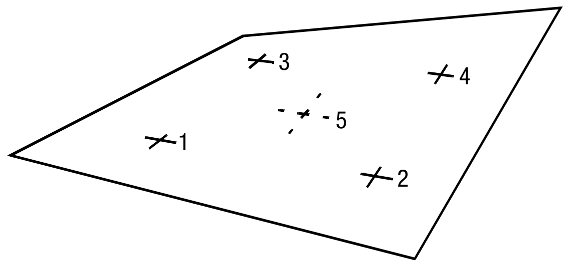


Figure 5-120 Integration Points for Element 72

Section Stress Integration

Integration through the shell thickness is performed numerically using Simpson's rule.

Use the SHELL SECT parameter to specify the number of integration points. This number must be odd. Three points are enough for linear response; seven points are enough for simple plasticity or creep analysis; eleven points are enough for complex plasticity or creep (for example, dynamic plasticity). The default is 11 points.

Beam Stiffeners

The element is fully compatible with beam element types 76 and 77.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – output for stresses and strains as for total Lagrangian approach.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 85. See Element 85 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered as a design variable.

Element 73

Axisymmetric, Eight-node Quadrilateral for Arbitrary Loading with Reduced Integration (Fourier)

Element type 73 is an eight-node, isoparametric, arbitrary quadrilateral written for axisymmetric geometries with arbitrary loading. This allows for variation in the response in the circumferential direction by decomposing the behavior using Fourier series.

This element uses biquadratic interpolation functions to represent the coordinates and displacements. Hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses.

The stiffness of this element is formed using four-point Gaussian integration. This is a reduced integration element – which may exhibit hourglass modes. This element should be used with caution. The mass matrix of this element is formed using nine-point Gaussian integration.

This Fourier element can only be used for linear elastic analyses. No contact is permitted with this element.

Quick Reference

Type 73

Second-order, isoparametric, distorted quadrilateral, for arbitrary loading of axisymmetric solids, formulated by means of the Fourier expansion technique, using reduced integration.

Connectivity

Corner numbered first in counterclockwise order (right-handed convention). The fifth node between first and second; the sixth node between second and third, etc. See Figure 5-121.

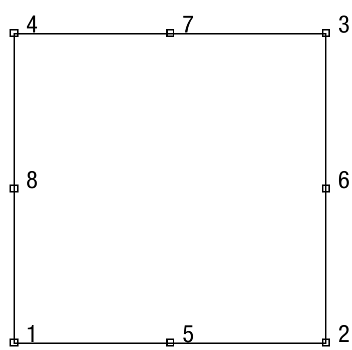


Figure 5-121 Nodal Numbering of Element 73

Geometry

No geometry input for this element.

Coordinates

Two global coordinates, z and r, at each node.

Degrees of Freedom

Three at each node:

1 = u = displacement in axial (z) direction
2 = v = displacement in radial (r) direction
3 = ϕ = circumferential displacement (θ) direction

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
6	Uniform shear force in direction 1⇒5⇒2 on 1-5-2 face.
7	Nonuniform shear force in direction 1⇒5⇒2 on 1-5-2 face.
8	Uniform pressure on 2-6-3.
9	Nonuniform pressure on 2-6-3.
10	Uniform pressure on 3-7-4.
11	Nonuniform pressure on 3-7-4.
12	Uniform pressure on 4-8-1 face.
13	Nonuniform pressure on 4-8-1 face.
14	Uniform shear in θ-direction (torsion) on 1-5-2 face.
15	Nonuniform shear in θ-direction (torsion) on 1-5-2 face.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

Body forces (per unit volume). Load type 2 is uniform body force in the z-direction (axial); load type 3 is nonuniform body force in the z-direction. Load type 4 is uniform body force in the r-direction (radial); load type 5 is nonuniform body force in the r-direction. For load types 3 and 5, the FORCEM user subroutine must supply the force magnitude.

Concentrated nodal loads must be the value of the load integrated around the circumference.

For varying load magnitudes in the θ-direction, each distributed load or concentrated force can be associated with a different Fourier expansion. If no Fourier series is specified for a given loading, it is assumed to be constant around the circumference.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid or element integration points (see Figure 5-122 and Output Points) in the following order.

1 = ε_{zz}, direct
2 = ε_{rr}, direct
3 = $\varepsilon_{\theta\theta}$, direct
4 = γ_{zr}, in-plane shear
5 = $\gamma_{r\theta}$, out-of-plane shear
6 = $\gamma_{\theta z}$, out-of-plane shear

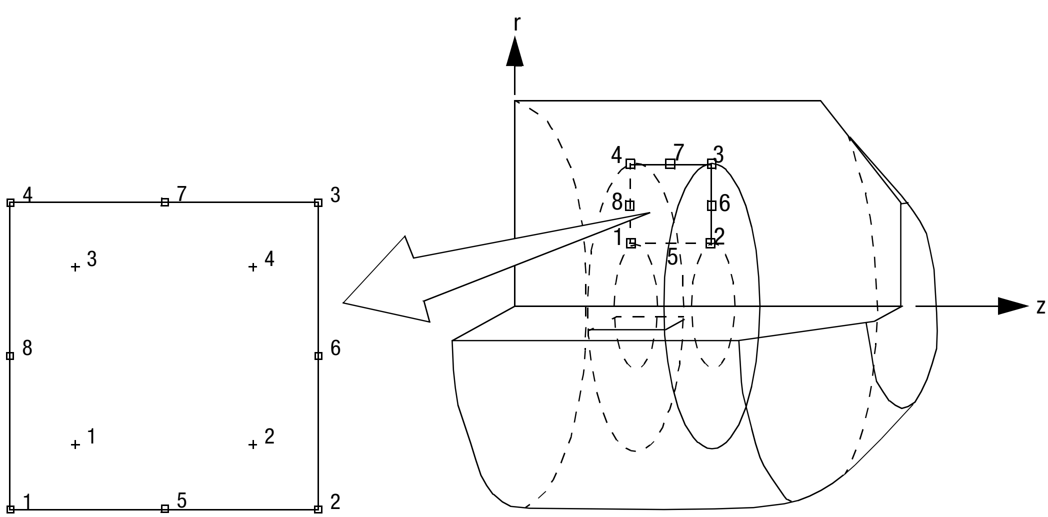


Figure 5-122 Integration Points of Eight-node 2-D Element

Output of Stresses

Same as for Output of Strains .

Transformation

Only in z-r plane.

Tying

Use the UFORMSN user subroutine.

Output Points

If the CENTROID parameter is used, the output occurs at the centroid of the element.

If the ALL POINTS parameter is used, the output is given for all four integration points.

Element 74

Axisymmetric, Eight-node Distorted Quadrilateral for Arbitrary Loading, Herrmann Formulation, with Reduced Integration (Fourier)

Element type 74 is an eight-node, isoparametric, arbitrary quadrilateral written for incompressible axisymmetric geometries with arbitrary loading. This allows for variation in the response in the circumferential direction by decomposing the behavior using Fourier series.

This element uses biquadratic interpolation functions to represent the coordinates and displacements. Hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is represented using bilinear interpolation functions based upon the extra degree of freedom at the corner nodes.

The stiffness of this element is formed using four-point Gaussian integration. This is a reduced integration element – which may exhibit hourglass modes. This element should be used with caution. The mass matrix of this element is formed using nine-point Gaussian integration.

This Fourier element can only be used for linear elastic analyses. No contact is permitted with this element.

Quick Reference

Type 74

Second-order, isoparametric quadrilateral for arbitrary loading of axisymmetric solids, formulated by means of the Fourier expansion technique. Mixed formulation for incompressible or nearly incompressible materials.

Connectivity

Corners numbered first in counterclockwise order. Then fifth node between first and second; the sixth node between second and third, etc. See Figure 5-123.

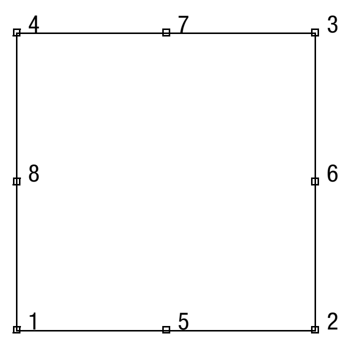


Figure 5-123 Eight-node Axisymmetric, Herrmann Fourier Element

Geometry

No geometry input for this element.

Coordinates

Two global coordinates, z and r, at each node.

Degrees of Freedom

1 = u = displacement in axial (z) direction
2 = v = displacement in radial (r) direction
3 = ϕ = circumferential displacement (j) direction.

Additional degree of freedom at each corner node:

4 = σ_{kk}/E	= mean pressure variable (isotropic)
= formula below (orthotropic)	

$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1+\frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2+\frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3\right]\frac{9}{E_1+E_2+E_3}$$

= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)
= p / K	(for Herrmann elements using additive decomposition; K here is the effective bulk modulus)

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
6	Uniform shear force in direction 1⇒5⇒2 on 1-5-2 face.
7	Nonuniform shear force in direction 1⇒5⇒2 on 1-5-2 face.
8	Uniform pressure on 2-6-3.
9	Nonuniform pressure on 2-6-3.
10	Uniform pressure on 3-7-4.
11	Nonuniform pressure on 3-7-4.
12	Uniform pressure on 4-8-1 face.
13	Nonuniform pressure on 4-8-1 face.
14	Uniform shear in θ-direction (torsion) on 1-5-2 face.
15	Nonuniform shear in θ-direction (torsion) on 1-5-2 face.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x-, y-, z-direction.

Load Type (IBODY)	Description
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

Body forces (per unit volume). Load type 2 is uniform body force in the z-direction (axial); load type 3 is non-uniform body force in the z-direction. Load type 4 is uniform body force in the r-direction (radial); load type 5 is nonuniform body force in the r-direction. For load types 3 and 5, the FORCEM user subroutine must supply the force magnitude.

Concentrated nodal loads must be the value of the load integrated around the circumference.

For varying load magnitudes in the θ-direction, each distributed load or concentrated force can be associated with a different Fourier expansion. If no Fourier series is specified for a given loading, it is assumed to be constant around the circumference.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid or element integration points (see Figure 5-124 and Output Points) in the following order:

1	$= \varepsilon_{zz}$	direct
2	$= \varepsilon_{rr}$	direct
3	$= \varepsilon_{\theta\theta}$	direct
4	$= \gamma_{zr}$	in-plane shear
5	$= \gamma_{z\theta}$	out-of-plane shear
6	$= \gamma_{\theta z}$	out-of-plane shear
7	$= \sigma_{kk}/E$	= mean pressure variable (isotropic)
	= formula below (orthotropic)	

$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1 + \frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\Delta_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2 + \frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3 \right] \frac{9}{E_1 + E_2 + E_3}$$

	= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)

The last component is not printed in the output file, but is accessible via the ELEVAR and PLOTV user subroutines.

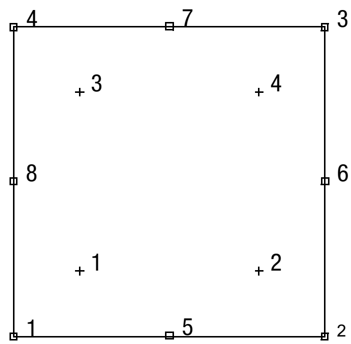


Figure 5-124 Integration Points for Reduced Integration Element

Output of Stresses

Same as for Output of Strains .

Transformation

Only in z-r plane.

Tying

Use the UFORMSN user subroutine.

Output Points

If the CENTROID parameter is used, the output occurs at the centroid of the element.

If the ALL POINTS parameter is used, the output is given for all four integration points.

Element 75

Bilinear Thick-shell Element

This is a four-node, thick-shell element with global displacements and rotations as degrees of freedom. Bilinear interpolation is used for the coordinates, displacements and the rotations. The membrane strains are obtained from the displacement field; the curvatures from the rotation field. The transverse shear strains are calculated at the middle of the edges and interpolated to the integration points. In this way, a very efficient and simple element is obtained which exhibits correct behavior in the limiting case of thin shells. The element can be used in curved shell analysis as well as in the analysis of complicated plate structures. For the latter case, the element is easy to use since connections between intersecting plates can be modeled without tying.

Due to its simple formulation when compared to the standard higher order shell elements, it is less expensive and, therefore, very attractive in nonlinear analysis. The element is not very sensitive to distortion, particularly if the corner nodes lie in the same plane. All constitutive relations can be used with this element.

Geometric Basis

The element is defined geometrically by the (x,y,z) coordinates of the four corner nodes. Due to the bilinear interpolation, the surface forms a hyperbolic paraboloid which is allowed to degenerate to a plate. The element thickness is specified in the GEOMETRY option.

The stress output is given in local orthogonal surface directions (V_1, V_2, and V_3) which, for the centroid, are defined in the following way (see Figure 5-125).

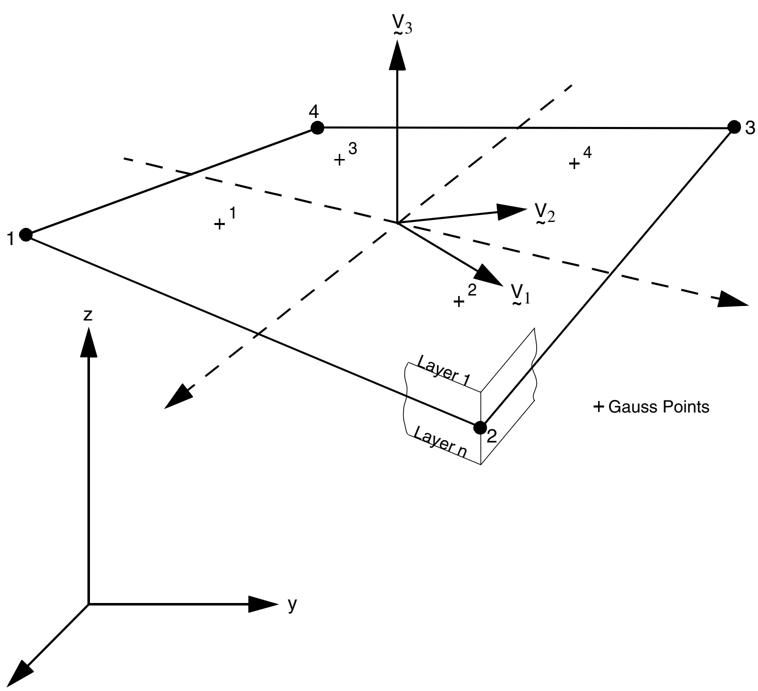


Figure 5-125 Form of Element 75

At the centroid, the vectors tangent to the curves with constant isoparametric coordinates are normalized.

$$t_1 = \frac{\partial x}{\partial \xi} / \left|\frac{\partial x}{\partial \xi}\right|, \quad t_2 = \frac{\partial x}{\partial \eta} / \left|\frac{\partial x}{\partial \eta}\right|$$

Now a new basis is being defined as:

$$s = t_1 + t_2, \quad d = t_1 - t_2$$

After normalizing these vectors by:

$$\bar{s} = s / \sqrt{2}|s| \quad \bar{d} = d / \sqrt{2}|d|$$

The local orthogonal directions are then obtained as:

$$\underset{\sim}{V}_1 = \bar{s} + \bar{d}, \ \underset{\sim}{V}_2 = \bar{s} - \bar{d} \text{, and } \underset{\sim}{V}_3 = \underset{\sim}{V}_1 \times \underset{\sim}{V}_2$$

In this way, the vectors $\frac{\partial x}{\partial \xi}$, $\frac{\partial x}{\partial \eta}$ and $\underset{\sim}{V}_1$, $\underset{\sim}{V}_2$ have the same bisecting plane.

The local directions at the Gaussian integration points are found by projection of the centroid directions. Hence, if the element is flat, the directions at the Gauss points are identical to those at the centroid.

Displacements

The six nodal displacement variables are as follows:

u, v, w	Displacement components defined in global Cartesian x,y,z coordinate system.
ϕ_x, ϕ_y, ϕ_z	Rotation components about global x-, y-, and z-axis, respectively.

Quick Reference

Type 75

Bilinear, four-node shell element including transverse shear effects.

Connectivity

Four nodes per element. The element can be collapsed to a triangle.

Geometry

Bilinear thickness variation is allowed in the plane of the element. Thicknesses at first, second, third and fourth nodes of the element are stored for each element in the first (`EGEOM1`), second (`EGEOM2`), third (`EGEOM3`) and fourth (`EGEOM4`), geometry data fields, respectively. If `EGEOM2 = EGEOM3 = EGEOM4 = 0`, then a constant thickness (`EGEOM1`) is assumed for the element.

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

If the element needs to be offset from the user-specified position, the eighth data field (`EGEOM8`) is set to -2.0. In this case, an extra line containing the offset information is provided (see the GEOMETRY option in Marc Volume C: Program Input). The offset magnitudes along the element normal for the four corner nodes are provided in the first, second, third, and fourth data fields of the extra line. A uniform offset for all nodes can be set by providing the offset magnitude in the first data field and then setting the constant offset flag (sixth data field of the extra line) to 1.

Coordinates

Three coordinates per node in the global x-, y-, and z-directions.

Degrees of Freedom

Six degrees of freedom per node:

1 = u = global (Cartesian) x-displacement
2 = v = global (Cartesian) y-displacement
3 = w = global (Cartesian) z-displacement
4 = ϕ_x = rotation about global x-axis
5 = ϕ_y = rotation about global y-axis
6 = ϕ_z = rotation about global z-axis

Distributed Loads

A table of distributed loads is listed below:

Load Type	Description
1	Uniform gravity load per surface area in -z-direction.
2	Uniform pressure with positive magnitude in -V_3-direction.
3	Nonuniform gravity load per surface area in -z-direction, magnitude given in the FORCEM user subroutine.
4	Nonuniform pressure with positive magnitude in -V_3-direction, magnitude given in the FORCEM user subroutine.
5	Nonuniform load per surface area in arbitrary direction, magnitude given in the FORCEM user subroutine.
11	Uniform edge load in the plane of the surface on the 1-2 edge and perpendicular to this edge.
12	Nonuniform edge load magnitude given in the FORCEM user subroutine in the plane of the surface on the 1-2 edge.
13	Nonuniform edge load magnitude and direction given in the FORCEM user subroutine on 1-2 edge.
14	Uniform load on the 1-2 edge of the shell, tangent to, and in the direction of the 1-2 edge
15	Uniform load on the 1-2 edge of the shell. Perpendicular to the shell; i.e., -v_3 direction
21	Uniform edge load in the plane of the surface on the 2-3 edge and perpendicular to this edge.
22	Nonuniform edge load magnitude given in the FORCEM user subroutine in the plane of the surface on the 2-3 edge.
23	Nonuniform edge load magnitude and direction given in the FORCEM user subroutine on 2-3 edge.
24	Uniform load on the 2-3 edge of the shell, tangent to and in the direction of the 2-3 edge

Load Type	Description
25	Uniform load on the 2-3 edge of the shell. Perpendicular to the shell; i.e., $-v_3$ direction
31	Uniform edge load in the plane of the surface on the 3-4 edge and perpendicular to this edge.
32	Nonuniform edge load magnitude given in the FORCEM user subroutine in the plane of the surface on the 3-4 edge.
33	Nonuniform edge load magnitude and direction given in the FORCEM user subroutine on 3-4 edge.
34	Uniform load on the 3-4 edge of the shell, tangent to and in the direction of the 3-4 edge
35	Uniform load on the 3-4 edge of the shell. Perpendicular to the shell; i.e., $-v_3$ direction
41	Uniform edge load in the plane of the surface on the 4-1 edge and perpendicular to this edge.
42	Nonuniform edge load magnitude given in the FORCEM user subroutine in the plane of the surface on the 4-1 edge.
43	Nonuniform edge load magnitude and direction given in the FORCEM user subroutine on 4-1 edge.
44	Uniform load on the 4-1 edge of the shell, tangent to and in the direction of the 4-1 edge
45	Uniform load on the 4-1 edge of the shell. Perpendicular to the shell; i.e., $-v_3$ direction
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

All edge loads require the input as force per unit length.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Point Loads

Point loads and moments can also be applied at the nodes.

Output Of Strains

Generalized strain components are:

Middle surface stretches:	ε_{11} ε_{22} ε_{12}
Middle surface curvatures:	κ_{11} κ_{22} κ_{12}
Transverse shear strains:	γ_{23} γ_{31}

in local $(\underline{V}_1, \underline{V}_2, \underline{V}_3)$ system.

Output Of Stresses

σ_{11}, σ_{22}, σ_{12}, σ_{23}, σ_{31} in local $(\underline{V}_1, \underline{V}_2, \underline{V}_3)$ system given at equally spaced layers though thickness. First layer is on positive V_3 direction surface.

Transformation

Displacement and rotation at corner nodes can be transformed to local direction.

Tying

Use the UFORMSN user subroutine

Updated Lagrange Procedure and Finite Strain Plasticity

Updated Lagrange capability is available. Note, however, that since the curvature calculation is linearized, you have to select your load steps such that the rotation remains small within a load step.

Section Stress - Integration

Integration through the shell thickness is performed numerically using Simpson's rule. Use the SHELL SECT parameter to specify the number of integration points. This number must be odd. Seven points are enough for simple plasticity or creep analysis. Eleven points are enough for complex plasticity or creep (e.q., thermal plasticity). The default is 11 points.

Beam Stiffeners

The element is fully compatible with open- and closed-section beam element types 78 and 79.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 85. See Element 85 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered as a design variable.

Element 76

Thin-walled Beam in Three Dimensions without Warping

This is a simple straight beam element with no warping of the section, but including twist. The default cross section is a thin-walled circular closed section beam; you can specify alternative closed cross sections through the BEAM SECT parameter. The large displacement formulation only affects the axial strain; large curvature and twist changes are neglected.

The degrees of freedom associated with the end nodes are three global displacements and three global rotations, all defined in a right-handed convention. The midnode has only one degree of freedom, rotation along the beam axis, to be compatible with shell element 49 or 72. The generalized strains are stretch, two curvatures, and twist. Stresses are direct (axial) and shear given at each point of the cross section. The local coordinate system which establishes the positions of those points on the section is defined by the fourth, fifth and sixth coordinates at the end nodes, which give the (x,y,z) global Cartesian coordinates of a point in space which locates the local x-axis of the cross-section. This axis lies in the plane defined by the beam nodes and this point, pointing from the beam, toward this point. Alternatively, this point can also be specified by the fourth, fifth, and sixth entry on the GEOMETRY option if the local coordinate system is constant over the element. The local z-axis is along the beam from the first to the second node, and the local y-axis forms a right-handed set with the local x and local z.

For other than the default (circular) section, the stress points are defined by you in the local x-y set through the BEAM SECT parameter set. For the circular hollow section, `EGEOM1` is the wall thickness and `EGEOM2` is the radius. Otherwise, `EGEOM2` gives the section choice from the BEAM SECT input. Section properties are obtained by numerical integration over the stress points of the section. The default is a circular cross section.

All constitutive relations can be used with this element.

Notes:	For noncircular sections, the BEAM SECT parameter must be used to describe the section.
	For all beam elements (13, 14, 25, 52, 76, 77, 78, 79, and 98), the default printout gives section forces and moments, plus stress at any layer with plastic, creep strain or nonzero temperature. This default printout can be changed via the PRINT ELEMENT option.
	This element can be used in combination with shell element 72 and open-section beam element 77 to model stiffened shell structures. No tyings are necessary in that case.

Quick Reference

Type 76

Closed-section beam, Euler-Bernoulli theory.

Connectivity

Three nodes per element. (Nodes 1 and 3 are the end nodes. See Figure 5-126)

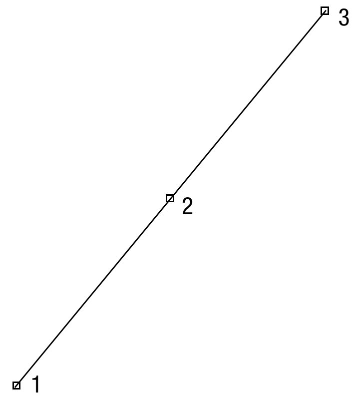


Figure 5-126 Closed-section Beam

Geometry

In the default section of a hollow circular cylinder, the first data field is for the thickness (EGEOM1).

For noncircular section, set EGEOM1 to 0.

For circular section, set EGEOM2 to radius.

For noncircular section, set EGEOM2 to the section number needed. (Sections are defined using the BEAM SECT parameter.)

Optional specification of cross-section direction with EGEOM4, EGEOM5, and EGEOM6 (see Coordinates).

If beam-to-beam contact is switched on (see CONTACT option), the radius used when the element comes in contact with other beam or truss elements must be entered in the seventh data field (EGEOM7).

For segment-segment beam contact, various flags that control the creation of patches are entered in the seventh data file (EGEOM7).

```
EGEOM7 = real(100 * NSEG + 10 * IPTCH + IESCAP)
```

where

NSEG = number of segments for hollow tubes (default = 32)		
IPTCH = patch flag for hollow tubes		
	1	Both outer and inner patches (default).
	3	Only outer patches (at radius + 1/2 * thickness).
	-3	Only outer patches (at radius - i.e., ignore thickness).
	5	Only inner patches (at radius - 1/2 * thickness).
	-5	Only inner patches (at radius - i.e., ignore thickness).
IESCAP = end cap/side cap flag		
	0	No end or side cap (default).

	1	End cap only.
	2	Side cap only.
	3	Average side only.
	4	End and side cap.
	5	End cap and average side.

It should be noted that the contact patches are created between the two end nodes of the beam and the midside node is ignored for the expanded beam representation.

Beam Offsets Pin Codes and Local Beam Orientation

If an offset is applied to the beam geometry or pin codes are applied to the degrees of freedom of the end point of the beams or the beam axis is defined with respect to a local coordinate system, then a code is placed in the 8th field of the 3rd data block of the GEOMETRY option (`EGEOM8`). This code is formed by the negative sum of ioffset, iorien, and ipin where:

ioffset	=	0	-	no offset
		1	-	beam offset
iorien	=	0	-	conventional definition of local beam orientation; beam axis given in 4th through 6th field with respect to global system
		10	-	the local beam orientation entered in the 4th through 6th field is given with respect to the coordinate system of the first beam node. This vector is transformed to the global coordinate system before any further processing.
ipin	=	0	-	no pin codes are used
		100	-	pin codes are used.

If ioffset = 1, an additional line is read in the GEOMETRY option (4th data block).

If ipin = 100, an additional line is read in the GEOMETRY option (5th data block)

The offset vector at the first corner node is provided in the first three fields and the offset vector at the second corner node is provided in the next 3 three fields of the extra line. The coordinate system used for the offset vectors at the two nodes are indicated by the corresponding flags in the eighth and ninth fields of the extra line. A flag setting of 0 indicates global coordinate system, 1 indicates local element coordinate system, 2 indicates local shell normal, and 3 indicates local nodal coordinate system. When the flag setting is 1, the local beam coordinate system is constructed by taking the local z axis along the beam, the local x axis as the user-specified vector (obtained from EGEOM4-EGEOM6) and the local y axis as perpendicular to both. The flag setting of 2 can be used when the beam nodes are also attached to a shell. In this case, the offset magnitude at the first and second nodes are specified in the first and fourth data fields of the extra line, respectively. The automatically calculated shell normals at the nodes are used to construct the offset vectors. It should be noted that for nonlinear problems, only the end node rotations are used to update the offset vectors and the midside node rotation is ignored.

Coordinates

Six coordinates at the end nodes. The first three coordinates are global (x,y,z). The fourth, fifth, and sixth coordinates are the global x,y,z coordinates of a point in space which locates the local x-axis of the cross section. The local x-axis is a vector normal to the beam axis through the point described by the fourth, fifth, and sixth coordinates. These coordinates can also be given as EGEOM4, EGEOM5, and EGEOM6 under the GEOMETRY option.

The local x-axis is positive progressing from the beam to the point.

Degrees of Freedom

At End Nodes:	At Middle Node:
1 = u	1 = ϕ_t (rotation around the beam axis; positive from the lower to the higher external end node number.)
2 = v	
3 = w	
4 = ϕ_x	
5 = ϕ_y	
6 = ϕ_z	

Distributed Loads

Distributed load types are as follows:

Load type	Description
1	Uniform load per unit length in global x-direction.
2	Uniform load per unit length in global y-direction.
3	Uniform load per unit length in global z-direction.
4	Nonuniform load per unit length, with magnitude and direction supplied via the FORCEM user subroutine.
11	Fluid drag/buoyancy loading – fluid behavior specified in the FLUID DRAG option.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Load type	Description
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Point Loads

Point loads and moments can be applied at the end nodes; a twisting moment can be applied at the midnode.

Output of Strains

Generalized strains:

1 = ε_{zz}	= axial stretch
2 = κ_{xx}	= curvature about local x-axis of cross section.
3 = κ_{yy}	= curvature about local y-axis of cross section.
4 = γ	= twist

Output of Stresses

Generalized stresses:

1 = axial stress
2 = local xx-moment
3 = local yy-movement
4 = axial torque

Stresses at integration points in the cross section are only printed if explicitly requested or if plasticity is present.

Transformation

Displacement and rotations at the end nodes can be transformed to a local coordinate system. Transformations should not be invoked for the midnode.

Tying

Use tying type 100 for fully moment-carrying joints. Use tying type 103 for pin joints.

Output Points

Centroid or two Gaussian integration points. The first point is near the first node in the Connectivity description of the element. The second point is near the third node in the **Connectivity** description of the element.

Updated Lagrange Procedure and Finite Strain Plasticity

The Updated Lagrange procedure is available for this element, and should be used if the element is subjected to large rotations. Since the cross-sectional dimensions of the element are not updated, the element should not be used for finite strain applications.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 36. See Element 36 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

For the default hollow circular section only, the wall thickness and the radius can be considered as design variables.

Element 77

Thin-walled Beam in Three Dimensions including Warping

This is a simple straight beam element that includes warping and twist of the section. You must specify open cross sections through the BEAM SECT parameter. The section number is given in the GEOMETRY data field. Primary warping effects are included, but twisting is assumed to be elastic. The large displacement formulation only affects the axial strain; large curvature, warping and twist changes are neglected.

The degrees of freedom associated with the end nodes are three global displacements and three global rotations, all defined in a right-handed convention and the warping. The midnode has only one degree of freedom, rotation along the beam axis, to be compatible with shell element 72. The generalized strains are stretch, two curvatures, warping, and twist. Stresses are direct (axial) given at each point of the cross section. The local coordinate system which establishes the positions of those points on the section is defined by the fourth, fifth and sixth coordinates at the end nodes, which give the (x,y,z) global Cartesian coordinates of a point in space which locates the local x-axis of the cross-section. This axis lies in the plane defined by the beam nodes and this point, pointing from the beam, toward this point. Alternatively, this point can also be specified by the fourth, fifth, and sixth entry on the GEOMETRY option if the local coordinate system is constant over the element. The local z-axis is along the beam from the first to the second node, and the local y-axis forms a right-handed set with the local x and local z.

All constitutive relations can be used with this element.

Notes:	The BEAM SECT parameter must be used to describe the section.
	For all beam elements (13, 14, 25, 52, 76, 77, 78, 79, and 98), the default printout gives section forces and moments, plus stress at any layer with plastic, creep strain or nonzero temperature. This default printout can be changed via the PRINT ELEMENT option.
	This element can be used in combination with shell element type 72 and closed-section beam element type 76 to model stiffened shell structures. No tyings are necessary in that case.

Quick Reference

Type 77

Open section beam including warping.

Connectivity

Three nodes per element (see Figure 3-127).

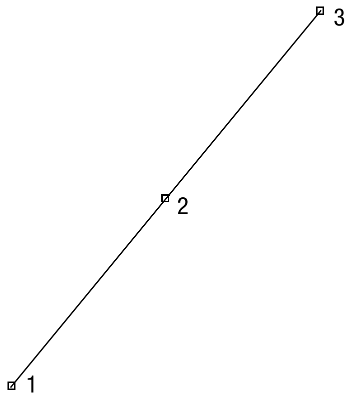


Figure 3-127 Open-section Beam

Geometry

Set EGEOM2 to the section number needed. (Sections are defined using the BEAM SECT parameter.)

Optional specification of cross-section direction with `EGEOM4`, `EGEOM5`, and `EGEOM6` (see Coordinates).

If beam-to-beam contact is switched on (see CONTACT option), the radius used when the element comes in contact with other beam or truss elements must be entered in the 7th data field (`EGEOM7`).

For segment-segment beam contact, various flags that control the creation of patches are entered in the seventh data file (`EGEOM7`).

```
EGEOM7 = real(100 * NSEG + 10 * IPTCH + IESCAP)
```

where

`NSEG` = number of segments for hollow tubes (default = 32)		
`IPTCH` = patch flag for hollow tubes		
	1	Both outer and inner patches (default).
	3	Only outer patches (at radius + 1/2 * thickness).
	-3	Only outer patches (at radius - i.e., ignore thickness).
	5	Only inner patches (at radius - 1/2 * thickness).
	-5	Only inner patches (at radius - i.e., ignore thickness).
`IESCAP` = end cap/side cap flag		
	0	No end or side cap (default).
	1	End cap only.
	2	Side cap only.
	3	Average side only.
	4	End and side cap.
	5	End cap and average side.

It should be noted that the contact patches are created between the two end nodes of the beam and the midside node is ignored for the expanded beam representation.

Beam Offsets Pin Codes and Local Beam Orientation

If an offset is applied to the beam geometry or pin codes are applied to the degrees of freedom of the end point of the beams or the beam axis is defined with respect to a local coordinate system, then a code is placed in the 8th field of the 3rd data block of the GEOMETRY option (`EGEOM8`). This code is formed by the negative sum of ioffset, iorien, and ipin where:

ioffset	=	0	-	no offset
		1	-	beam offset
iorien	=	0	-	conventional definition of local beam orientation; beam axis given in 4th through 6th field with respect to global system
		10	-	the local beam orientation entered in the 4th through 6th field is given with respect to the coordinate system of the first beam node. This vector is transformed to the global coordinate system before any further processing.
ipin	=	0	-	no pin codes are used
		100	-	pin codes are used.

If ioffset = 1, an additional line is read in the GEOMETRY option (4th data block).

If ipin = 100, an additional line is read in the GEOMETRY option (5th data block)

The offset vector at the first corner node is provided in the first three fields and the offset vector at the second corner node is provided in the next three fields of the extra line. The coordinate system used for the offset vectors at the two nodes are indicated by the corresponding flags in the eighth and ninth fields of the extra line. A flag setting of 0 indicates global coordinate system, 1 indicates local element coordinate system, 2 indicates local shell normal, and 3 indicates local nodal coordinate system. When the flag setting is 1, the local beam coordinate system is constructed by taking the local z axis along the beam, the local x axis as the user specified vector (obtained from `EGEOM4`-`EGEOM6`) and the local y axis as perpendicular to both. The flag setting of 2 can be used when the beam nodes are also attached to a shell. In this case, the offset magnitude at the first and second nodes are specified in the first and fourth data fields of the extra line respectively and the automatically calculated shell normals at the nodes are used to construct the offset vectors. It should be noted that for nonlinear problems, only the end node rotations are used to update the offset vectors and the midside node rotation is ignored.

Coordinates

Six coordinates at the end nodes; the first three are global (x,y,z), the fourth, fifth and sixth are the global x,y,z coordinates of a point in space which locates the local x-axis of the cross section. The local x-axis is a vector normal to the beam axis through the point described by the fourth, fifth, and sixth coordinates. These coordinates can also be given as `EGEOM4`, `EGEOM5`, and `EGEOM6` under the GEOMETRY option.

The local x-axis is positive progressing from the beam to the point.

Degrees of Freedom

At End Nodes:	At Middle Node:
1 = u	1 = ϕ_t (rotation around the beam axis; positive from the lower to the higher external end node number.)
2 = v	
3 = w	
4 = ϕ_x	
5 = ϕ_y	
6 = ϕ_z	
7 = η = warping degree of freedom	

Distributed Loads

Distributed load types as follows:

Load Type	Description
1	Uniform load per unit length in global x-direction.
2	Uniform load per unit length in global y-direction.
3	Uniform load per unit length in global z-direction.
4	Nonuniform load per unit length; magnitude and direction supplied via the FORCEM user subroutine.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Point Loads

Point loads and moments can be applied at the end nodes; a twisting moment can be applied at the midnode.

Output of Strains

Generalized strains:

$1 = \varepsilon_{xx}$	= axial stretch
$2 = \kappa_{xx}$	= curvature about local x-axis of cross section.
$3 = \kappa_{yy}$	= curvature about local y-axis of cross section.
$4 = \eta$	= warping
$5 = \gamma$	= twist

Output of Stresses

Generalized stresses:

1 = axial stress
2 = local xx-moment
3 = local yy-movement
4 = bimoment
5 = axial torque

Stresses at integration points in the cross section are only printed if explicitly requested or if plasticity is present.

Transformation

Displacement and rotations at the end nodes can be transformed to a local coordinate reference.

Tying

Use tying type 100 for fully moment-carrying joints. Use tying type 103 for pin joints.

Output Points

Centroid or two Gaussian integration points. The first point is near the first node in the connectivity description of the element. The second point is near the third node in the connectivity description of the element.

Updated Lagrange Procedure and Finite Strain Plasticity

The Updated Lagrange procedure is available for this element, and should be used if the element is subjected to large rotations. Since the cross-sectional dimensions of the element are not updated, the element should not be used for finite strain applications.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 36. See Element 36 for a description of the conventions used for entering the flux and film data for this element.

Element 78

Thin-walled Beam in Three Dimensions without Warping

This is a simple, straight beam element with no warping of the section, but includes twist. The default cross section is a thin-walled, circular, closed-section beam. You can specify alternative closed cross sections through the BEAM SECT parameter. The large displacement formulation only affects the axial strain; large curvature and twist changes are neglected.

The degrees of freedom associated with each node are three global displacements and three global rotations, all defined in a right-handed convention. The generalized strains are stretch, two curvatures, and twist. Stresses are direct (axial) and shear given at each point of the cross section. The local coordinate system which establishes the positions of those points on the section is defined by the fourth, fifth and sixth coordinates at each node, which give the (x,y,z) global Cartesian coordinates of a point in space which locates the local x-axis of the cross section. This axis lies in the plane defined by the beam nodes and this point, pointing from the beam, toward this point. This point can also be specified by the fourth, fifth and sixth entry on the GEOMETRY option if the local coordinate system is constant over the element. The local z-axis is along the beam from the first to the second node, and the local y-axis forms a right-handed set with the local x and local z.

For other than the default (circular) section, the stress points are defined by you in the local x-y set through the BEAM SECT parameter. For the circular hollow section, `EGEOM1` is the wall thickness, `EGEOM2` is the radius. Otherwise, `EGEOM2` gives the section choice from the BEAM SECT input. Section properties are obtained by numerical integration over the stress points of the section. The default is a circular cross section.

All constitutive relations can be used with this element.

Notes:	For noncircular sections, the BEAM SECT parameter must be used to describe the section.
	For all beam elements (13, 14, 25, 52, 76, 77, 78, 79, and 98), the default printout gives section forces and moments, plus stress at any layer with plastic, creep strain or nonzero temperature. This default printout can be changed via the PRINT ELEMENT option.
	This element can be used in combination with shell element type 75 and open-section beam element type 79 to model stiffened shell structures. No tyings are necessary in that case.

Quick Reference

Type 78

Closed-section beam, Euler-Bernoulli theory.

Connectivity

Two nodes per element (see Figure 3-128).

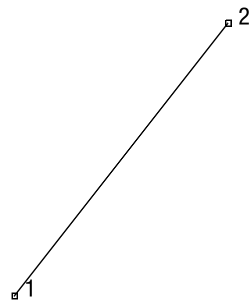


Figure 3-128 Closed-section Beam

Geometry

In the default section of a hollow, circular cylinder, the first data field is for the thickness (EGEOM1).

For noncircular section, set EGEOM1 to 0.

For circular section, set EGEOM2 to radius.

For noncircular section, set EGEOM2 to the section number needed. (Sections are defined using the BEAM SECT parameter.)

Optional specification of cross-section direction with EGEOM4, EGEOM5, and EGEOM6 (see Coordinates).

If beam-to-beam contact is switched on (see CONTACT option), the radius used when the element comes in contact with other beam or truss elements must be entered in the 7th data field (EGEOM7).

For segment-segment beam contact, various flags that control the creation of patches are entered in the seventh data file (EGEOM7).

```
EGEOM7 = real(100 * NSEG + 10 * IPTCH + IESCAP)
```

where

NSEG = number of segments for hollow tubes (default = 32)		
IPTCH = patch flag for hollow tubes		
	1	Both outer and inner patches (default).
	3	Only outer patches (at radius + 1/2 * thickness).
	-3	Only outer patches (at radius - i.e., ignore thickness).
	5	Only inner patches (at radius - 1/2 * thickness).
	-5	Only inner patches (at radius - i.e., ignore thickness).
IESCAP = end cap/side cap flag		
	0	No end or side cap (default).
	1	End cap only.
	2	Side cap only.
	3	Average side only.

	4	End and side cap.
	5	End cap and average side.

Beam Offsets Pin Codes and Local Beam Orientation

If an offset is applied to the beam geometry or pin codes are applied to the degrees of freedom of the end point of the beams or the beam axis is defined with respect to a local coordinate system, then a code is placed in the 8th field of the 3rd data block of the GEOMETRY option (EGEOM8). This code is formed by the negative sum of ioffset, iorien, and ipin where:

ioffset	=	0	-	no offset
		1	-	beam offset
iorien	=	0	-	conventional definition of local beam orientation; beam axis given in 4th through 6th field with respect to global system
		10	-	the local beam orientation entered in the 4th through 6th field is given with respect to the coordinate system of the first beam node. This vector is transformed to the global coordinate system before any further processing.
ipin	=	0	-	no pin codes are used
		100	-	pin codes are used.

If ioffset = 1, an additional line is read in the GEOMETRY option (4th data block).

If ipin = 100, an additional line is read in the GEOMETRY option (5th data block)

The offset vector at the first corner node is provided in the first three fields and the offset vector at the second corner node is provided in the next three fields of the extra line. The coordinate system used for the offset vectors at the two nodes is indicated by the corresponding flags in the eighth and ninth fields of the extra line. A flag setting of 0 indicates global coordinate system, 1 indicates local element coordinate system, 2 indicates local shell normal, and 3 indicates local nodal coordinate system. When the flag setting is 1, the local beam coordinate system is constructed by taking the local z axis along the beam, the local x axis as the user -specified vector (obtained from EGEOM4-EGEOM6), and the local y axis as perpendicular to both. The flag setting of 2 can be used when the beam nodes are also attached to a shell. In this case, the offset magnitude at the first and second nodes are specified in the first and fourth data fields of the extra line respectively and the automatically calculated shell normals at the nodes are used to construct the offset vectors.

Coordinates

Six coordinates per node. The first three coordinates are global (x,y,z). The fourth, fifth, and sixth coordinates are the global x,y,z coordinates of a point in space which locates the local x-axis of the cross section. The local x-axis is a vector normal to the beam axis through the point described by the fourth, fifth, and sixth coordinates. These coordinates can also be given as EGEOM4, EGEOM5, and EGEOM6 under the GEOMETRY option.

The local x-axis is positive progressing from the beam to the point.

Degrees of Freedom

1 = u
2 = v
3 = w
4 = ϕ_x
5 = ϕ_y
6 = ϕ_z

Distributed Loads

Distributed load types as follows:

Load Type	Description
1	Uniform load per unit length in global x-direction.
2	Uniform load per unit length in global y-direction.
3	Uniform load per unit length in global z-direction.
4	Nonuniform load per unit length, with magnitude and direction supplied via the FORCEM user subroutine.
11	Fluid drag/buoyancy loading – fluid behavior specified in the FLUID DRAG option.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Point Loads

Point loads and moments can be applied at the end nodes.

Output Of Strains

Generalized strains:

$1 = \varepsilon_{zz}$	= axial stretch.
$2 = \kappa_{xx}$	= curvature about local x-axis of cross section.
$3 = \kappa_{yy}$	= curvature about local y-axis of cross section.
$4 = \gamma$	= twist per unit length.

Output of Stresses

Generalized stresses:

1 = axial force
2 = local xx-moment
3 = local yy-movement
4 = axial torque

Stresses at integration points in the cross section are only printed if explicitly requested or if plasticity is present.

Transformation

Displacement and rotations at the nodes can be transformed to a local coordinate system.

Tying

Use tying type 100 for fully moment-carrying joints. Use tying type 103 for pin joints.

Output Points

Centroid or two Gaussian integration points. The first point is near the first node in the connectivity description of the element. The second point is near the second node in the connectivity description of the element.

Updated Lagrange Procedure and Finite Strain Plasticity

The Updated Lagrange procedure is available for this element, and should be used if the element is subjected to large rotations. Since the cross-sectional dimensions of the element are not updated, the element should not be used for finite strain applications.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 36. See Element 36 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

For the default hollow circular section only, the wall thickness and/or the radius can be considered as design variables.

Element 79

Thin-walled Beam in Three Dimensions including Warping

This is a simple, straight beam element that includes warping and twisting of the section. You must specify open cross sections through the BEAM SECT parameter. The section number is given in the GEOMETRY option. Primary warping effects are included, but twisting is assumed to be elastic. The large displacement formulation only affects the axial strain; large curvature, warping and twist changes are neglected.

The degrees of freedom associated with the nodes are three global displacements and three global rotations, all defined in a right-handed convention and the warping. The generalized strains are stretch, two curvatures, warping, and twist. Stresses are direct (axial) given at each point of the cross section. The local coordinate system which establishes the positions of those points on the section is defined by the fourth, fifth and sixth coordinates at the end nodes, which give the (x,y,z) global Cartesian coordinates of a point in space which locates the local x-axis of the cross section. This axis lies in the plane defined by the beam nodes and this point, pointing from the beam, toward this point. Alternatively, this point can also be specified by the fourth, fifth, and sixth entry on the GEOMETRY option if the local coordinate system is constant over the element. The local z-axis is along the beam from the first to the second node, and the local y-axis forms a right-handed set with the local x and local z.

All constitutive relations can be used with this element.

Notes:	The BEAM SECT parameter must be used to describe the section.
	For all beam elements (13, 14, 25, 52, 76, 77, 78, 79, and 98), the default printout gives section forces and moments, plus stress at any layer with plastic, creep strain or nonzero temperature. This default printout can be changed via the PRINT ELEMENT option.
	This element can be used in combination with shell element 75 and closed section beam element 78 to model stiffened shell structures. No tyings are necessary in that case.

Quick Reference

Type 79

Open-section beam including warping.

Connectivity

Two nodes per element (see Figure 3-129).

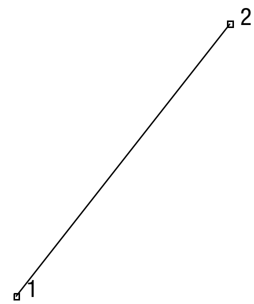


Figure 3-129 Open-section Beam

Geometry

Set `EGEOM2` to the section number needed. (Sections are defined using the BEAM SECT parameter.)

Optional specification of cross-section direction with `EGEOM4`, `EGEOM5`, and `EGEOM6` (see Coordinates).

If beam-to-beam contact is switched on (see CONTACT option), the radius used when the element comes in contact with other beam or truss elements must be entered in the seventh data field (`EGEOM7`).

For segment-segment beam contact, various flags that control the creation of patches are entered in the seventh data file (`EGEOM7`).

```
EGEOM7 = real(100 * NSEG + 10 * IPTCH + IESCAP)
```

where

`NSEG` = number of segments for hollow tubes (default = 32)		
`IPTCH` = patch flag for hollow tubes		
	1	Both outer and inner patches (default).
	3	Only outer patches (at radius + 1/2 * thickness).
	-3	Only outer patches (at radius - i.e., ignore thickness).
	5	Only inner patches (at radius - 1/2 * thickness).
	-5	Only inner patches (at radius - i.e., ignore thickness).
`IESCAP` = end cap/side cap flag		
	0	No end or side cap (default).
	1	End cap only.
	2	Side cap only.
	3	Average side only.
	4	End and side cap.
	5	End cap and average side.

Beam Offsets Pin Codes and Local Beam Orientation

If an offset is applied to the beam geometry or pin codes are applied to the degrees of freedom of the end point of the beams or the beam axis is defined with respect to a local coordinate system, then a code is placed in the 8th field of the 3rd data block of the GEOMETRY option (EGEOM8). This code is formed by the negative sum of ioffset, iorien, and ipin where:

ioffset	=	0	-	no offset
		1	-	beam offset
iorien	=	0	-	conventional definition of local beam orientation; beam axis given in 4th through 6th field with respect to global system
		10	-	the local beam orientation entered in the 4th through 6th field is given with respect to the coordinate system of the first beam node. This vector is transformed to the global coordinate system before any further processing.
ipin	=	0	-	no pin codes are used
		100	-	pin codes are used.

If ioffset = 1, an additional line is read in the GEOMETRY option (4th data block).

If ipin = 100, an additional line is read in the GEOMETRY option (5th data block)

The offset vector at the first corner node is provided in the first three fields and the offset vector at the second corner node is provided in the next three fields of the extra line. The coordinate system used for the offset vectors at the two nodes is indicated by the corresponding flags in the eighth and ninth fields of the extra line. A flag setting of 0 indicates global coordinate system, 1 indicates local element coordinate system, 2 indicates local shell normal, and 3 indicates local nodal coordinate system. When the flag setting is 1, the local beam coordinate system is constructed by taking the local z axis along the beam, the local x axis as the user-specified vector (obtained from EGEOM4-EGEOM6), and the local y axis as perpendicular to both. The flag setting of 2 can be used when the beam nodes are also attached to a shell. In this case, the offset magnitude at the first and second nodes are specified in the first and fourth data fields of the extra line respectively. The automatically calculated shell normals at the nodes are used to construct the offset vectors.

Coordinates

Six coordinates at each node. The first three coordinates are global (x,y,z). The fourth, fifth, and sixth coordinates are the global x,y,z coordinates of a point in space which locates the local x-axis of the cross section. The local x-axis is a vector normal to the beam axis through the point described by the fourth, fifth, and sixth coordinates. These coordinates can also be given as EGEOM4, EGEOM5, and EGEOM6 under the GEOMETRY option.

The local x-axis is positive progressing from the beam to the point.

Degrees of Freedom

$1 = u$
$2 = v$
$3 = w$
$4 = \phi_x$
$5 = \phi_y$
$6 = \phi_z$
$7 = \eta$ = warping degrees of freedom

Distributed Loads

Distributed load types as follows:

Load Type	Description
1	Uniform load per unit length in global x-direction.
2	Uniform load per unit length in global y-direction.
3	Uniform load per unit length in global z-direction.
4	Nonuniform load per unit length; magnitude and direction supplied via the FORCEM user subroutine.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Point Loads

Point loads and moments can be applied at the end nodes.

Output of Strains

Generalized strains:

1 = ε_{zz}	= axial stretch
2 = κ_{xx}	= curvature about local x-axis of cross section.
3 = κ_{yy}	= curvature about local y-axis of cross section.
4 = η	= warping.
5 = γ	= twist.

Output of Stresses

Generalized stresses:

1 = axial force
2 = local xx-moment
3 = local yy-movement
4 = bimoment
5 = axial torque

Layer stresses are only printed if explicitly requested or if plasticity is present.

Transformation

Displacement and rotations at the nodes can be transformed to a local coordinate reference.

Tying

Use tying type 100 for fully moment-carrying joints, tying type 103 for pin joints.

Output Points

Centroid or two Gaussian integration points. The first point is near the first node in the Connectivity description of the element. The second point is near the second node in the **Connectivity** description of the element.

Updated Lagrange Procedure and Finite Strain Plasticity

The Updated Lagrange procedure is available for this element, and should be used if the element is subjected to large rotations. Since the cross-sectional dimensions of the element are not updated, the element should not be used for finite strain applications.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 36. See Element 36 for a description of the conventions used for entering the flux and film data for this element.

Element 80

Arbitrary Quadrilateral Plane Strain, Herrmann Formulation

Element type 80 is a four-node, isoparametric, arbitrary quadrilateral written for plane strain incompressible applications. As this element uses bilinear interpolation functions, the strains tend to be constant throughout the element. This results in a poor representation of shear behavior. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is constant in this element.

In general, you need more of these lower-order elements than the higher-order elements such as types 32 or 58. Hence, use a fine mesh.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using four-point Gaussian integration.

This element is designed to be used for incompressible elasticity only. It can be used for either small strain behavior or large strain behavior using the Mooney or Ogden models.This element can be used in conjunction with other elements such as type 11 when other material behavior, such as plasticity, must be represented.

Quick Reference

Type 80

Plane strain quadrilateral, Herrmann formulation.

Connectivity

Five nodes per element. Node numbering for the corners follows right-handed convention (counterclockwise). The fifth node only has a pressure degree of freedom and is not to be shared with other elements.

Note:	Avoid reducing this element into a triangle.

Geometry

The thickness is entered in the first data field (`EGEOM1`). Defaults to unit thickness.

Coordinates

Two coordinates in the global x and y directions for the corner nodes. No coordinates are necessary for the fifth hydrostatic pressure node.

Degrees of Freedom

Global displacement degrees of freedom at the corner nodes:

1 = u displacement (x-direction)
2 = v displacement (y-direction)

One degree of freedom (negative hydrostatic pressure) at the fifth node.

1	$= \sigma_{kk}/E$	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1+\frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2+\frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3\right]\frac{9}{E_1+E_2+E_3}$$	
	= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)
	= p / K	(for Herrmann elements using additive decomposition; K here is the effective bulk modulus)

Distributed Loads

Load types for distributed loads as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force by unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element.
4	Nonuniform body force per unit volume in first coordinate direction.
5	Nonuniform body force per unit volume in second coordinate direction.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element.
8	Uniform pressure on 3-4 face of the element.
9	Nonuniform pressure on 3-4 face of the element.
10	Uniform pressure on 4-1 face of the element.
11	Nonuniform pressure on 4-1 face of the element.
20	Uniform shear force on 1-2 face in the 1-2 direction.
21	Nonuniform shear force on side 1 - 2.
22	Uniform shear force on side 2 - 3 (positive from 2 to 3).
23	Nonuniform shear force on side 2 - 3.
24	Uniform shear force on side 3 - 4 (positive from 3 to 4).
25	Nonuniform shear force on side 3 - 4.
26	Uniform shear force on side 4 - 1 (positive from 4 to 1).
27	Nonuniform shear force on side 4 - 1.

Load Type	Description
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

All pressures are positive when directed into the element. In addition, point loads can be applied at the nodes.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid of the element in global coordinates are:

1	= ε_{xx}	
2	= ε_{yy}	
3	= ε_{zz}	
4	= γ_{xy}	
5	= σ_{kk}/E	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1+\frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2+\frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3\right]\frac{9}{E_1+E_2+E_3}$$	
	= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)

The last component is not printed in the output file, but is accessible via the ELEVAR and PLOTV user subroutines.

Output of Stresses

Four stresses corresponding to the first four Output of Strains .

Transformation

The two global degrees of freedom at the corner nodes can be transformed into local coordinates.

Output Points

Output is available at the centroid or at the four Gaussian points shown in Figure 3-130.

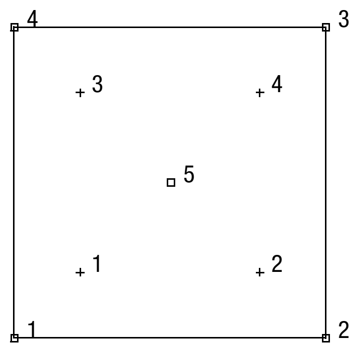


Figure 3-130 Gaussian Integration Points for Element Type 80

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available. Finite elastic strains can be calculated with the MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 39. See Element 39 for a description of the conventions used for entering the flux and film data for this element.

Element 81

Generalized Plane Strain Quadrilateral, Herrmann Formulation

This element is an extension of the plane strain isoparametric quadrilateral (element 80) to the generalized plane strain case, and modified for the Herrmann variational principle. This is an easy element for use in incompressible analysis. The fifth node, which should not be shared by other elements, represents the hydrostatic pressure. The generalized plane strain condition is obtained by allowing two extra (sixth and seventh) nodes in each element. One of these nodes has the single degree of freedom of change of distance between the top and bottom of the structure at one point (that is, change in thickness at that point) while the other additional node contains the relative rotations θ_{xx} and θ_{yy} of the top surface with respect to the bottom surface about the same point. The generalized plane strain condition is attained by allowing these two nodes to be shared by all elements forming the structure. Note that this is an extension of the usual generalized plane strain condition which is obtained by setting the relative rotations to zero (by kinematic boundary conditions), and from there the element could become a plane strain element if the relative displacement is also made zero. Note that the sharing of the two extra nodes by all elements implies two full nodal rows in the generalized plane strain part of the assembled stiffness matrix; considerable CPU time savings are achieved if these two nodes are given the highest node numbers in that part of the structure. The generalized plane strain formulation follows that of Marc Volume A: Theory and User Information.

As this element uses bilinear interpolation functions, the strains tend to be constant throughout the element. This results in a poor representation of shear behavior. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is constant in this element.

In general, you need more of these lower-order elements than the higher-order elements such as types 34 or 60. Hence, use a fine mesh.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using four-point Gaussian integration.

This element is designed to be used for incompressible elasticity only. It can be used for either small strain behavior or large strain behavior using the Mooney or Ogden models.This element can be used in conjunction with other elements such as type 19 when other material behavior, such as plasticity, must be represented.

This element cannot be used with the CASI iterative solver.

Note:	Because this element is most likely used in geometrically nonlinear analysis, it should not be used in conjunction with the CENTROID parameter.

Quick Reference

Type 81

Generalized plane strain quadrilateral, Herrmann formulation.

Connectivity

Seven nodes per element (see Figure 3-131). Node numbering is as follows:

> The first four nodes are the corners of the element in the x-y plane. The numbering must proceed counterclockwise around the element when the x-y plane is viewed from the positive z-side.

The fifth node has only a pressure degree of freedom and is not shared with other elements. The sixth and seventh nodes are shared by all generalized plane strain elements in this part of the structure. These two nodes should have the highest node numbers in the generalized plane strain part of the structure, to reduce matrix solution time.

Note:	Avoid reducing this element into a triangle.

Geometry

The thickness is entered in the first data field (`EGEOM1`). Default is unit thickness.

Coordinates

Coordinates are X and Y at all nodes. The coordinates of node 5 do not need to be defined. Note the position of the first shared node (node 6 of each element) determines the point where the thickness change is measured. You choose the location of nodes 6 and 7. These nodes should be at the same location in space.

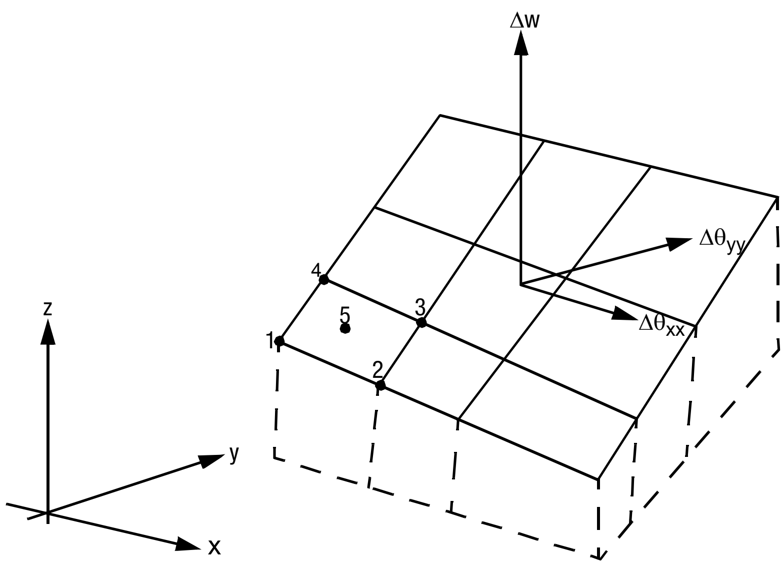


Figure 3-131 Generalized Plane Strain Elements

Degrees of Freedom

Global displacement degrees of freedom:

1 = u displacement (parallel to x-axis)
2 = v displacement (parallel to y-axis)

at the four corner nodes.

For node 5:

1	= negative hydrostatic pressure	
	$= \sigma_{kk}/E$	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}} \sigma_1 + \frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}} \sigma_2 + \frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}} \sigma_3 \right] \frac{9}{E_1 + E_2 + E_3}$$	
	= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)
	= p / K	(for Herrmann elements using additive decomposition; K here is the effective bulk modulus)

For the first shared node (node 6 of each element):

1 = Δw = thickness change at that point
2 – is not used.

For the second shared node (node 7 of each element):

1 = $\Delta\theta_{xx}$ =	relative rotation of top surface of generalized plane strain section of structure, with respect to its bottom surface, about the x-axis.
2 = $\Delta\theta_{yy}$ =	relative rotation of the top surface with respect to the bottom surface about the y-axis.

Distributed Loads

Load types for distributed loads as follows:

Load Type	Description
* 0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force per unit volume in first coordinate direction.
* 3	Nonuniform pressure on 1-2 face of the element.
4	Nonuniform body force per unit volume in first coordinate direction.
5	Nonuniform body force per unit volume in first coordinate direction.
* 6	Uniform pressure on 2-3 face of the element.
* 7	Nonuniform pressure on 2-3 face of the element.
* 8	Uniform pressure on 3-4 face of the element.
* 9	Nonuniform pressure on 3-4 face of the element.
* 10	Uniform pressure on 4-1 face of the element.
* 11	Nonuniform pressure on 4-1 face of the element.
* 20	Uniform shear force on 1-2 face in the 1-2 direction.

Load Type	Description
* 21	Nonuniform shear force on side 1 - 2.
* 22	Uniform shear force on side 2 - 3 (positive from 2 to 3).
* 23	Nonuniform shear force on side 2 - 3.
* 24	Uniform shear force on side 3 - 4 (positive from 3 to 4).
* 25	Nonuniform shear force on side 3 - 4
* 26	Uniform shear force on side 4 - 1 (positive from 4 to 1).
* 27	Nonuniform shear force on side 4 - 1.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine. Load types shown with an asterisk (*) require the magnitude of the load to be entered as force per unit area. To prescribe these loads in force per unit length, add 50 to the load type. This is often useful in design optimization where the thickness changes, but it is desired that the applied force remain the same.

All pressures are positive when directed into the element. In addition, point loads can be applied at the nodes.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

1	$= \varepsilon_{xx}$
2	$= \varepsilon_{yy}$
3	$= \varepsilon_{zz}$
4	$= \gamma_{xy}$,

5	$= \sigma_{kk}/E$	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1 + \frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2 + \frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3 \right] \frac{9}{E_1 + E_2 + E_3}$$	
	= -p	= negative pressure (for Mooney, Ogden, or Soil)

The last component is not printed in the output file, but is accessible via the ELEVAR and PLOTV user subroutines.

Output of Stresses

Four stresses, corresponding to the first four Output of Strains .

Transformation

In the x-y plane for the corner nodes only.

Tying

Use the UFORMSN user subroutine.

Output Points

Centroid or four Gaussian integration points.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available. Finite elastic strains can be calculated with the MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 39. See Element 39 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered as a design variable.

Element 82 — Arbitrary Quadrilateral Axisymmetric Ring, Herrmann Formulation

Element type 82 is a four-node, isoparametric arbitrary quadrilateral written for axisymmetric incompressible applications. As this element uses bilinear interpolation functions, the strains tend to be constant throughout the element. This results in a poor representation of shear behavior. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is constant in this element. In general, you need more of these lower order elements than the higher-order elements such as types 33 or 59. Hence, use a fine mesh.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using four-point Gaussian integration.

This element is designed to be used for incompressible elasticity only. It can be used for either small strain behavior or large strain behavior using the Mooney or Ogden models. This element can be used in conjunction with other elements such as type 10 when other material behavior, such as plasticity, must be represented.

Note:	Because this element is most likely used in geometrically nonlinear analysis, it should not be used in conjunction with the CENTROID parameter.

Quick Reference

Type 82

Axisymmetric, arbitrary ring with a quadrilateral cross section, Herrmann formulation.

Connectivity

Five nodes per element. Node numbering for the corner nodes follows right-handed convention (counterclockwise). The fifth node only has a pressure degree of freedom and is not to be shared with other elements.

Note:	Avoid reducing this element into a triangle.

Coordinates

Two coordinates in the global z- and r-directions for the corner nodes. No coordinates are necessary for the hydrostatic pressure node.

Degrees of Freedom

Global displacement degrees of freedom at the corner nodes:

1 = axial displacement (in z-direction)
2 = radial displacement (in r-direction)

One degree of freedom (negative hydrostatic pressure) at the fifth node.

1	= negative hydrostatic pressure	
	$= \sigma_{kk}/E$	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1+\frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2+\frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3\right]\frac{9}{E_1+E_2+E_3}$$	
	= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)
	= p / K	(for Herrmann elements using additive decomposition; K here is the effective bulk modulus)

Distributed Loads

Load types for distributed loads are as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force by unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element.
4	Nonuniform body force per unit volume in first coordinate direction.
5	Nonuniform body force per unit volume in second coordinate direction.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element.
8	Uniform pressure on 3-4 face of the element.
9	Nonuniform pressure on 3-4 face of the element.
10	Uniform pressure on 4-1 face of the element.
11	Nonuniform pressure on 4-1 face of the element.
20	Uniform shear force on 1-2 face in the 1-2 direction.
21	Nonuniform shear force on side 1 - 2.
22	Uniform shear force on side 2 - 3 (positive from 2 to 3).
23	Nonuniform shear force on side 2 - 3.
24	Uniform shear force on side 3 - 4 (positive from 3 to 4).
25	Nonuniform shear force on side 3 - 4
26	Uniform shear force on side 4 - 1 (positive from 4 to 1).
27	Nonuniform shear force on side 4 - 1.

Load Type	Description
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

All pressures are positive when directed into the element. In addition, point loads can be applied at the nodes.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid of the element in global coordinates is as follows:

1	$= \varepsilon_{zz}$	
2	$= \varepsilon_{rr}$	
3	$= \varepsilon_{\theta\theta}$	
4	$= \gamma_{rz}$	
5	$= \sigma_{kk}/E$	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	= -p	= negative pressure (for Mooney, Ogden, or Soil)

$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1+\frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2+\frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3\right]\frac{9}{E_1+E_2+E_3}$$

The last component is not printed in the output file, but is accessible via the ELEVAR and PLOTV user subroutines.

Output of Stresses

Four stresses corresponding to the first four Output of Strains .

Transformation

The global degrees of freedom at the corner nodes can be transformed into local coordinates. No transformation for the pressure node.

Tying

Can be tied to axisymmetric shell type 1 using standard tying type 23.

Output Points

Output is available at the centroid or at the 4 Gaussian points shown in Figure 3-132.

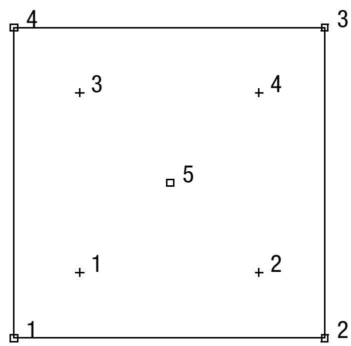


Figure 3-132 Integration Points for Element 82

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available. Finite elastic strains can be calculated with the MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 40. See Element 40 for a description of the conventions used for entering the flux and film data for this element.

Element 83 Axisymmetric Torsional Quadrilateral, Herrmann Formulation

Element type 83 is a four-node, isoparametric, arbitrary quadrilateral written for axisymmetric incompressible applications with torsional strains. It is assumed that there are no variations in the circumferential direction.

As this element uses bilinear interpolation functions, the strains tend to be constant throughout the element. This results in a poor representation of shear behavior. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is constant in this element. In general, you need more of these lower-order elements than the higher-order elements such as type 66. Hence, use a fine mesh.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using four-point Gaussian integration.

This element is designed to be used for incompressible elasticity only. It can be used for either small strain behavior or large strain behavior using the Mooney or Ogden models.This element can be used in conjunction with other elements such as type 20 when other material behavior, such as plasticity, must be represented.

Notice that there is no friction contribution in the torsional direction when the CONTACT option is used.

If the analysis involves large rotations about the symmetry axis, it is recommended to use a full 3-D model.

Note:	Because this element is most likely used in geometrically nonlinear analysis, it should not be used in conjunction with the CENTROID parameter.

Quick Reference

Type 83

Axisymmetric, arbitrary, ring with a quadrilateral cross section, Herrmann formulation.

Connectivity

Five nodes per element. Node numbering for the corner nodes in a right-handed manner (counterclockwise). The fifth node has only a pressure degree of freedom and is not to be shared with other elements.

Note:	Avoid reducing this element into a triangle.

Coordinates

Two coordinates in the global z and r directions respectively. No coordinates are necessary for the hydrostatic pressure node.

Degrees of Freedom

Global displacement degrees of freedom for the corner nodes:

1 = axial displacement (in z-direction)
2 = radial displacement (in r-direction)
3 = angular rotation about the symmetry axis (θ-direction, measured in radians)

For node 5:

1	= negative hydrostatic pressure	
	= σ_{kk}/E	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
		$H = \left[\dfrac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}} \sigma_1 + \dfrac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}} \sigma_2 + \dfrac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}} \sigma_3 \right] \dfrac{9}{E_1 + E_2 + E_3}$
	= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)

Distributed Loads

Load types for distributed loads as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force by unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element.
4	Nonuniform body force per unit volume in first coordinate direction.
5	Nonuniform body force per unit volume in second coordinate direction.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element.
8	Uniform pressure on 3-4 face of the element.
9	Nonuniform pressure on 3-4 face of the element.
10	Uniform pressure on 4-1 face of the element.
11	Nonuniform pressure on 4-1 face of the element.
20	Uniform shear force on 1-2 face in the 1-2 direction.
21	Nonuniform shear force on side 1 - 2.
22	Uniform shear force on side 2 - 3 (positive from 2 to 3).
23	Nonuniform shear force on side 2 - 3.

Load Type	Description
24	Uniform shear force on side 3 - 4 (positive from 3 to 4).
25	Nonuniform shear force on side 3 - 4
26	Uniform shear force on side 4 - 1 (positive from 4 to 1).
27	Nonuniform shear force on side 4 - 1.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

All pressures are positive when directed into the element. In addition, point loads and torques can be applied at the nodes. The magnitude of concentrated loads must correspond to the load integrated around the circumference.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid of the element in global coordinates.

1	= ε_{zz}	
2	= ε_{rr}	
3	= $\varepsilon_{\theta\theta}$	
4	= γ_{zr}	
5	= $\gamma_{r\theta}$	
6	= $\gamma_{\theta z}$	
7	= σ_{kk}/E	= mean pressure variable (isotropic)
	= formula below (orthotropic)	

		$H = \left[\dfrac{\dfrac{\nu_{12}\nu_{31}}{E_1 E_3}}{\dfrac{\nu_{12}\nu_{31}}{E_1 E_3} + \dfrac{\nu_{12}\nu_{23}}{E_1 E_2} + \dfrac{\nu_{31}\nu_{23}}{E_3 E_2}} \sigma_1 + \dfrac{\dfrac{\nu_{12}\nu_{23}}{E_1 E_2}}{\dfrac{\nu_{12}\nu_{31}}{E_1 E_3} + \dfrac{\nu_{12}\nu_{23}}{E_1 E_2} + \dfrac{\nu_{31}\nu_{23}}{E_3 E_2}} \sigma_2 + \dfrac{\dfrac{\nu_{31}\nu_{23}}{E_3 E_2}}{\dfrac{\nu_{12}\nu_{31}}{E_1 E_3} + \dfrac{\nu_{12}\nu_{23}}{E_1 E_2} + \dfrac{\nu_{31}\nu_{23}}{E_3 E_2}} \sigma_3 \right] \dfrac{9}{E_1 + E_2 + E_3}$
	= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)

The last component is not printed in the output file, but is accessible via the ELEVAR and PLOTV user subroutines.

Output of Stresses

Output for stress is the same direction as for Output of Strains .

Transformation

First two degrees of freedom at the corner nodes can be transformed to local degrees of freedom.

Tying

Use the UFORMSN user subroutine.

Output Points

Centroid or four Gaussian integration points (see Figure 3-133).

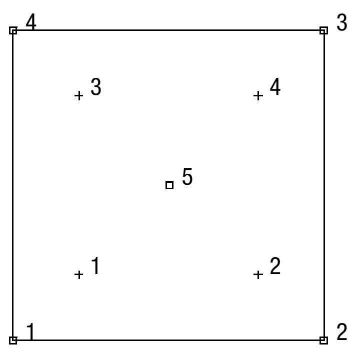


Figure 3-133 Gaussian Integration Points for Element Type 83

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available. Finite elastic strains can be calculated with the MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 40. See Element 40 for a description of the conventions used for entering the flux and film data for this element.

Element 84

Three-dimensional Arbitrarily Distorted Brick, Herrmann Formulation

This is an eight-node, isoparametric element with an additional ninth node for the pressure. The element is based on the following type of displacement assumption and mapping into a cube in the (g-h-r) space:

$$x = a_0 + a_1 g + a_2 h + a_3 r + a_4 gh + a_5 hr + a_6 gr + a_7 ghr$$
$$u = b_0 + b_1 g + b_2 h + b_3 r + b_4 gh + b_5 hr + b_6 gr + b_7 ghr$$

The pressure is assumed constant throughout the element. The 24 generalized displacements are related to the u-v-w displacements (in global coordinates) at the eight corners of the distorted cube. The stiffness of the element is formed by numerical integration using eight points defined in the (g-h-r) space.

Quick Reference

Type 84

Nine-node, three-dimensional, first-order isoparametric element (arbitrarily distorted brick) with mixed formulation.

Connectivity

Nine nodes per element (Figure 3-134).

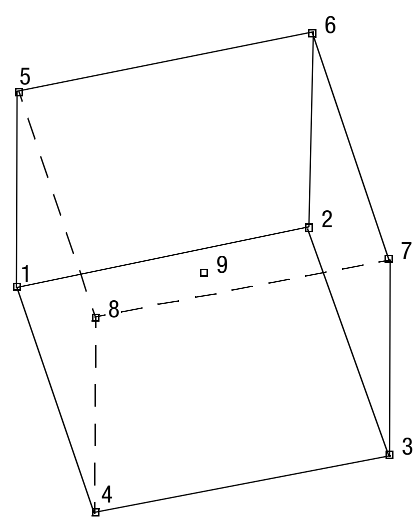


Figure 3-134 Element 84 with 9 Nodes

Node numbering must follow the scheme below:

> Nodes 1, 2, 3, and 4 are corners of one face, given in counterclockwise order when viewed from inside the element. Node 5 is the same edge as node 1, node 6 as node 2, node 7 as node 3, and node 8 as node 4. The node with the pressure degree of freedom is the last node in the connectivity list, and should not be shared with other elements.

Note:	Avoid reducing this element into a tetrahedron or a wedge.

Coordinates

Three coordinates in the global x-, y-, and z-directions for the first eight nodes. No coordinates are necessary for the pressure node.

Degrees of Freedom

Three global degrees of freedom u, v, and w at the first eight nodes. One degree of freedom (negative hydrostatic pressure) at the last node.

1	= σ_{kk}/E	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1 + \frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2 + \frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3 \right] \frac{9}{E_1 + E_2 + E_3}$$	
	= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)
	= p / K	(for Herrmann elements using additive decomposition; K here is the effective bulk modulus)

Distributed Loads

Distributed loads chosen by value of IBODY as follows:

Load Type	Description
0	Uniform pressure on 1-2-3-4 face.
1	Nonuniform pressure on 1-2-3-4 face; magnitude supplied through the FORCEM user subroutine.
2	Uniform body force per unit volume in -z direction.
3	Nonuniform body force per unit volume (e.g., centrifugal force); magnitude and direction supplied through the FORCEM user subroutine.
4	Uniform pressure on 6-5-8-7 face.
5	Nonuniform pressure on 6-5-8-7 face (FORCEM user subroutine).
6	Uniform pressure on 2-1-5-6 face.
7	Nonuniform pressure on 2-1-5-6 face (FORCEM user subroutine).
8	Uniform pressure on 3-2-6-7 face.
9	Nonuniform pressure on 3-2-6-7 face (FORCEM user subroutine).
10	Uniform pressure on 4-3-7-8 face.
11	Nonuniform pressure on 4-3-7-8 face (FORCEM user subroutine).

Load Type	Description
12	Uniform pressure on 1-4-8-5 face.
13	Nonuniform pressure on 1-4-8-5 face (FORCEM user subroutine).
20	Uniform pressure on 1-2-3-4 face.
21	Nonuniform load on 1-2-3-4 face; magnitude and direction supplied in the FORCEM user subroutine.
22	Uniform body force per unit volume in -z direction.
23	Nonuniform body force per unit volume (e.g., centrifugal force); magnitude and direction supplied through the FORCEM user subroutine.
24	Uniform pressure on 6-5-8-7 face.
25	Nonuniform load on 6-5-8-7 face; magnitude and direction supplied in the FORCEM user subroutine.
26	Uniform pressure on 2-1-5-6 face.
27	Nonuniform load on 2-1-5-6 face; magnitude and direction supplied in the FORCEM user subroutine.
28	Uniform pressure on 3-2-6-7 face.
29	Nonuniform load on 3-2-6-7 face; magnitude and direction supplied in the FORCEM user subroutine.
30	Uniform pressure on 4-3-7-8 face.
31	Nonuniform load on 4-3-7-8 face; magnitude and direction supplied in the FORCEM user subroutine.
32	Uniform pressure on 1-4-8-5 face.
33	Nonuniform load on 1-4-8-5 face; magnitude and direction supplied in the FORCEM user subroutine.
40	Uniform shear 1-2-3-4 face in 12 direction.
41	Nonuniform shear 1-2-3-4 face in 12 direction.
42	Uniform shear 1-2-3-4 face in 23 direction.
43	Nonuniform shear 1-2-3-4 face in 23 direction.
48	Uniform shear 6-5-8-7 face in 56 direction.
49	Nonuniform shear 6-5-8-7 face in 56 direction.
50	Uniform shear 6-5-8-7 face in 67 direction.
51	Nonuniform shear 6-5-8-7 face in 67 direction.
52	Uniform shear 2-1-5-6 face in 12 direction.
53	Nonuniform shear 2-1-5-6 face in 12 direction.
54	Uniform shear 2-1-5-6 face in 15 direction.
55	Nonuniform shear 2-1-5-6 face in 15 direction.
56	Uniform shear 3-2-6-7 face in 23 direction.
57	Nonuniform shear 3-2-6-7 face in 23 direction.
58	Uniform shear 3-2-6-7 face in 26 direction.
59	Nonuniform shear 2-3-6-7 face in 26 direction.

Load Type	Description
60	Uniform shear 4-3-7-8 face in 34 direction.
61	Nonuniform shear 4-3-7-8 face in 34 direction.
62	Uniform shear 4-3-7-8 face in 37 direction.
63	Nonuniform shear 4-3-7-8 face in 37 direction.
64	Uniform shear 1-4-8-5 face in 41 direction.
65	Nonuniform shear 1-4-8-5 face in 41 direction.
66	Uniform shear 1-4-8-5 face in 15 direction.
67	Nonuniform shear 1-4-8-5 face in 15 direction.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Pressure forces are positive into element face.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

1	= ε_{xx}	= global xx strain
2	= ε_{yy}	= global yy strain
3	= ε_{zz}	= global zz strain
4	= γ_{xy}	= global xy strain
5	= γ_{yz}	= global yz strain
6	= γ_{zx}	= global zx strain

7	= σ_{kk}/E	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1+\frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2+\frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3\right]\frac{9}{E_1+E_2+E_3}$$	
	= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)

The last component is not printed in the output file, but is accessible via the ELEVAR and PLOTV user subroutines.

Output Of Stresses

Six stresses, corresponding to first six Output of Strains .

Transformation

Standard transformation of three global degrees of freedom to local degrees of freedom at the corner nodes. No transformation for the pressure node.

Tying

No special tying available.

Output Points

Centroid or the eight integration points as shown in Figure 3-135.

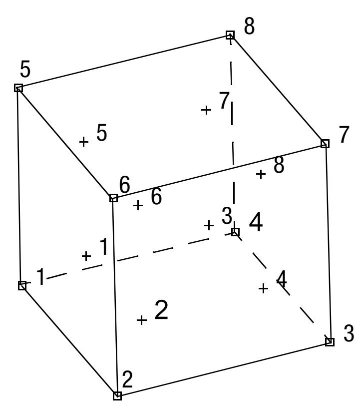


Figure 3-135 Gaussian Integration for Element Type 84

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available with the ELASTICITY,2 or PLASTICITY,5 parameters. Finite elastic strains can be calculated with the MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 43. See Element 43 for a description of the conventions used for entering the flux and film data for this element.

Element 85

Four-node Bilinear Shell (Heat Transfer Element)

This is a four-node heat transfer shell element with temperatures as nodal degrees of freedom. Bilinear interpolation is used for the temperatures in the plane of the shell and either a linear or a quadratic temperature distribution is assumed in the shell thickness direction (see the HEAT parameter). A four-point Gaussian integration is chosen for the element in the plane of the shell and a eleven-point Simpson's rule is used in the thickness direction. This element is compatible with stress shell element types 72 and 75 in thermal-stress analysis and can be used in conjunction with three-dimensional heat transfer brick elements through tying for heat transfer analysis.

Geometric Basis

Similar to element types 72 and 75, the element is defined geometrically by the (x,y,z) coordinates of the four corner nodes. Due to the bilinear interpolation, the surface forms a hyperbolic paraboloid which is allowed to degenerate to a plate. The element thickness is specified in the GEOMETRY option.

Local orthogonal surface directions (V_1, V_2, and V_3), with the centroid of the element as origin, are defined below (see Figure 3-136):

At the centroid, the vectors tangent to the curves with constant isoparametric coordinates are normalized.

$$t_1 = \frac{\partial x}{\partial \xi} \Big/ \left|\frac{\partial x}{\partial \xi}\right|, \quad t_2 = \frac{\partial x}{\partial \eta} \Big/ \left|\frac{\partial x}{\partial \eta}\right|$$

Now a new basis is being defined as:

$$s = t_1 + t_2, \quad d = t_1 - t_2$$

After normalizing these vectors by:

$$\bar{s} = (s / \sqrt{2})\ |s|, \quad \bar{d} = (d / \sqrt{2})\ |d|$$

the local orthogonal directions are then obtained as:

$$V_1 = \bar{s} + \bar{d}\ ,\ V_2 = \bar{s} - \bar{d}\ , \text{and}\ V_3 = V_1 \times V_2$$

In this way the vectors $\frac{\partial x}{\partial \xi}$, $\frac{\partial x}{\partial \eta}$ and V_1, V_2 have the same bisecting plane.

The local directions for the Gaussian integrations points are found by projection of the centroid directions. Hence, if the element is flat, the directions at the Gauss points are identical to those at the centroid.

In addition, this element can be used for an electrostatic problem. A description of this option can be found in Marc Volume A: Theory and User Information.

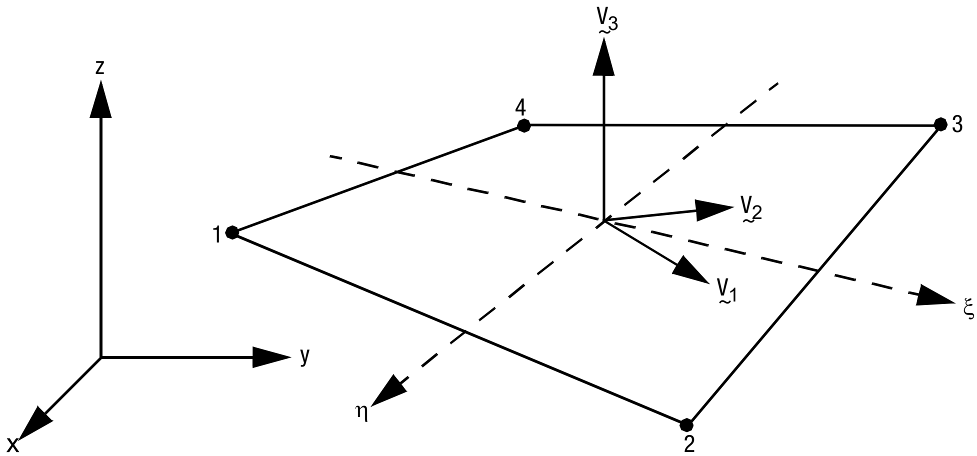


Figure 3-136 Four-node Heat Transfer Shell Element

Quick Reference

Type 85

Four-node bilinear, heat transfer shell element.

Connectivity

Four nodes per element. The element can be collapsed to a triangle.

Geometry

Bilinear thickness variation is allowed in the plane of the element. Thicknesses at first, second, third and fourth nodes of the element are stored for each element in the first (EGEOM1), second (EGEOM2), third (EGEOM3) and fourth (EGEOM4), geometry data fields, respectively. If EGEOM2 = EGEOM3 = EGEOM4 = 0, then a constant thickness (EGEOM1) is assumed for the element.

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

If the element needs to be offset from the user-specified position, the eighth data field (EGEOM8) is set to -2.0. In this case, an extra line containing the offset information is provided (see the GEOMETRY option in Marc Volume C: Program Input). The offset magnitudes along the element normal for the four corner nodes are provided in the first, second, third, and fourth data fields of the extra line. A uniform offset for all nodes can be set by providing the offset magnitude in the first data field and then setting the constant offset flag (sixth data field of the extra line) to 1.

Coordinates

Three coordinates per node in the global x, y, and z direction.

Degrees of Freedom

The definition of the degrees of freedom depends on the temperature distribution through the thickness and is defined on the HEAT parameter.

The number of degrees of freedom per node is N.

Linear distribution through the thickness (N=2):

1	=	Top Surface Temperature
2	=	Bottom Surface Temperature

Quadratic distribution through the thickness (N=3):

1	=	Top Surface Temperature
2	=	Bottom Surface Temperature
3	=	Mid Surface Temperature

Linear distribution through every layer in Composites with M layers (N=M+1):

1	=	Top Surface Temperature
2	=	Temperature at Interface between Layer 1 and Layer 2
3	=	Temperature at Interface between Layer 2 and Layer 3 etc
N = M+1	=	Bottom Surface Temperature

Quadratic distribution through every layer in Composites with M layers (N=2*M+1):

1	=	Top Surface Temperature
2	=	Temperature at Interface between Layer 1 and Layer 2
3	=	Mid Temperature of Layer 1
4	=	Temperature at Interface between Layer 2 and Layer 3
5	=	Mid Temperature of Layer 2
	...	
N-1 = 2*M	=	Bottom Surface Temperature
N = 2*M+1	=	Mid Temperature of Layer N

Piecewise Quadratic distribution through thickness in Non-Composites with M layers (SHELL SECT,**M**) (N=M+1):

1	=	Top Surface Temperature (thickness position 0.)
X	=	Temperature at thickness position (X-1)/M
N = M+1	=	Bottom Surface Temperature (thickness position 1.

Fluxes

Two types of distributed fluxes:

Volumetric Fluxes

Flux Type	Description
1	Uniform flux per unit volume on whole element.
3	Nonuniform flux per unit volume on whole element; magnitude of flux is defined in the FLUX user subroutine.

Surface Fluxes

Flux Type	Description
5	Uniform flux per unit surface area on top surface.
6	Nonuniform flux per unit surface area on top surface; magnitude of flux is in the FLUX user subroutine.
2	Uniform flux per unit surface area on bottom surface.
4	Nonuniform flux per unit surface area on bottom surface, magnitude of flux is defined in the FLUX user subroutine.

Surface fluxes are assumed positive when directed into the element and are evaluated using a 4-point integration scheme where the integration points have the same location as the nodal points.

Point fluxes can also be applied at nodal degrees of freedom.

Films

Same specification as Fluxes .

Tying

Standard types 85 and 86 with three-dimensional heat transfer brick elements.

Shell Sect – Integration Through The Shell Thickness Direction

The number of layer through the thickness is controlled by the SHELL SECT parameter (must be an odd number) with the default of 11 layers or via the COMPOSITE model definition option. The accuracy is controlled by the number of temperature degrees of freedom through the thickness.

Output Points

Temperatures are printed out at the integration points through the thickness of the shell, at the centroid or the four Gaussian integration points in the plane of the shell. The first point in the thickness direction is on the surface of the positive normal.

Joule Heating

Capability is not available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Charge

Same specifications as Fluxes .

Element 86

Eight-node Curved Shell (Heat Transfer Element)

This is a eight-node heat transfer shell element with temperatures as nodal degrees of freedom. Quadratic interpolation is used for the temperatures in the plane of the shell and either a linear or a quadratic temperature distribution is assumed in the shell thickness direction (see the HEAT parameter). A nine-point Gaussian integration is chosen for the element in the plane of the shell and Simpson's rule is used in the thickness direction. When used in conjunction with three-dimensional heat transfer brick elements, use tying types 85 or 86 to tie nodes of this element to nodes of the brick element. Note that only centroid or four Gaussian points are used for the output of element temperatures.

Geometric Basis

Similar to element type 22, the element is defined geometrically by the (x, y, z) coordinates of the four corner nodes and four midside nodes. The element thickness is specified in the GEOMETRY option. Local orthogonal surface directions (V_1, V_2, and V_3) for each integration point are defined below (see Figure 3-137):

At each of the integration points, the vectors tangent to the curves with constant isoparametric coordinates are normalized.

$$t_1 = \frac{\partial x}{\partial \xi} \Big/ \left|\frac{\partial x}{\partial \xi}\right|, \quad t_2 = \frac{\partial x}{\partial \eta} \Big/ \left|\frac{\partial x}{\partial \eta}\right|$$

Now a new basis is being defined as:

$$s = t_1 + t_2, \quad d = t_1 - t_2$$

After normalizing these vectors by:

$$\bar{s} = s / (\sqrt{2}|s|) \qquad \bar{d} = d / (\sqrt{2}|d|) ,$$

the local orthogonal directions are then obtained as:

$$V_1 = \bar{s} + \bar{d}, V_2 = \bar{s} - \bar{d}, \text{and } V_3 = V_1 \times V_2$$

In this way, the vectors $\frac{\partial x}{\partial \xi}$, $\frac{\partial x}{\partial \eta}$ and V_1, V_2 have the same bisecting plane.

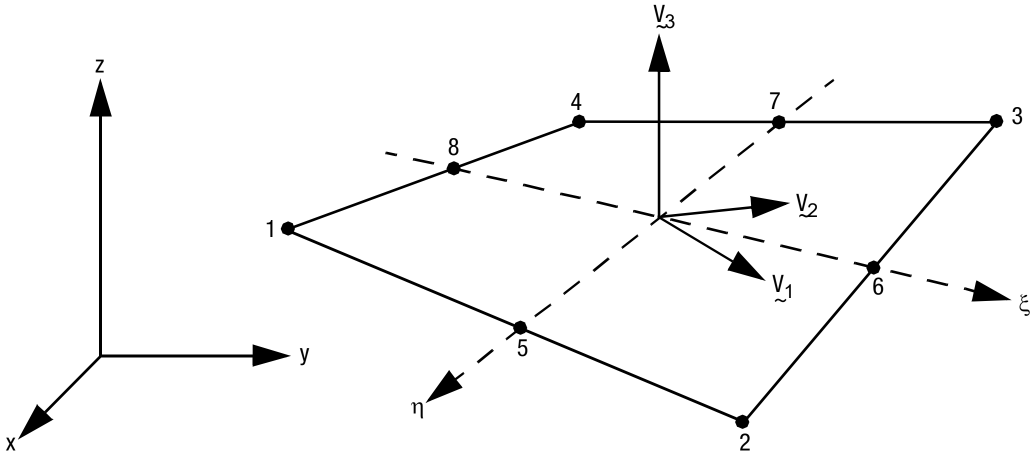


Figure 3-137 Form of Element 86

Quick Reference

Type 86

Eight-node curved heat transfer shell element.

Connectivity

Eight nodes per element.

The connectivity is specified as follows: nodes 1, 2, 3, 4 form the corners of the element, then node 5 lies at the middle of the 1-2 edge; node 6 at the middle of the 2-3 edge, etc. See Figure 3-137.

Geometry

Bilinear thickness variation is allowed in the plane of the element. Thicknesses at first, second, third, and fourth nodes of the element are stored for each element in the first (EGEOM1), second (EGEOM2), third (EGEOM3) and fourth (EGEOM4), geometry data fields, respectively. If EGEOM2 = EGEOM3 = EGEOM4 = 0, then a constant thickness (EGEOM1) is assumed for the element.

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

If the element needs to be offset from the user-specified position, the eighth data field (EGEOM8) is set to -2.0. In this case, an extra line containing the offset information is provided (see the GEOMETRY option in Marc Volume C: Program Input). The offset magnitudes along the element normal for the four corner nodes are provided in the first, second, third, and fourth data fields of the extra line. If the interpolation flag (5th data field) is set to 1, then the offset values at the midside nodes is obtained by interpolating from the corresponding corner values. A uniform offset for all nodes can be set by providing the offset magnitude in the first data field and then setting the constant offset flag (sixth data field of the extra line) to 1.

Coordinates

Three coordinates per node in the global x-, y-, and z-direction.

Degrees of Freedom

The definition of the degrees of freedom depends on the temperature distribution through the thickness and is defined on the HEAT parameter.

The number of degrees of freedom per node is N.

Linear distribution through the thickness (N=2):

1	=	Top Surface Temperature
2	=	Bottom Surface Temperature

Quadratic distribution through the thickness (N=3):

1	=	Top Surface Temperature
2	=	Bottom Surface Temperature
3	=	Mid Surface Temperature

Linear distribution through every layer in Composites with M layers (N=M+1):

1	=	Top Surface Temperature
2	=	Temperature at Interface between Layer 1 and Layer 2
3	=	Temperature at Interface between Layer 2 and Layer 3 etc
N = M+1	=	Bottom Surface Temperature

Quadratic distribution through every layer in Composites with M layers (N=2*M+1):

1	=	Top Surface Temperature
2	=	Temperature at Interface between Layer 1 and Layer 2
3	=	Mid Temperature of Layer 1
4	=	Temperature at Interface between Layer 2 and Layer 3
5	=	Mid Temperature of Layer 2
...		
N-1 = 2*M	=	Bottom Surface Temperature
N = 2*M+1	=	Mid Temperature of Layer N

Piecewise Quadratic distribution through thickness in Non-Composites with M layers (SHELL SECT,M) (N=M+1):

1	=	Top Surface Temperature (thickness position 0.)
X	=	Temperature at thickness position (X-1)/M
N = M+1	=	Bottom Surface Temperature (thickness position 1.

Fluxes

Two types of distributed fluxes:

Volumetric Fluxes

Flux Type	Description
1	Uniform flux per unit volume on whole element.
3	Nonuniform flux per unit volume on whole element; magnitude of flux is defined in the FLUX user subroutine.

Surface Fluxes

Flux Type	Description
5	Uniform flux per unit surface area on top surface.
6	Nonuniform flux per unit surface area on top surface, magnitude of flux is in the FLUX user subroutine.
2	Uniform flux per unit surface area on bottom surface.
4	Nonuniform flux per unit surface area on bottom surface, magnitude of flux is defined in the FLUX user subroutine.

Point fluxes can also be applied at nodal degrees of freedom.

Films

Same specification as Fluxes .

Tying

Standard types 85 and 86 with three-dimensional heat transfer brick elements.

Shell Sect – Integration Through The Shell Thickness Direction

Integration through the shell thickness is performed numerically using Simpson's rule. Use the SHELL SECT parameter to specify the number of integration points. This number must be odd. Three points are enough for linear response. The default is 11 points.

Output Points

Temperatures are printed out at the integration points through the thickness of the shell at the centroid or Gaussian integration points in the plane of the shell. The first point in the thickness direction is on the surface of the positive normal.

Joule Heating

Capability is not available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Charge

Same specifications as Fluxes .

Element 87 Three-node Axisymmetric Shell (Heat Transfer Element)

This is a three-node, axisymmetric shell, heat transfer element with temperatures as nodal degrees of freedom. Quadratic interpolation is used for the temperatures in the plane of the shell, and either a linear or a quadratic temperature distribution is assumed in the shell thickness direction (see the HEAT parameter). A three-point Gaussian integration is chosen for the element in the plane of the shell and Simpson's rule is used in the thickness direction. This element is compatible with stress shell element type 89 in thermal-stress analysis and can be used in conjunction with axisymmetric heat transfer elements through tying for heat transfer analysis. Note that only the centroid or two Gaussian points are used for the output of element temperatures.

In addition, this element can be used for an electrostatic problem. A description of this option can be found in Marc Volume A: Theory and User Information.

Quick Reference

Type 87

Three-node axisymmetric, curved heat transfer-shell element.

Connectivity

Three nodes per element (see Figure 3-138).

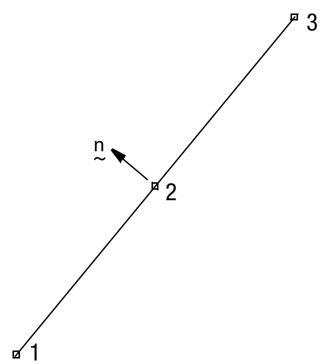


Figure 3-138 Three-node Axisymmetric, Heat Transfer-Shell Element

Geometry

Linear thickness variation is allowed along the length of the element. For each element, thickness at first node of the element is stored in the first data field (EGEOM1); thickness at third node is stored in the third data field (EGEOM3). If EGEOM3=0, a constant thickness is assumed for the element.

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

If the element needs to be offset from the user-specified position, the eighth data field (EGEOM8) is set to -2.0. In this case, an extra line containing the offset information is provided (see the GEOMETRY option in Marc Volume C: Program Input). The offset magnitudes at the two corner nodes are taken to be along the element normal and are provided in the first and second data fields of the extra line, respectively. If the interpolation flag (fifth data field) is set to 1, then the offset value at the midside node is obtained by interpolating from the corresponding corner values. A uniform offset for the element can be set by providing the offset magnitude in the first data field and then setting the constant offset flag (sixth data field of the extra line) to 1.

Coordinates

1 = z
2 = r

Degrees of Freedom

The definition of the degrees of freedom depends on the temperature distribution through the thickness and is defined on the HEAT parameter.

The number of degrees of freedom per node is N.

Linear distribution through the thickness (N=2):

1	=	Top Surface Temperature
2	=	Bottom Surface Temperature

Quadratic distribution through the thickness (N=3):

1	=	Top Surface Temperature
2	=	Bottom Surface Temperature
3	=	Mid Surface Temperature

Linear distribution through every layer in Composites with M layers (N=M+1):

1	=	Top Surface Temperature
2	=	Temperature at Interface between Layer 1 and Layer 2
3	=	Temperature at Interface between Layer 2 and Layer 3 etc
N = M+	=	Bottom Surface Temperature

Quadratic distribution through every layer in Composites with M layers (N=2*M+1):

1	=	Top Surface Temperature
2	=	Temperature at Interface between Layer 1 and Layer 2
3	=	Mid Temperature of Layer 1
4	=	Temperature at Interface between Layer 2 and Layer 3
5	=	Mid Temperature of Layer 2
...		

N-1 = 2*M	=	Bottom Surface Temperature
N = 2*M+1	=	Mid Temperature of Layer N

Piecewise Quadratic distribution through thickness in Non-Composites with M layers (SHELL SECT,M) (N=M+1):

1	=	Top Surface Temperature (thickness position 0.)
X	=	Temperature at thickness position (X-1)/M
N = M+1	=	Bottom Surface Temperature (thickness position 1.

Fluxes

Two types of distributed fluxes are as follows:

Volumetric Fluxes

Flux Type	Description
1	Uniform flux per unit volume on whole element.
3	Nonuniform flux per unit volume on whole element; magnitude of flux is defined in the FLUX user subroutine.

Surface Fluxes

Flux Type	Description
2	Uniform flux per unit surface area on bottom surface.
4	Nonuniform flux per unit surface area on bottom surface; magnitude of flux is defined in the FLUX user subroutine.
5	Uniform flux per unit surface area on top surface.
6	Nonuniform flux per unit surface area on top surface; magnitude of flux is defined in the FLUX user subroutine.

Point fluxes can also be applied at the degrees of freedom and must be integrated around the circumference.

Films

Same specification as Fluxes .

Tying

Standard types 85 and 86 with axisymmetric heat transfer elements.

Shell Sect – Integration Through The Shell Thickness Direction

The number of layer through the thickness is controlled by the SHELL SECT parameter (must be an odd number) with the default of 11 layers or via the COMPOSITE model definition option. The accuracy is controlled by the number of temperature degrees of freedom through the thickness.

Output Points

Temperatures are printed out at the layer points through the thickness of the shell, at the centroid or three Gaussian integration points. The first point in the thickness direction is on the surface of the positive normal.

There are two integration points per layer written to the post file, so this element may be used in a thermal stress analysis with element type 89. These values are obtained by extrapolating to the element nodes and then interpolating to the output integration points.

Joule Heating

Capability is not available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Charge

Same specifications as Fluxes .

Element 88

Two-node Axisymmetric Shell (Heat Transfer Element)

This is a two-node, axisymmetric shell, heat transfer element with temperatures as nodal degrees of freedom. Linear interpolation is used for the temperatures in the plane of the shell and either a linear or a quadratic temperature distribution is assumed in the shell thickness direction (see the HEAT parameter). A three-point Gaussian integration is chosen for the element in the plane of the shell and Simpson's rule is used in the thickness direction. This element is compatible with stress shell element type 1 in thermal-stress analysis and can be used in conjunction with axisymmetric heat transfer elements through tying for heat transfer analysis. Note that only the centroid is used for the output of element temperatures.

In addition, this element can be used for an electrostatic problem. A description of this option can be found in Marc Volume A: Theory and User Information.

Quick Reference

Type 88

Two-node axisymmetric, heat transfer shell element.

Connectivity

Two nodes per element.

Geometry

Constant thickness of the shell is stored in the first data field (EGEOM1).

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

If the element needs to be offset from the user-specified position, then the 8th data field (`EGEOM8`) is set to -2.0. In this case, an extra line containing the offset information is provided (see the GEOMETRY option in Marc Volume C: Program Input). The offset magnitudes at the first and second nodes are taken to be along the element normal and are provided in the 1st and 2nd data fields of the extra line, respectively. A uniform offset for the element can be set by providing the offset magnitude in the 1st data field and then setting the constant offset flag (6th data field of the extra line) to 1.

Coordinates

Two coordinates are required, z and r, as shown in Figure 3-139.

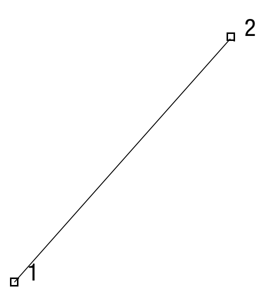


Figure 3-139 Two-node Axisymmetric Heat Transfer Shell Element

Degrees of Freedom

The definition of the degrees of freedom depends on the temperature distribution through the thickness and is defined on the HEAT parameter.

The number of degrees of freedom per node is N.

Linear distribution through the thickness (N=2):

1	=	Top Surface Temperature
2	=	Bottom Surface Temperature

Quadratic distribution through the thickness (N=3):

1	=	Top Surface Temperature
2	=	Bottom Surface Temperature
3	=	Mid Surface Temperature

Linear distribution through every layer in Composites with M layers (N=M+1):

1	=	Top Surface Temperature
2	=	Temperature at Interface between Layer 1 and Layer 2
3	=	Temperature at Interface between Layer 2 and Layer 3 etc
N = M+1	=	Bottom Surface Temperature

Quadratic distribution through every layer in Composites with M layers (N=2*M+1):

1	=	Top Surface Temperature
2	=	Temperature at Interface between Layer 1 and Layer 2
3	=	Mid Temperature of Layer 1
4	=	Temperature at Interface between Layer 2 and Layer 3
5	=	Mid Temperature of Layer 2
...		
N-1 = 2*M	=	Bottom Surface Temperature
N = 2*M+1	=	Mid Temperature of Layer N

Piecewise Quadratic distribution through thickness in Non-Composites with M layers (SHELL SECT,M) (N=M+1):

1	=	Top Surface Temperature (thickness position 0.)
X	=	Temperature at thickness position (X-1)/M
N = M+1	=	Bottom Surface Temperature (thickness position 1.

Fluxes

Two types of distributed fluxes:

Volumetric Fluxes

Flux Type	Description
1	Uniform flux per unit volume on whole element.
3	Nonuniform flux per unit volume on whole element; magnitude of flux is defined in the FLUX user subroutine.

Surface Fluxes

Flux Type	Description
2	Uniform flux per unit surface area on bottom surface.
5	Uniform flux per unit surface area on top surface.
6	Nonuniform flux per unit surface area on top surface; magnitude of flux is defined in the FLUX user subroutine.
4	Nonuniform flux per unit surface area on bottom surface; magnitude of flux is defined in the FLUX user subroutine.

Point fluxes can also be applied at the degrees of freedom and must be integrated around the circumference.

Films

Same specification as Fluxes .

Tying

Standard types 85 and 86 with axisymmetric heat transfer elements.

Shell Sect – Integration Through The Shell Thickness Direction

The number of layer through the thickness is controlled by the SHELL SECT parameter (must be an odd number) with the default of 11 layers or via the COMPOSITE model definition option. The accuracy is controlled by the number of temperature degrees of freedom through the thickness.

Output Points

Temperatures are printed out at the integration points through the thickness of the shell at the centroid or three Gaussian integration points in the plane of the shell. The first point in the thickness direction is on the surface of the positive normal.

Only one integration point per layer is written to the post file, so this element may be used in a thermal stress analysis with element type 1. This value is obtained by extrapolating to the nodes and the interpolating to the Centroid.

Joule Heating

Capability is not available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Charge

Same specifications as Fluxes .

Element 89

Thick Curved Axisymmetric Shell

This is a three-node, axisymmetric, thick-shell element, with a quadratic displacement assumption based on the global displacements and rotation. The strain-displacement relationships used are suitable for large displacements with small strains. Two-point Gaussian integration is used along the element for the stiffness calculation and three-point integration is used for the mass and pressure determination. All constitutive relations can be used with this element.

Quick Reference

Type 89

Axisymmetric, curved, thick-shell element.

Connectivity

Three nodes per element (see Figure 3-140).

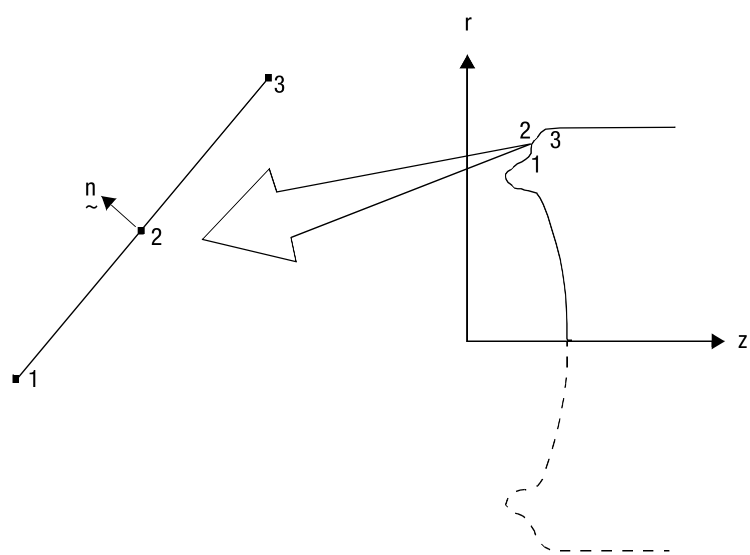


Figure 3-140 Axisymmetric, Curved Thick-Shell Element

Geometry

Linear thickness variation along length of the element. Thickness at first node of the element store in the first data field (EGEOM1).

Thickness at third node in the third data field (EGEOM3).

If EGEOM3=0, constant thickness assumed. Notice that the linear thickness variation is only taken into account if the ALL POINTS parameter is used since, in the other case, section properties formed at the centroid of the element are used for all integration points.

The second data field is not used (EGEOM2).

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

If the element needs to be offset from the user-specified position, the eighth data field (EGEOM8) is set to -2.0. In this case, an extra line containing the offset information is provided (see the GEOMETRY option in Marc Volume C: Program Input). The offset magnitudes at the two corner nodes are taken to be along the element normal and are provided in the first and second data fields of the extra line, respectively. If the interpolation flag (fifth data field) is set to 1, then the offset value at the midside node is obtained by interpolating from the corresponding corner values. A uniform offset for the element can be set by providing the offset magnitude in the first data field and then setting the constant offset flag (sixth data field of the extra line) to 1.

Coordinates

1 = z
2 = r

Degrees of Freedom

1 = u = axial (parallel to symmetry axis)
2 = v = radial (normal to symmetry axis)
3 = ϕ = right hand rotation

Tractions

Distributed loads selected with IBODY are as follows:

Load Type	Description
0	Uniform pressure.
5	Nonuniform pressure.

Pressure assumed positive in the negative normal (-n) direction.

Load Type	Description
1	Uniform load in 1 direction (force per unit area).
2	Uniform load in 2 direction (force per unit area).
3	Nonuniform load in 1 direction (force per unit area).
4	Nonuniform load in 2 direction (force per unit area).
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.

Load Type	Description
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

In the nonuniform cases (IBODY = 3, 4, or 5), the load magnitude must be supplied by the FORCEM user subroutine.

Concentrated loads applied at the nodes must be integrated around the circumference.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Generalized strains are:

1 = ε_s = meridional membrane (stretch)

2 = ε_θ = circumferential membrane (stretch)

3 = γ_t = transverse shear strain

4 = χ_s = meridional curvature

5 = χ_θ = circumferential curvature

Output Of Stresses

Stresses are output at the integration points through the thickness of the shell. The first point is on the surface of the positive normal. The positive normal is opposite to the direction of positive pressure as shown in Figure 3-140.

1 = meridional stress
2 = circumferential stress
3 = transverse shear.

Transformation

The degrees of freedom can be transformed to local directions.

Output Points

Centroid or two Gaussian integration points. The first Gaussian integration point is close to the first node as defined in the connectivity data. The second integration point is close to the third node as defined in the connectivity data.

Section Stress Integration

Simpson's rule is used to integrate through the thickness. Use the SHELL SECT parameter to specify the number of layers. This number must be odd. Three points are enough for linear material response. Seven points are enough for simple plasticity or creep analysis. Eleven points are enough for complex plasticity or creep (e.g., thermal plasticity). The default is 11 points.

Updated Lagrange Procedure And Finite Strain Plasticity

Capability is available – output of stress and strain in meridional and circumferential direction. Thickness is updated.

Note:	Shell theory only applies if strain variation through the thickness is small.

Note, however, that since the curvature calculation is linearized, you have to select you load steps such that the rotations increments remain small within a load step.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 87. See Element 87 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered as a design variable.

Element 90

Thick Curved Axisymmetric Shell – for Arbitrary Loading (Fourier)

This is a three-node, thick-shell element for the analysis of arbitrary loading of axisymmetric shells. Quadratic interpolation functions are used on the global displacements and rotations. Two-point Gaussian integration is used along the element for the stiffness calculation and three-point integration is used for the mass and pressure determination. This Fourier element can only be used for linear elastic analysis. No contact is permitted with this element.

Quick Reference

Type 90

Second-order isoparametric, curved, thick-shell element for arbitrary loading of axisymmetric shells, formulated by means of the Fourier expansion technique.

Connectivity

Three nodes per element (see Figure 3-141).

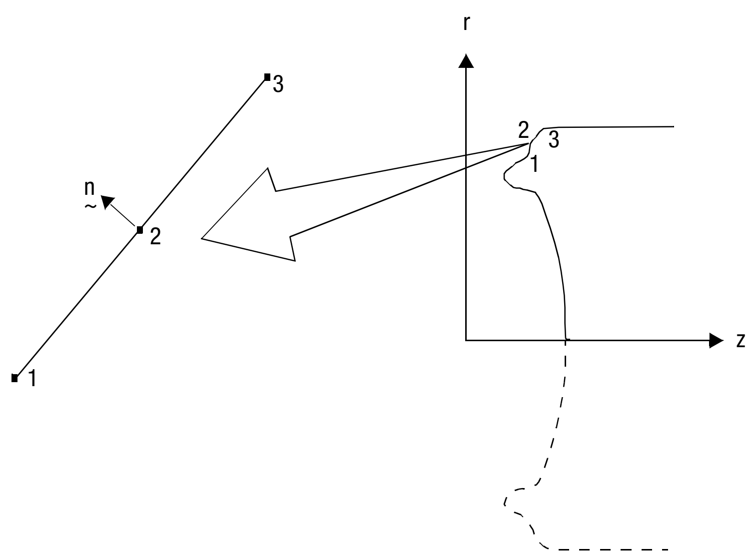


Figure 3-141 Axisymmetric, Curved Thick-Shell Element for Arbitrary Loading

Geometry

Linear thickness variation along length of the element. Thickness at first node of the element is stored in the first data field (`EGEOM1`).

Thickness at third node in the third data field (`EGEOM3`).

If `EGEOM3=0`, constant thickness assumed. Notice that the linear thickness variation is only taken into account if the ALL POINTS parameter is used since, in the other case, section properties formed at the centroid of the element are used for all integration points.

The second data field is not used (EGEOM2).

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

Coordinates

1 = z
2 = r

Degrees of Freedom

1 = u = axial (parallel to symmetry axis)
2 = v = radial (normal to symmetry axis)
3 = w = circumferential displacement
4 = ϕ = right handed rotation in the z-r plane
5 = ψ = right handed rotation about meridian

Tractions

Distributed loads selected with IBODY are as follows:

Load Type	Description
0	Uniform pressure
5	Nonuniform pressure

Pressure assumed positive in the negative normal (-n) direction (see Figure 3-141).

Load Type	Description
1	Uniform load in 1 direction (force per unit area).
2	Uniform load in 2 direction (force per unit area).
3	Nonuniform load in 1 direction (force per unit area).
4	Nonuniform load in 2 direction (force per unit area).
6	Uniform shear in 3 direction (moment per unit area).
7	Nonuniform shear in 3 direction (moment per unit area).
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Load Type	Description
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

In the nonuniform cases (IBODY = 3, 4, or 5), the load magnitude must be supplied by the FORCEM user subroutine.

Concentrated loads applied at the nodes must be integrated around the circumference.

For loads varying in the 3 (circumferential) direction, each distributed load or concentrated force can be associated with a different Fourier expansion. If no Fourier series is specified for a given loading, it is assumed to be constant around the circumference.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Generalized strains are:

1 = ε_s	= meridional membrane (stretch)
2 = ε_θ	= circumferential membrane (stretch)
3 = $\gamma_{s\theta}$	= in plane shear
4 = $\gamma_{\theta n}$	= transverse shear strain in meridional direction
5 = γ_{ns}	= transverse shear strain in circumferential direction
6 = χ_s	= meridional curvature
7 = χ_θ	= circumferential curvature
8 = $\chi_{s\theta}$	= shear curvature (twist)

Output of Stresses

Stresses are output at the integration points through the thickness of the shell. The first point is on the surface of the positive normal. The positive normal is opposite to the direction of positive pressure as shown in Figure 3-141.

1 = meridional stress
2 = circumferential stress
3 = out of plane shear
4 = circumferential transverse shear
5 = meridional transverse shear

Transformation

The degrees of freedom can be transformed to local directions.

Output Points

Centroid or two Gaussian integration points. The first Gaussian integration point is close to the first node as defined in the connectivity data. The second integration point is close to the third node as defined in the connectivity data.

Section Stress Integration

Simpson's rule is used to integrate through the thickness. Use the SHELL SECT parameter to specify the number of layers. This number must be odd. Three points are enough for linear material response. Seven points are enough for simple plasticity or creep analysis. Eleven points are enough for complex plasticity or creep (e.g., thermal plasticity). The default is 11 points.

Element 91

Linear Plane Strain Semi-infinite Element

This is a six-node, plane strain element that can be used to model an unbounded domain in one direction. This element is used in conjunction with the usual linear element. The interpolation functions are linear in the 1-2 direction, and cubic in the 2-5-3 direction. Mappings are such that the element expands to infinity. Displacements at infinity are implied to be zero; it is unnecessary to put boundary conditions at these nodes. This element does not have nonlinear capability. This element cannot be used with the CONTACT option. There is no mass matrix associated with this element.

Quick Reference

Type 91

Plane strain semi-infinite element (see Figure 3-142).

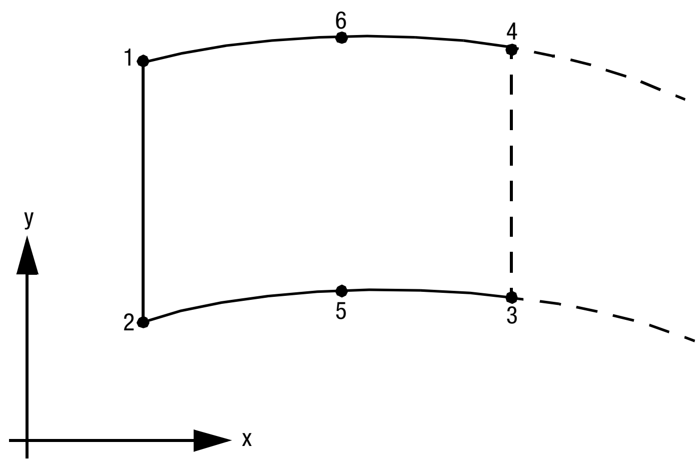


Figure 3-142 Plane Strain Semi-infinite Element

Connectivity

Six nodes per element. Counterclockwise numbering. 1-2 face should be connected to a standard element. 2-3 and 4-1 face should be either connected to another semi-infinite element or no other element. 3-4 face should not be connected to any other elements.

Coordinates

Two coordinates in the global x and y direction.

Geometry

The thickness is given in the first field `EGEOM1`.

Degrees of Freedom

Global displacement degrees of freedom.

1 = u displacement (x-direction)
2 = v displacement (y-direction)

Tractions

Distributed loads are listed below.

Load Type	Description
0	Uniform pressure on 1-2 face.
1	Nonuniform pressure on 1-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in x-direction.
4	Uniform body force in y-direction.
5	Nonuniform body force in y-direction.
6	Uniform shear force in direction 1 to 2 on 1-2 face.
7	Nonuniform shear force in direction 1 to 2 on 1-2 face.
8	Uniform pressure on 2-5-3 face.
9	Nonuniform pressure on 2-5-3 face.
12	Uniform pressure on 3-6-1 face.
13	Nonuniform pressure on 3-6-1 face.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Stress and Strains

1=xx
2=yy
3=zz
4=xy

Output Points

Centroid or six Gaussian integration points (see Figure 3-143).

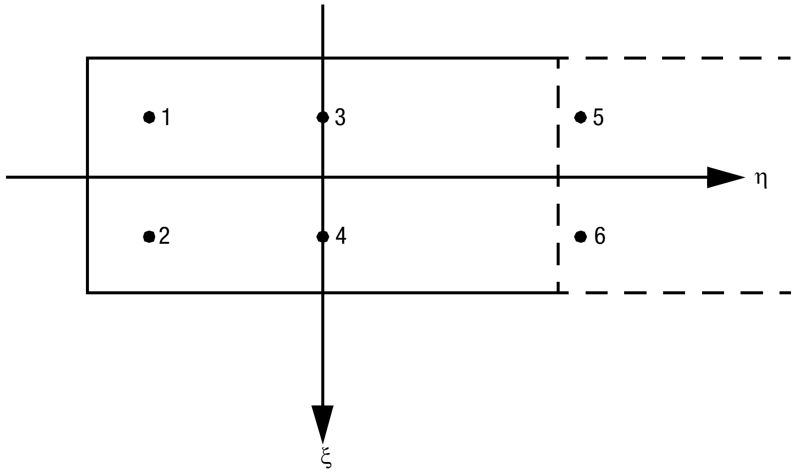


Figure 3-143 Integration Point Locations

Transformations

Two global degrees of freedom can be transformed into a local system.

Tying

Use the UFORMSN user subroutine.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 101. See Element 101 for a description of the conventions used for entering the flux and film data for this element.

Note: No boundary conditions at infinity are required. The locations of nodes 5 and 6 express the decay of functions. Along the edge 2-5-3, the interpolation function is given by:

$$f(\eta) = \frac{-\eta(\alpha-\eta)(1-\eta)}{2(1+\alpha)}f_2 + \frac{\eta(1-\eta^2)}{\alpha(1-\alpha^2)}f_3 + \frac{(1-\eta^2)(\alpha-\eta)}{\alpha}f_5,$$

in which α is the η -coordinate of node 3, defined as ,

$$\alpha = \frac{\|\underset{\sim}{x}_3 - \underset{\sim}{x}_5\|}{\|\underset{\sim}{x}_3 - \underset{\sim}{x}_5\| + 2\|\underset{\sim}{x}_5 - \underset{\sim}{x}_2\|}$$

where $\underset{\sim}{x}_i$ denotes the position vector of node i. So if node 5 is positioned at the middle of edge 2-5-3, .

$$\alpha = \frac{1}{3}$$

Element 92

Linear Axisymmetric Semi-infinite Element

This is a six-node, axisymmetric element that can be used to model an unbounded domain in one direction. This element is used in conjunction with the usual linear element. The interpolation functions are linear in the 1-2 direction and cubic in the 2-5-3 direction. Mappings are such that the element expands to infinity. Displacements at infinity are implied to be zero; it is unnecessary to put boundary conditions at these nodes. This element does not have nonlinear capability. This element cannot be used with the CONTACT option. There is no mass matrix associated with this element.

Quick Reference

Type 92

Axisymmetric semi-infinite element (see Figure 3-144).

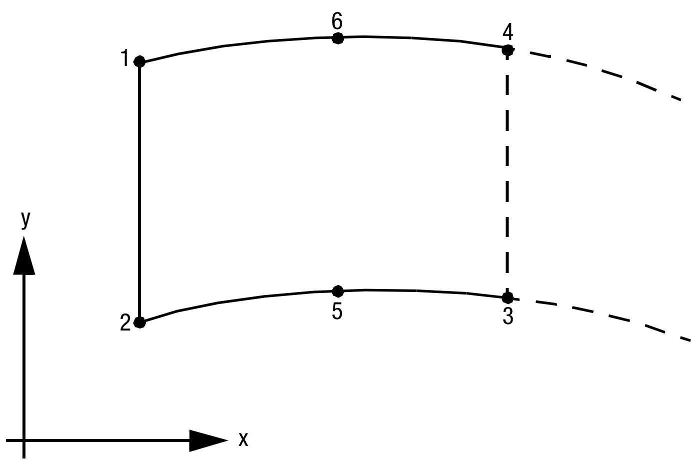


Figure 3-144 Axisymmetric Semi-infinite Element

Connectivity

Six nodes per element. Counterclockwise numbering. 1-2 face should be connected to a standard element. 3-4 face should not be connected to any elements.

Coordinates

Two coordinates in the global z and r directions.

Geometry

Not necessary for this element.

Degrees of Freedom

Global displacement degrees of freedom.

1 = u displacement (z-direction)
2 = v displacement (r-direction)

Tractions

Distributed loads are listed below:

Load Type	Description
0	Uniform pressure on 1-2 face.
1	Nonuniform pressure on 1-2 face.
2	Uniform body force in z-direction.
3	Nonuniform body force in z-direction.
4	Uniform body force in r-direction.
5	Nonuniform body force in r-direction.
6	Uniform shear force in direction 1 to 2 on 1-2 face.
7	Nonuniform shear force in direction 1 to 2 on 1-2 face.
8	Uniform pressure on 2-5-3 face
9	Nonuniform pressure on 2-5-3 face.
12	Uniform pressure on 3-7-1 face.
13	Nonuniform pressure on 3-6-1 face.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Stress and Strain

1 = zz
2 = rr
3 = θθ
4 = zr

Output Points

Centroid or six Gaussian integration points (see Figure 3-145).

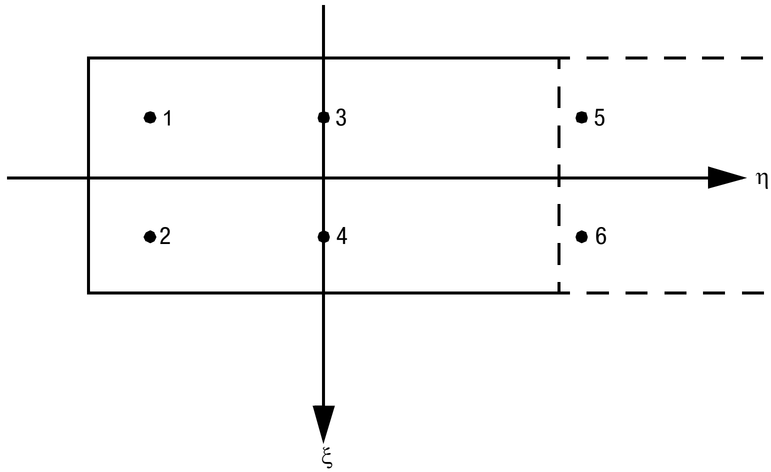


Figure 3-145 Integration Point Location

Transformations

Two global degrees of freedom can be transformed into a local system.

Tying

Use the UFORMSN user subroutine.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 102. See Element 102 for a description of the conventions used for entering the flux and film data for this element.

Note:	No boundary conditions at infinity are required. Locations of nodes 5 and 6 express the decay of functions. See the description of element type 91 for more information.

Element 93

Quadratic Plane Strain Semi-infinite Element

This is a nine-node, plane strain, semi-infinite element that can be used with usual quadratic elements to solve the problems involving unbounded domains. Interpolation functions are parabolic in 1-5-2 direction, and cubic in 2-6-3 direction. Mappings are such that the element expands to infinity. Displacements at the infinity are implied to be zero. This element does not have any nonlinear capability. This element cannot be used with the CONTACT option. There is no mass matrix associated with this element.

Quick Reference

Type 93

Plane strain, semi-infinite element (see Figure 3-146).

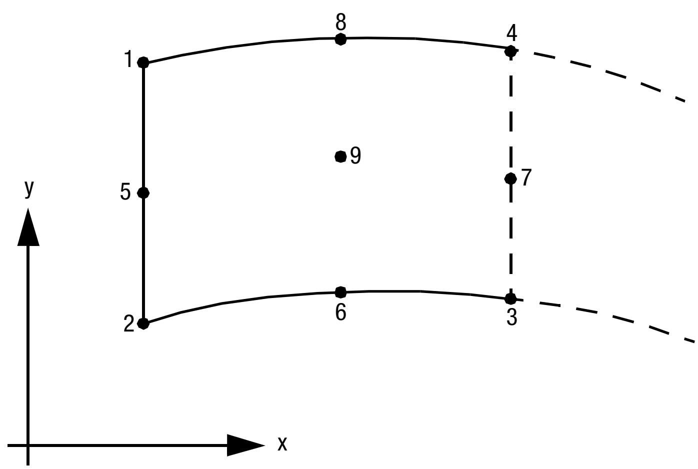


Figure 3-146 Plane Strain Semi-infinite Element

Connectivity

Nine nodes per element. Counterclockwise numbering. 1-5-2 face should be connected to a standard element. 3-7-4 face should not be connected to any elements.

Coordinates

Two coordinates in the global x and y directions.

Geometry

The thickness is given in the first field, `EGEOM1`.

Degrees of Freedom

Global displacement degrees of freedom.

1 = u displacement (x-direction)
2 = v displacement (y-direction)

Tractions

Distributed loads are listed below:

Load Type	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in y-direction.
4	Uniform body force in y-direction.
5	Nonuniform body force in y-direction.
6	Uniform shear force in direction 1-5-2 on 1-5-2 face.
7	Nonuniform shear force in direction 1-5-2 on 1-5-2 face.
8	Uniform pressure on 2-6-3 face.
9	Nonuniform pressure on 2-6-3 face.
12	Uniform pressure on 4-7-1 face.
13	Nonuniform pressure on 4-7-1 face.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Stresses and Strains

1 = xx
2 = yy
3 = zz
4 = xy

Output Points

Centroid or nine Gaussian integration points (see Figure 3-147).

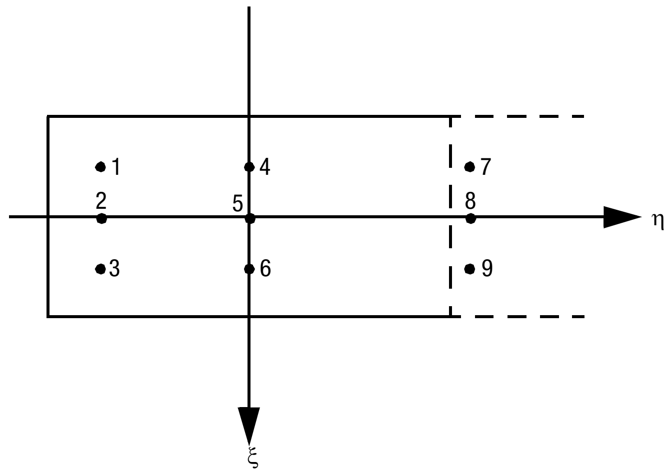


Figure 3-147 Integration Point Locations

Transformations

Two global degrees of freedom can be transformed into a local system.

Tying

Use the UFORMSN user subroutine.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 105. See Element 105 for a description of the conventions used for entering the flux and film data for this element.

Note:	No boundary conditions at infinity are required. Locations of nodes 6, 8, and 9 express the decay of functions.

Element 94

Quadratic Axisymmetric Semi-infinite Element

This is a nine-node, axisymmetric, semi-infinite element that can be used with the usual quadratic elements to solve the problems involving unbounded domains. Interpolation functions are parabolic in 1-5-2 direction, and cubic in 2-6-3 direction. Mappings are such that the element expands to infinity. Displacements at the infinity are implied to be zero. This element does not have any nonlinear capability. This element cannot be used with the CONTACT option. There is no mass matrix associated with this element.

Quick Reference

Type 94

Axisymmetric, semi-infinite element.

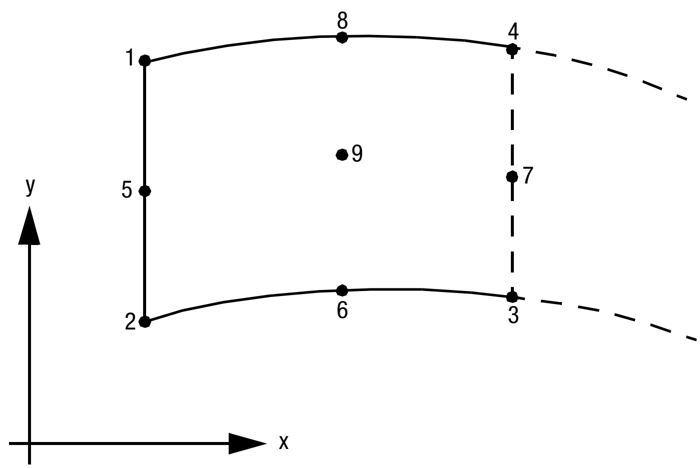


Figure 3-148 Axisymmetric Semi-infinite Element

Connectivity

Nine nodes per element. Counterclockwise numbering. 1-5-2 face should be connected to a standard element. 3-7-4 face should not be connected to any elements.

Coordinates

Two coordinates in the global z and r directions.

Geometry

No geometry option is necessary.

Degrees of Freedom

Global displacement degrees of freedom.

1 = u displacement (z-direction)
2 = v displacement (r-direction)

Tractions

Distributed loads are listed below:

Load Type	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in z-direction.
3	Nonuniform body force in z-direction.
4	Uniform body force in r-direction.
5	Nonuniform body force in r-direction.
6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
7	Nonuniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
8	Uniform pressure on 2-6-3 face.
9	Nonuniform pressure on 2-6-3 face.
12	Uniform pressure on 4-7-1 face.
13	Nonuniform pressure on 4-7-1 face.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Stresses and Strains

1 = zz
2 = rr
3 = θθ
4 = zr

Output Points

Centroid or nine Gaussian integration points (Figure 3-149).

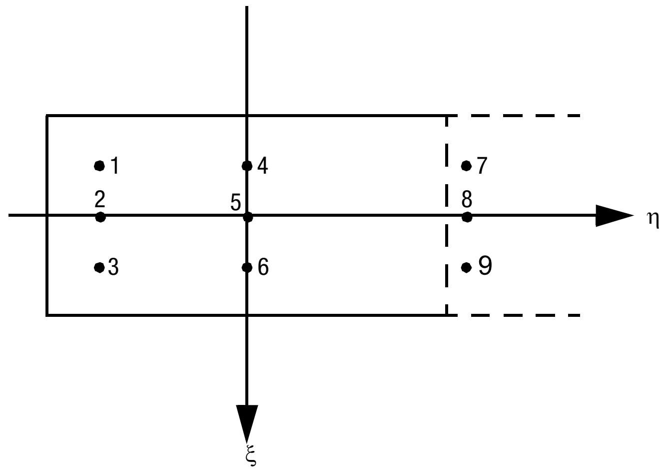


Figure 3-149 Integration Point Locations

Transformations

Two global degrees of freedom can be transformed into a local system.

Tying

Use the UFORMSN user subroutine.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 104. See Element 104 for a description of the conventions used for entering the flux and film data for this element.

Note:	No boundary conditions at infinity are required. Locations of nodes 6, 8, and 9 express the decay of functions.

Element 95

Axisymmetric Quadrilateral with Bending

This is the same formulation as element type 10, with bending effects included. Element type 95 provides a capability to do efficient analysis of axisymmetric structures deforming axisymmetrically and in bending. The elements are based on the usual (isoparametric) displacement formulation in the z-r plane, whereas in the circumferential direction sinusoidal variation is assumed, which can be expressed by:

$$u_z(\theta) = u_z(1 + \cos\theta)/2 + \bar{u}_z(1 - \cos\theta)/2$$

$$u_r(\theta) = u_r(1 + \cos\theta)/2 + \bar{u}_r(1 - \cos\theta)/2$$

$$u_\theta(\theta) = u_\theta \sin\theta$$

The element is integrated numerically in the z-r plane using the usual Gaussian quadrature formulas, whereas numerical integration with an equidistant scheme is used along the circumference. The number of points along the circumference is chosen with the SHELL SECT parameter, and must be at least equal to 3. For linear elastic material behavior, this element furnishes “exact” results (for the circumferential variation) for axisymmetric and bending deformation even with the minimum number of circumferential integration points.

Because of the numerical integration scheme, the elements can also be used if material nonlinearity (creep or plasticity) plays a role. It should be noted that the exact solution does not necessarily contain the sinusoidal variation as given above, and, in that sense, the solution obtained is an approximate one. However, experience obtained so far indicates that for thick-walled members, where ovalization of the cross section does not play a significant role, the solution is sufficiently accurate for most practical purposes. Note that if nonlinear effects are present, the number of integration points along the circumference should be at least 5, and more if desired.

This element cannot be used with the CONTACT option; use gap element type 97 instead.

Quick Reference

Type 95

Axisymmetric, arbitrary ring with a quadrilateral cross section and bending effects included. This is achieved by including additional degrees of freedom representing the displacements at the point 180° along the circumference.

Connectivity

Four nodes per element. Node numbering follows right-handed convention (counterclockwise). See Figure 3-150.

Geometry

No geometry input is required for this element.

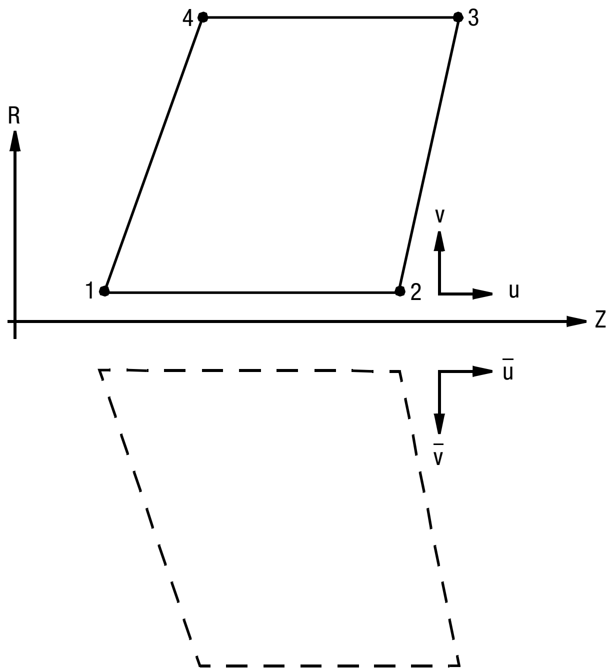


Figure 3-150 Axisymmetric Ring with Bending

Coordinates

Two coordinates in the global z and r directions.

Degrees of Freedom

Global displacement degrees of freedom:

1 = u = z displacement (along symmetry axis).
2 = v = radial displacement.
3 = $\bar{u}$ = z displacement of reverse side.
4 = $\bar{v}$ = radial displacement of reverse side.
5 = w = circumferential displacement at 90° angle.

Distributed Loads

Load types for distributed loads are as follows:

Load Type	Description
0,50	Uniform pressure distributed on 1-2 face of the element.
1,51	Uniform body force per unit volume in first coordinate direction.
2,52	Uniform body force by unit volume in second coordinate direction.
3,53	Nonuniform pressure on 1-2 face of the element.

Load Type	Description
4,54	Nonuniform body force per unit volume in first coordinate direction.
5,55	Nonuniform body force per unit volume in second coordinate direction.
6,56	Uniform pressure on 2-3 face of the element.
7,57	Nonuniform pressure on 2-3 face of the element.
8,58	Uniform pressure on 3-4 face of the element.
9,59	Nonuniform pressure on 3-4 face of the element.
10,60	Uniform pressure on 4-1 face of the element.
11,61	Nonuniform pressure on 4-1 face of the element.
20	Uniform shear force on 1-2 face in the 1-2 direction.
21	Nonuniform shear force on side 1-2.
22	Uniform shear force on side 2-3 (positive from 2 to 3).
23	Nonuniform shear force on side 2-3.
24	Uniform shear force on side 3-4 (positive from 3 to 4).
25	Nonuniform shear force on side 3-4.
26	Uniform shear force on side 4-1 (positive from 4 to 1).
37	Nonuniform shear force on side 4-1.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

All pressures are positive when directed into the element. In addition, point loads can be applied at the nodes. The load types 0-11 correspond to axisymmetric loading. The load type 50-61 correspond to “bending” loads. In that case, the circumferential variation of the distributed load is equal to $p(\theta) = p_o \cos(\theta)$.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid of the element or at the Gauss points in global coordinates is:

$1 = \varepsilon_{zz}$

$2 = \varepsilon_{rr}$

$3 = \varepsilon_{\theta\theta}$

$4 = \gamma_{rz}$

$5 = \gamma_{z\theta}$

$6 = \gamma_{\theta r}$

Output of Stresses

Same as for Output of Strains .

Transformation

The transformation on degrees of freedom 3 and 4 are the same as on degrees of freedom 1 and 2. Four global degrees of freedom can be transformed into local coordinates.

Tying

Use the UFORMSN user subroutine.

Output Points

Output is available at the centroid or at the 4 Gaussian points shown in Figure 3-151.

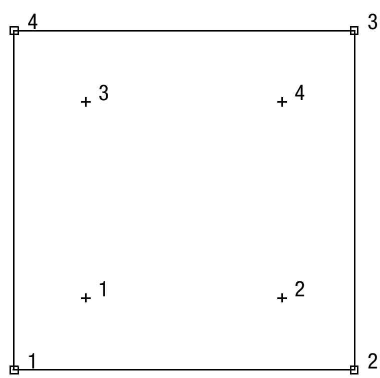


Figure 3-151 Integration Point Locations

Integration Along Circumference

The element is integrated numerically in the circumferential direction. The number of integration points is given on the SHELL SECT parameter. The points are equidistant.

Large Displacement

This element has only geometrically linear behavior. Neither the LARGE DISP or the LARGE STRAIN parameter has any effect on this element.

Element 96 Axisymmetric, Eight-node Distorted Quadrilateral with Bending

This element follows the same second-order isoparametric formulation as the regular axisymmetric element type 28, but has the possibility to deform during bending. Element type 96 provides a capability to do efficient analysis of axisymmetric structures deforming axisymmetrically and in bending. The elements are based on the usual (isoparametric) displacement formulation in the z-r plane, whereas in the circumferential direction sinusoidal variation is assumed, which can be expressed by:

$$u_z(\theta) = u_z(1 + \cos\theta)/2 + \bar{u}_z(1 - \cos\theta)/2$$

$$u_r(\theta) = u_r(1 + \cos\theta)/2 + \bar{u}_r(1 - \cos\theta)/2$$

$$u_\theta(\theta) = u_\theta \sin\theta$$

The element is integrated numerically in the z-r plane using the usual Gaussian quadrature formulas, whereas numerical integration with an equidistant scheme is used along the circumference. The number of points along the circumference is chosen with the SHELL SECT parameter, and must be at least equal to three. For linear elastic material behavior, this element furnishes “exact” results (for the circumferential variation) for axisymmetric and bending deformation even with the minimum number of circumferential integration points.

Because of the numerical integration scheme, the elements can also be used if material nonlinearity (creep or plasticity) plays a role. It should be noted that the exact solution does not necessarily contain the sinusoidal variation as given above, and in that sense the solution obtained is an approximate one. However, experience obtained so far indicates that for thick-walled members, where ovalization of the cross section does not play a significant role, the solution is sufficiently accurate for most practical purposes Note that if nonlinear effects are present, the number of integration points along the circumference should at least be 5, and more if desired.

This element cannot be used with the CONTACT option; use gap element type 97 instead.

Quick Reference

Type 96

Second order, isoparametric, distorted quadrilateral. Axisymmetric formulation with bending deformation.

Connectivity

Eight nodes per element.

Corners numbered first, in counterclockwise order (right-handed convention in z-r plane). Then fifth node between first and second. The sixth node between second and third, etc. See Figure 3-152.

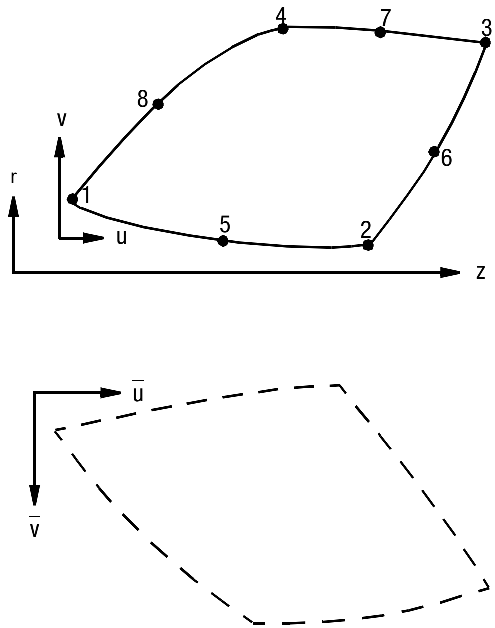


Figure 3-152 Axisymmetric Ring with Bending

Geometry

No geometry input for this element.

Coordinates

Two global coordinates, z and r, at each node.

Degrees of Freedom

Two at each node:

1 = u = z-direction displacement (axial)
2 = v = r-direction displacement (radial)

3 = $\bar{u}$ = z-direction displacement at reverse side (axial)

4 = $\bar{v}$ = r-direction displacement at reverse side (radial)
5 = w = circumferential displacement at 90° angle.

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type	Description
0,50	Uniform pressure on 1-5-2 face.
1,51	Nonuniform pressure on 1-5-2 face.
2,52	Uniform body force in x-direction.
3,53	Nonuniform body force in the x-direction.
4,54	Uniform body force in y-direction.
5,55	Nonuniform body force in the y-direction.
6,56	Uniform shear force in direction 1⇒5⇒2 on 1-5-2 face.
7,57	Nonuniform shear in direction 1⇒5⇒2 on 1-5-2 face.
8,58	Uniform pressure on 2-6-3 face.
9,59	Nonuniform pressure on 2-6-3 face.
10,60	Uniform pressure on 3-7-4 face.
11,61	Nonuniform pressure on 3-7-4 face.
12,62	Uniform pressure on 4-8-1 face.
13,63	Nonuniform pressure on 4-8-1 face.
20	Uniform shear force on 1-5-2 face in the 1⇒5⇒2 direction.
21	Nonuniform shear force on side 1-5-2.
22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
23	Nonuniform shear force on side 2-6-3.
24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
25	Nonuniform shear force on side 3-7-4.
26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
27	Nonuniform shear force on side 4-8-1.
90	Torsional load on 1-5-2 face.
91	Nonuniform torsional load on 1-5-2 face.
92	Torsional load on 2-6-3 face.
93	Nonuniform torsional load on 2-6-3 face.
94	Torsional load on 3-7-4 face.
95	Nonuniform torsional load on 3-7-4 face.
96	Torsional load on 4-8-1 face.
97	Nonuniform torsional load on 4-8-1 face.

Load Type	Description
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

Concentrated nodal loads must be the value of the load integrated around the circumference.

The load types 0-13 correspond to axisymmetric loading, whereas the load types 50-63 correspond to “bending” loading. In that case, the circumferential variation of the distributed load is equal to $p(\theta) = p_o \cos(\theta)$.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Six strain components are printed in the order listed below:

1 = ε_{zz}, direct

2 = ε_{rr}, direct

3 = $\varepsilon_{\theta\theta}$, hoop direct

4 = γ_{zr}, shear in an axial section

5 = $\gamma_{r\theta}$, shear in a radial section

6 = $\gamma_{\theta z}$, shear in a circumferential section

Output of Stresses

Output for stresses is the same as for Output of Strains .

Transformation

Only in z-r plane. The transformation degrees on freedom 3 and 4 is the same as on degrees of freedom 1 and 2.

Tying

Use the UFORMSN user subroutine.

Output Points

If the CENTROID parameter is used, the output occurs at the centroid of the element, given as point 5 in Figure 3-153.

If the ALL POINTS parameter is used, nine output points are given, as shown in Figure 3-153. This is the usual option for a second-order element.

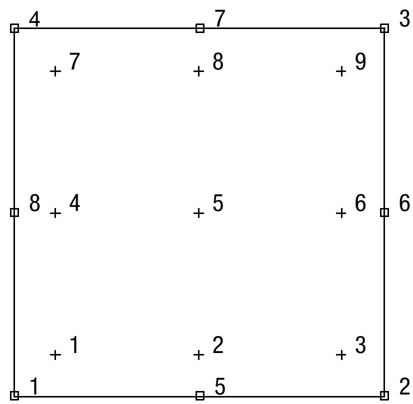


Figure 3-153 Integration Point Locations

Integration Along Circumference

The element is integrated numerically in the circumferential direction. The number of integration points is given on the SHELL SECT parameter. The points are equidistant.

Large Displacement

This element has only geometrically linear behavior. Neither the LARGE DISP or the UPDATE parameter has any effect on this element.

Element 97

Special Gap and Friction Link for Bending

This element provides gap and friction capability for the bending element types 95 and 96. The element works in a similar fashion as the regular fixed direction gap 12; however, extra degrees of freedom have been included to account for independent contact and friction at either side of the bending elements. In contrast to the regular gap element, however, the element cannot be used in full three-dimensional situations, since gap and friction motion can only occur in the 1-2 plane. In all other important aspects, the element is identical to element type 12. Lagrange multipliers are used to enforce the constraint conditions due to contact and friction, and the iterative algorithm is also unchanged. It should be noted that for each element two independent contact and friction conditions can occur, which both must satisfy the convergence criteria.

Quick Reference

Type 97

Four node gap and friction link with double contact and friction conditions. It is designed specifically for use with element types 95 and 96.

Connectivity

Four-nodes per element. Nodes 1 and 4 are the end of the link to connect to the rest of the structure. Node 2 is the “gap” node associated with the contact conditions between the end nodes. Node 3 is the “friction” node associated with the friction conditions between the end nodes.

Coordinates

Nodes 1 and 4 are the physical positions of the end nodes. For use with the axisymmetric bending elements, the first is the z-coordinate and the second the r-coordinate.

Node 2 is the gap direction cosine. Only two values (n_z and n_r) need to be entered because the element is always located in the 1-2 plane. If no values are entered, Marc calculates the direction as:

$$\underset{\sim}{n} = (\underset{\sim}{x}_4 - \underset{\sim}{x}_1) / |\underset{\sim}{x}_4 - \underset{\sim}{x}_1|$$

Node 3 is the friction direction cosine. Only two values (t_z and t_r) are used since the element is located in the 1-2 plane. It is not necessary to enter these directions since they are uniquely determined. The direction must be orthogonal to the gap direction. Marc calculates these at $t_z = n_r$, $t_r = -n_z$.

Gap Data

First data field:	The closure distance U_{c1}. Note that since the initial geometry is assumed to be symmetric, the same closure distance is used on either side.
Second data field:	The coefficient of friction μ.

Third data field:	This field is used to define the elastic stiffness (spring stiffness) of the closed gap in the gap direction. If the field is left blank, the gap is assumed rigid if closed.
Fourth data field:	This field is used to define the elastic stiffness of the closed, nonslipping gap in the friction direction. If this entry is left blank, the nonslipping closed gap is assumed rigid in the friction direction. This only applies if nonzero coefficient of friction is used.

Degrees Of Freedom

Nodes 1 and 4	$1 = u_z$	=	axial displacement associated with the first gap.
	$2 = u_r$	=	radial displacement associated with the first gap.
	$3 = \bar{u}_z$	=	axial displacement associated with second gap (opposite side of axisymmetric (opposite side of axisymmetric structure).structure).
	$4 = \bar{u}_r$	=	radial displacement associated with second gap.
Node 2	$1 = N$	=	normal forces in first gap.
	$3 = \bar{N}$	=	normal forces in second gap.
Note: Degrees of freedom 2 and 4 are not used.			
Node 3	$1 = F$	=	friction force in first gap.
	$2 = s$	=	accumulated slip in first gap.
	$3 = \bar{F}$	=	friction force in second gap.
	$4 = \bar{s}$	=	accumulated slip in second gap.

It is assumed that the circumferential variation of the contact force and friction force is of the form:

$$n = N(1 + \cos\theta) + \bar{N}(1 - \cos\theta)$$

$$f = F(1 + \cos\theta) + \bar{F}(1 - \cos\theta)$$

Transformations

The transformation option can be used for nodes 1 and/or 4. Note that the second two degrees of freedom ($\bar{u}_z$ and $\bar{u}_r$) are transformed in the same way as the first two degrees of freedom.

Element 98

Elastic or Inelastic Beam with Transverse Shear

This is a straight beam in space which includes transverse shear effects with linear elastic material response as its standard material response, but it also allows nonlinear elastic and inelastic material response. The large displacement formulation only affects the axial strain; large curvature changes are neglected. Linear interpolation is used for the axial and the transverse displacements as well as for the rotations.

This element can be used to model linear or nonlinear elastic response by direct entry of the cross-section properties. Nonlinear elastic response can be modeled when the material behavior is given in the UBEAM user subroutine (see Marc Volume D: User Subroutines and Special Routines). This element can also be used to model inelastic and nonlinear elastic material response when employing numerical integration over the cross section. Standard cross sections and arbitrary cross sections are entered through the BEAM SECT parameter if the element is to use numerical cross-section integration. Inelastic material response includes plasticity models, creep models, and shape memory models, but excludes powder models, soil models, concrete cracking models, and rigid plastic flow models. Elastic material response includes isotropic elasticity models and NLELAST nonlinear elasticity models, but excludes finite strain elasticity models like Mooney, Ogden, Gent, Arruda-Boyce, Foam, and orthotropic or anisotropic elasticity models. With numerical integration, the HYPELA2 user subroutine can be used to model arbitrary nonlinear material response (see Note). Arbitrary sections can be used in a pre-integrated way. In that case, only linear elasticity and nonlinear elasticity through the UBEAM user subroutine are available.

Geometric Basis

The element uses a local (x,y,z) set for section properties. Local x and y are the principal axes of the cross section. Local z is along the beam axis. Using fields 4, 5, and 6 in the GEOMETRY option, a vector in the plane of the local x-axis and the beam axis must be specified. If no vector is defined here, the local coordinate system can alternatively be defined by the global (x,y,z) coordinates at the two nodes and by (x_1, x_2, x_3), a point in space which locates the local x-axis of the cross section. This axis lies in the plane defined by the beam nodes and this point, pointing from the beam axis toward the point. The local x-axis is normal to the beam axis.

The local z-axis goes from node 1 to node 2, and the local y-axis forms a right-handed set with local x and z.

Numerical Integration

The element uses a one-point integration scheme. This point is at the midspan location. This leads to an exact calculation for bending and a reduced integration scheme for shear. The mass matrix of this element is formed using three-point Gaussian integration.

Quick Reference

Type 98

Elastic straight beam. Linear interpolation for displacements and rotations. Transverse shear included.

Connectivity

Two nodes. The local z-axis goes from the first node to the second node (see Figure 3-154).

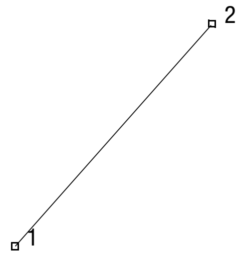


Figure 3-154 Elastic Beam Element

Geometry

First geometry data field	A	area, or enter a zero if beam definition given through the BEAM SECT parameter.
Second geometry data field	I_{xx}	moment of inertia of section about local x-axis, or enter the section number if the BEAM SECT parameter is to be used.
Third geometry data field	I_{yy}	moment of inertia of section about local y-axis

The bending stiffnesses of the section are calculated as EI_{xx} and EI_{yy}.

The torsional stiffness of the section is calculated as $\frac{E}{2(1+\nu)}(I_{xx}+I_{yy})$.

Here, E and υ are Young's modulus and Poisson's ratio calculated as functions of temperature.

If a zero is entered in the first geometry field, Marc uses the beam section data corresponding to the section number given in the second geometry field. (Sections are defined using the BEAM SECT parameter set.) This allows specification of the torsional stiffness factor K unequal to $I_{xx} + I_{yy}$, as well as definition of the effective transverse shear areas A_x^s and A_y^s unequal to the area A, or the specification of arbitrary sections using numerical section integration. See Solid Sections in Chapter 1 for more details.

EGEOM4-EGEOM6:	Components of a vector in the plane of the local x-axis and the beam axis. The local x-axis lies on the same side as the specified vector.

If beam-to-beam contact is switched on (see CONTACT option), the radius used when the element comes in contact with other beam or truss elements must be entered in the 7th data field (EGEOM7).

For segment-segment beam contact, various flags that control the creation of patches are entered in the seventh data file (EGEOM7).

```
EGEOM7 = real(100 * NSEG + 10 * IPTCH + IESCAP)
```

where

NSEG = number of segments for solid circular and solid elliptical sections (default = 32)	
IPTCH =	Not applicable.

IESCAP = end cap/side cap flag		
	0	No end or side cap (default).
	1	End cap only.
	2	Side cap only.
	3	Average side only.
	4	End and side cap.
	5	End cap and average side.

Beam Offsets Pin Codes and Local Beam Orientation

If an offset is applied to the beam geometry or pin codes are applied to the degrees of freedom of the end point of the beams or the beam axis is defined with respect to a local coordinate system, then a code is placed in the 8th field of the 3rd data block of the GEOMETRY option (EGEOM8). This code is formed by the negative sum of ioffset, iorien, and ipin where:

ioffset	=	0	-	no offset
		1	-	beam offset
iorien	=	0	-	conventional definition of local beam orientation; beam axis given in 4th through 6th field with respect to global system
		10	-	the local beam orientation entered in the 4th through 6th field is given with respect to the coordinate system of the first beam node. This vector is transformed to the global coordinate system before any further processing.
ipin	=	0	-	no pin codes are used
		10 0	-	pin codes are used.

If ioffset = 1, an additional line is read in the GEOMETRY option (4th data block).

If ipin = 100, an additional line is read in the GEOMETRY option (5th data block)

The offset vector at the first corner node is provided in the first three fields and the offset vector at the second corner node is provided in the next three fields of the extra line. The coordinate system used for the offset vectors at the two nodes are indicated by the corresponding flags in the eighth and ninth fields of the extra line. A flag setting of 0 indicates global coordinate system, 1 indicates local element coordinate system, 2 indicates local shell normal, and 3 indicates local nodal coordinate system. When the flag setting is 1, the local beam coordinate system is constructed by taking the local z axis along the beam, the local x axis as the user specified vector (obtained from EGEOM4-EGEOM6), and the local y axis as perpendicular to both. The flag setting of 2 can be used when the beam nodes are also attached to a shell. In this case, the offset magnitude at the first and second nodes are specified in the first and fourth data fields of the extra line, respectively. The automatically calculated shell normals at the nodes are used to construct the offset vectors.

Coordinates

First three coordinates – (x, y, z) global Cartesian coordinates.

Fourth, fifth, and sixth coordinates at each node – global Cartesian coordinates of a point in space which locates the local x-axis of the cross section. This axis lies in the plane defined by the beam nodes and this point, pointing from the beam towards this point. The local x-axis is normal to the beam axis. The fourth, fifth, and sixth coordinates are only used if the local x-axis direction is not specified in the GEOMETRY option.

Degrees of Freedom

1 = u_x = global Cartesian x-direction displacement
2 = u_y = global Cartesian y-direction displacement
3 = u_z = global Cartesian z-direction displacement
4 = θ_x = rotation about global x-direction
5 = θ_y = rotation about global y-direction
6 = θ_z = rotation about global z-direction

Tractions

The four types of distributed loading are as follows:

Load Type	Description
1	Uniform load per unit length in global x-direction.
2	Uniform load per unit length in global y-direction.
3	Uniform load per unit length in global z-direction.
4	Nonuniform load per unit length; magnitude and direction supplied via the FORCEM user subroutine.
11	Fluid drag/buoyancy loading – fluid behavior specified in FLUID DRAG option.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Point loads and moments can be applied at the nodes.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output Of Strains

Generalized strain components are as follows:

Axial stretch ε
Local γ_{xz} shear
Local γ_{yz} shear
Curvature about local x-axis of cross section κ_{xx}
Curvature about local y-axis of cross section κ_{yy}
Twist about local z-axis of cross section κ_{zz}

Output of Stresses

Generalized stresses:

Axial force
Local T_x shear force
Local T_y shear force
Bending moment about x-axis of cross section
Bending moment about y-axis of cross section
Torque about beam axis

Layer stresses:

Layer stresses in the cross section are only available if the element uses numerical cross-section integration (and the section is not pre-integrated) and are only printed if explicitly requested or if plasticity is present.

$1 = \sigma_{zz}$

$2 = \tau_{zx}$

$3 = \tau_{zy}$

Transformation

Displacements and rotations can be transformed to local directions.

Tying

For interacting beam, use tying type 100 for fully moment carrying joint. Use tying type 52 for pinned joint.

Output Points

Centroidal section of the element.

Updated Lagrange Procedure and Finite Strain Plasticity

The Updated Lagrange procedure is available for this element, and should be used if the element is subjected to large rotations. Since the cross-sectional dimensions of the element are not updated, the element should not be used for finite strain applications.

Note:	Nonlinear elasticity without numerical cross-section integration can be implemented with the HYPOELASTIC option and the UBEAM user subroutine. Arbitrary nonlinear material behavior with numerical cross-section integration can be implemented with the HYPOELASTIC option and the HYPELA2 user subroutine. However, the deformation gradient, the rotation tensor, and the stretch ratios are not available in the subroutine. It is also possible to use the NLELAST option with this element.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 36. See Element 36 for a description of the conventions used for entering the flux and film data.

Design Variables

The cross-sectional area (A) and moments of inertia (I_{xx}, I_{yy}) can be considered as design variables when the cross section does not employ numerical section integration.

Element 99

Three-dimensional Link (Heat Transfer Element)

This element is a simple, linear, straight link with constant cross-sectional area. It is the heat transfer equivalent of element type 9 written to be compatible with element types 5, 13, 14, 16, 25, and 52 in a coupled analysis.

This element can be used as a convection-radiation link for the simulation of convective and/or radiative boundary conditions (known ambient temperatures) or, for the situation of cross convection and/or cross radiation. In order to use this element as a convection-radiation link, additional input data must be entered using the GEOMETRY option.

The conductivity and specific heat capacity of this element are formed using three-point Gaussian integration.

Quick Reference

Type 36

Three-dimensional, two-node, heat transfer link.

Connectivity

Two nodes per element (see Figure 3-155).

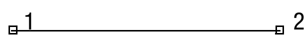


Figure 3-155 Three-dimensional Heat Link

Geometry

The cross-sectional area is input in the first data field (`EGEOM1`); the other fields are not used. If not specified, the cross-sectional area defaults to 1.0.

If the element is used as a convection-radiation link, the following data must be entered:

`EGEOM2` = emissivity

`EGEOM5` = constant film coefficient

The Stefan-Boltzmann constant and the conversion to absolute temperatures is entered through the PARAMETERS model definition option.

If the element needs to be offset from the user-specified position, the eighth data field (`EGEOM8`) is set to -1.0. In this case, an extra line containing the offset information is provided (see the GEOMETRY option in Marc Volume C: Program Input). The offset vector at the first corner node is provided in the first three fields and the offset vector at the second corner node is provided in the next three fields of the extra line. The coordinate system used for the offset vectors at the two nodes are indicated by the corresponding flags in the eighth and ninth fields of the extra line. A flag setting of 0 indicates global coordinate system, 2 indicates local shell normal, and 3 indicates local nodal coordinate system. The flag setting of 2 can be used when the beam nodes are also attached to a shell. In this case, the offset magnitude at the first and second nodes are specified in the first and fourth data fields of the extra line, respectively. The automatically calculated shell normals at the nodes are used to construct the offset vectors.

Coordinates

Three coordinates per node in the global x, y, and z direction.

Degrees of Freedom

One degree of freedom per node:

1 = temperature

Fluxes

Distributed fluxes according to value of IBODY are as follows:

Flux Type	Description
0	Uniform flux on first node (per cross-sectional area).
1	Volumetric flux on entire element (per volume).
2	Nonuniform flux given in the FLUX user subroutine on first node (per cross-sectional area).
3	Nonuniform volumetric flux given in the FLUX user subroutine on entire element (per volume).

Films

Same specification as Fluxes .

Tying

Use the UFORMSN user subroutine.

Joule Heating

Capability is available.

Current

Same specification as Fluxes .

Output Points

Either three values or a single point if the CENTROID parameter is used.

Element 100

Three-dimensional Link (Heat Transfer Element)

This element is a simple, linear, straight link with constant cross-sectional area. It is the heat transfer equivalent of element type 9 written to be compatible with element types 76, 77, 78, and 79 in a coupled analysis.

This element can be used as a convection-radiation link for the simulation of convective and/or radiative boundary conditions (known ambient temperatures) or, for the situation of cross convection and/or cross radiation. In order to use this element as a convection-radiation link, additional input data must be entered using the GEOMETRY option.

The conductivity and specific heat capacity of this element are formed using two-point Gaussian integration.

Quick Reference

Type 36

Three-dimensional, two-node, heat transfer link.

Connectivity

Two nodes per element (see Figure 3-156).

Figure 3-156 Three-dimensional Heat Link

Geometry

The cross-sectional area is input in the first data field (`EGEOM1`); the other fields are not used. If not specified, the cross-sectional area defaults to 1.0.

If the element is used as a convection-radiation link, the following data must be entered:

`EGEOM2` = emissivity

`EGEOM5` = constant film coefficient

The Stefan-Boltzmann constant and the conversion to absolute temperatures is entered through the PARAMETERS model definition option.

If the element needs to be offset from the user-specified position, the eighth data field (`EGEOM8`) is set to -1.0. In this case, an extra line containing the offset information is provided (see the GEOMETRY option in Marc Volume C: Program Input). The offset vector at the first corner node is provided in the first three fields and the offset vector at the second corner node is provided in the next three fields of the extra line. The coordinate system used for the offset vectors at the two nodes are indicated by the corresponding flags in the eighth and ninth fields of the extra line. A flag setting of 0 indicates global coordinate system, 2 indicates local shell normal, and 3 indicates local nodal coordinate system. The flag setting of 2 can be used when the beam nodes are also attached to a shell. In this case, the offset magnitude at the first and second nodes are specified in the first and fourth data fields of the extra line, respectively. The automatically calculated shell normals at the nodes are used to construct the offset vectors.

Coordinates

Three coordinates per node in the global x, y, and z direction.

Degrees of Freedom

One degree of freedom per node:

1 = temperature

Fluxes

Distributed fluxes according to value of IBODY are as follows:

Flux Type	Description
0	Uniform flux on first node (per cross-sectional area).
1	Volumetric flux on entire element (per volume).
2	Nonuniform flux given in the FLUX user subroutine on first node (per cross-sectional area).
3	Nonuniform volumetric flux given in the FLUX user subroutine on entire element (per volume).

Films

Same specification as Fluxes .

Tying

Use the UFORMSN user subroutine.

Joule Heating

Capability is available.

Current

Same specification as Fluxes .

Output Points

Either two values or a single point if the CENTROID parameter is used.

Element 101

Six-node Plane Semi-infinite Heat Transfer Element

This is a six-node planar semi-infinite heat transfer element that can be used to model an unbounded domain in one direction. Element type 101 is used in conjunction with the usual linear element. The interpolation functions are linear in the 1-2 direction, and cubic in the 2-5-3 direction. Mappings are such that the element expands to infinity. Both the conductivity and the capacity matrices are numerically integrated using six (2 x 3) integration points. This element cannot be used with the CONTACT option. There is no specific heat matrix associated with this element.

In addition, this element can be used for an electrostatic or a magnetostatic problem. A description of these two options can be found in Marc Volume A: Theory and User Information .

Quick Reference

Type 101

Six-node planar semi-infinite heat transfer element.

Connectivity

Six nodes per element (see Figure 3-157).

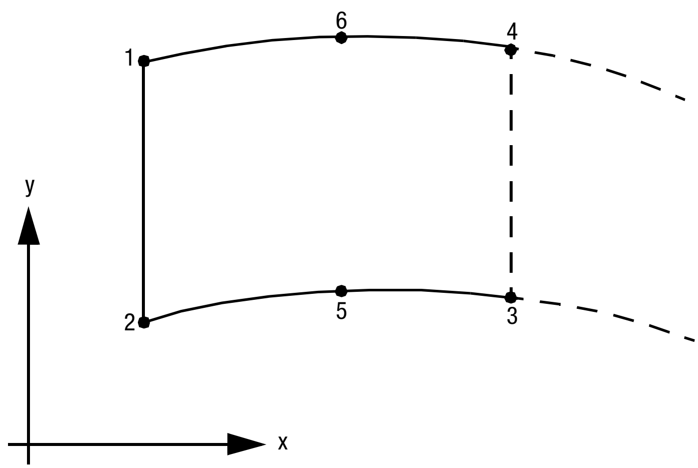


Figure 3-157 Six-node Planar Semi-infinite Heat Transfer Element

Counterclockwise numbering. The 1-2 face should be connected to a standard four-node plane heat transfer element and the 2-3 and 4-1 faces should be either connected to another six-node plane semi-infinite heat transfer element or be a free surface. The 3-4 face should not be connected to any other elements.

Coordinates

Two coordinates in the global x- and y-directions.

Geometry

Thickness of the element is given in the first field EGEOM1.

Degrees of Freedom

1 = temperature (heat transfer)
1 = potential (electrostatic)
1 = potential (magnetostatic)

Fluxes

Distributed fluxes are listed below:

Flux Type	Description
0	Uniform flux per unit area on 1-2 face of the element.
1	Nonuniform flux per unit area on 1-2 face of the element; magnitude given in the FLUX. user subroutine
2	Uniform flux per unit volume on whole element.
3	Nonuniform flux per unit volume on whole element; magnitude given in the FLUX user subroutine.
8	Uniform flux per unit area on 2-5-3 face of the element.
9	Nonuniform flux per unit area on 2-5-3 face of the element; magnitude given in the FLUX user subroutine.
12	Uniform flux per unit area on 4-6-1 face of the element.
13	Nonuniform flux per unit area on 4-6-1 face of the element; magnitude given in the FLUX user subroutine.

All fluxes are positive when adding heat to the element. In addition, point fluxes can be applied at the nodes.

Films

Same specification as Fluxes .

Joule Heating

Capability is not available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Current

Same specifications as Fluxes .

Charge

Same specifications as Fluxes .

Output Points

Center or six Gaussian integration points (see Figure 3-158).

Tying

Use the UFORMSN user subroutine.

Coupled Thermal-Stress Analysis

Capability is available.

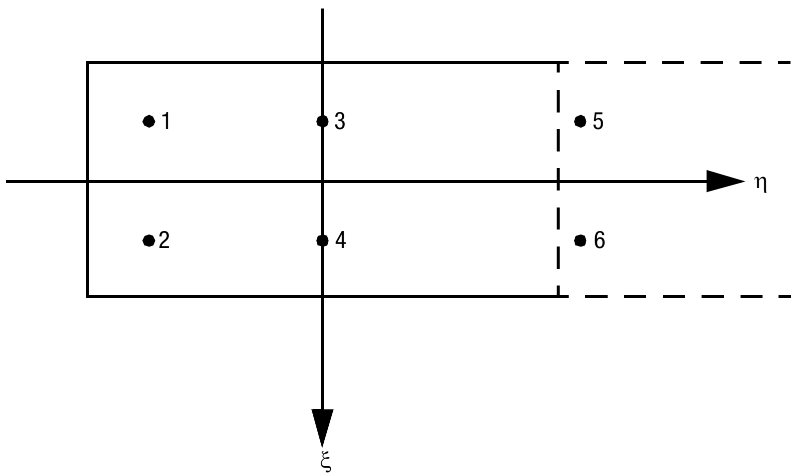


Figure 3-158 Integration Point Locations for Element 101

Element 102

Six-node Axisymmetric Semi-infinite Heat Transfer Element

This is a six-node axisymmetric semi-infinite heat transfer element that can be used to model an unbounded domain in one direction. This element is used in conjunction with the usual linear element. The interpolation functions are linear in the 1-2 direction, and cubic in the 2-5-3 direction. Mappings are such that the element expands to infinity. Both the conductivity and the capacity matrices are numerically integrated using six (2 x 3) integration points. This element cannot be used with the CONTACT option. There is no specific heat matrix associated with this element.

In addition, this element can be used for an electrostatic or a magnetostatic problem. A description of these two options can be found in Marc Volume A: Theory and User Information .

Quick Reference

Type 102

Six-node axisymmetric semi-infinite heat transfer element.

Connectivity

Six nodes per element (see Figure 3-159).

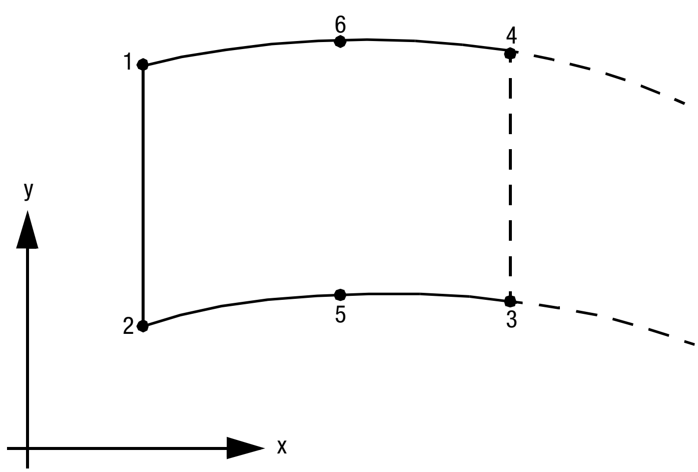


Figure 3-159 Six-node Axisymmetric Semi-infinite Heat Transfer Element

Counterclockwise numbering. The 1-2 face should be connected to a standard four-node axisymmetric heat transfer element and the 2-3 and 4-1 faces should be either connected to another six-node axisymmetric semi-infinite heat transfer element or be a free surface. The 3-4 face should not be connected to any other elements.

Coordinates

Two coordinates in the global z and r directions.

Geometry

Not required.

Degrees of Freedom

1 = temperature (heat transfer)
1 = potential (electrostatic)
1 = potential (magnetostatic)

Fluxes

Distributed fluxes are listed below:

Flux Type	Description
0	Uniform flux per unit area on 1-2 face of the element.
1	Nonuniform flux per unit area on 1-2 face of the element; magnitude given in the FLUX user subroutine.
2	Uniform flux per unit volume on whole element.
3	Nonuniform flux per unit volume on whole element; magnitude given in the FLUX user subroutine.
8	Uniform flux per unit area on 2-5-3 face of the element.
9	Nonuniform flux per unit area on 2-5-3 face of the element; magnitude given in the FLUX user subroutine.
12	Uniform flux per unit area on 4-6-1 face of the element.
13	Nonuniform flux per unit area on 4-6-1 face of the element; magnitude given in the FLUX user subroutine.

All fluxes are positive when adding heat to the element. In addition, point fluxes can be applied at the nodes.

Films

Same specification as Fluxes .

Joule Heating

Capability is not available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Charge

Same specifications as Fluxes .

Current

Same specifications as Fluxes .

Output Points

Center or six Gaussian integration points (see Figure 3-160).

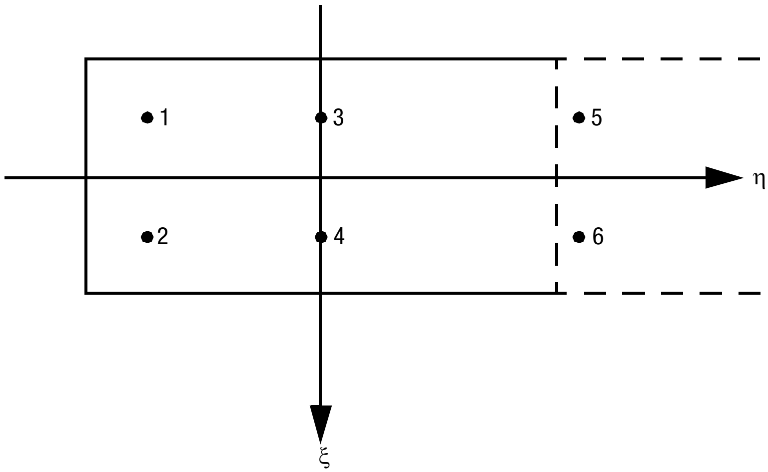


Figure 3-160 Integration Point Locations for Element 102

Tying

Use the UFORMSN user subroutine.

Coupled Thermal-Stress Analysis

Capability is available.

Element 103

Nine-node Planar Semi-infinite Heat Transfer Element

This is a nine-node planar semi-infinite heat transfer element that can be used to model an unbounded domain in one direction. This element is used in conjunction with the usual quadratic element. The interpolation functions are parabolic in the 1-5-2 direction, and cubic in the 2-6-3 direction. Mappings are such that the element expands to infinity. Both the conductivity and the capacity matrices are numerically integrated using nine (3 x 3) integration points. This element cannot be used with the CONTACT option. There is no specific heat matrix associated with this element.

In addition, this element can be used for an electrostatic or a magnetostatic problem. A description of these two options can be found in Marc Volume A: Theory and User Information .

Quick Reference

Type 103

Nine-node planar semi-infinite heat transfer element.

Connectivity

Nine nodes per element (see Figure 3-161).

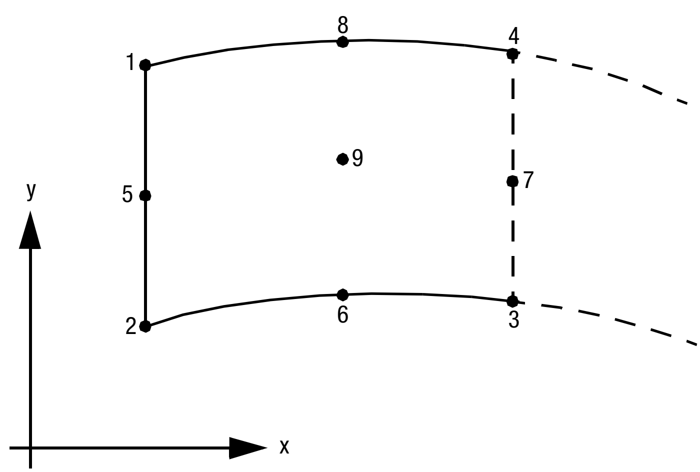


Figure 3-161 Nine-node Planar Semi-infinite Heat Transfer Element

Counterclockwise numbering. The 1-5-2 face should be connected to a standard eight-node planar heat transfer element and the 2-6-3 and 4-8-1 faces should be either connected to another nine-node plane semi-infinite heat transfer element or be a free surface. The 3-7-4 face should not be connected to any other elements.

Coordinates

Two coordinates in the global x and y directions.

Geometry

Thickness of the element is given in the first field EGEOM1.

Degrees of Freedom

1 = temperature (heat transfer)
1 = potential (electrostatic)
1 = potential (magnetostatic)

Fluxes

Distributed fluxes are listed below:

Flux Type	Description
0	Uniform flux per unit area on 1-5-2 face of the element.
1	Nonuniform flux per unit area on 1-5-2 face of the element; magnitude given in the FLUX user subroutine.
2	Uniform flux per unit volume on whole element.
3	Nonuniform flux per unit volume on whole element; magnitude given in the FLUX user subroutine.
8	Uniform flux per unit area on 2-6-3 face of the element.
9	Nonuniform flux per unit area on 2-6-3 face of the element; magnitude given in the FLUX user subroutine.
12	Uniform flux per unit area on 4-8-1 face of the element.
13	Nonuniform flux per unit area on 4-8-1 face of the element; magnitude given in the FLUX user subroutine.

All fluxes are positive when adding heat to the element. In addition, point fluxes can be applied at the nodes.

Films

Same specification as Fluxes .

Charge

Same specifications as Fluxes .

Current

Same specifications as Fluxes .

Joule Heating

Capability is not available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Output Points

Center or nine Gaussian integration points (see Figure 3-162).

Tying

Use the UFORMSN user subroutine.

Coupled Thermal-Stress Analysis

Capability is available.

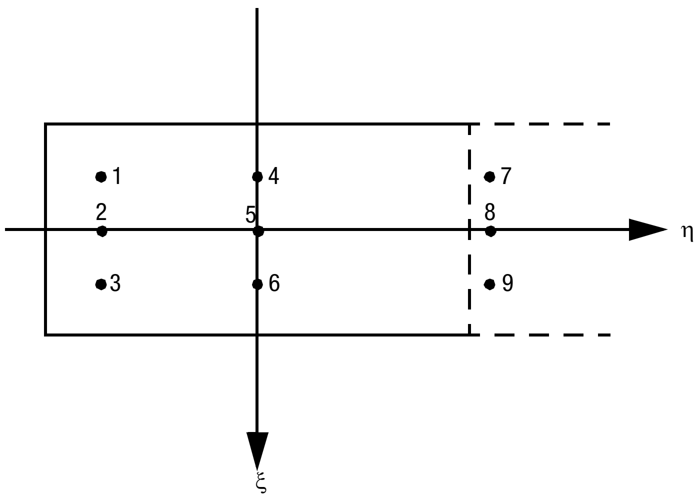


Figure 3-162 Integration Point Locations for Element 103

Element 104 Nine-node Axisymmetric Semi-infinite Heat Transfer Element

This is a nine-node axisymmetric semi-infinite heat transfer element that can be used to model an unbounded domain in one direction. This element is used in conjunction with the usual quadratic element. The interpolation functions are parabolic in the 1-5-2 direction, and cubic in the 2-6-3 direction. Mappings are such that the element expands to infinity. Both the conductivity and the capacity matrices are numerically integrated using nine (3 x 3) integration points. This element cannot be used with the CONTACT option. There is no specific heat matrix associated with this element.

In addition, this element can be used for an electrostatic or a magnetostatic problem. A description of these two options can be found in Marc Volume A: Theory and User Information .

Quick Reference

Type 104

Nine-node axisymmetric semi-infinite heat transfer element.

Connectivity

Nine nodes per element (see Figure 3-161).

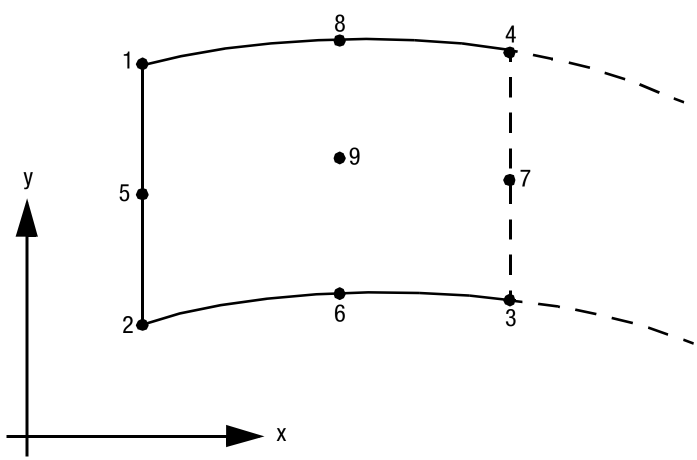


Figure 3-161 Nine-node Axisymmetric Semi-infinite Heat Transfer Element

Counterclockwise numbering. The 1-5-2 face should be connected to a standard eight-node axisymmetric heat transfer element and the 2-6-3 and 4-8-1 faces should be either connected to another nine-node axisymmetric semi-infinite heat transfer element or be a free surface. The 3-7-4 face should not be connected to any other elements.

Coordinates

Two coordinates in the global z- and r-directions.

Geometry

Not required.

Degrees of Freedom

1 = temperature (heat transfer)
1 = potential (electrostatic)
1 = potential (magnetostatic)

Fluxes

Distributed fluxes are listed below:

Flux Type	Description
0	Uniform flux per unit area on 1-5-2 face of the element.
1	Nonuniform flux per unit area on 1-5-2 face of the element; magnitude is given in the FLUX user subroutine.
2	Uniform flux per unit volume on whole element.
3	Nonuniform flux per unit volume on whole element; magnitude is given in the FLUX user subroutine.
8	Uniform flux per unit area on 2-6-3 face of the element.
9	Nonuniform flux per unit area on 2-6-3 face of the element; magnitude is given in the FLUX user subroutine.
12	Uniform flux per unit area on 4-8-1 face of the element.
13	Nonuniform flux per unit area on 4-8-1 face of the element; magnitude is given in the FLUX user subroutine.

All fluxes are positive when adding heat to the element. In addition, point fluxes can be applied at the nodes.

Films

Same specification as Fluxes .

Joule Heating

Capability is not available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Charge

Same specifications as Fluxes .

Current

Same specifications as Fluxes.

Output Points

Center or nine Gaussian integration points (see Figure 3-162).

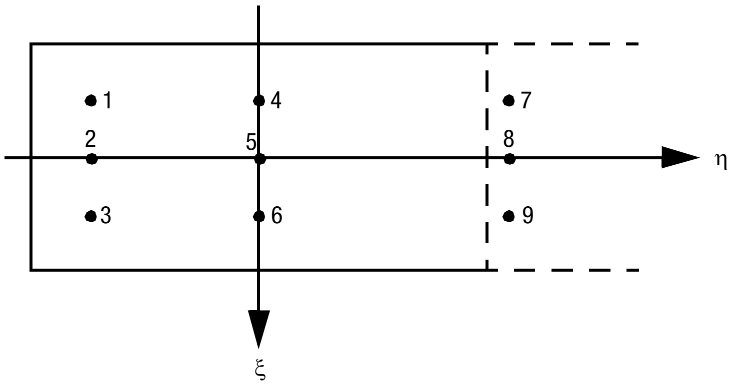


Figure 3-162 Integration Point Locations for Element 104

Tying

Use the UFORMSN user subroutine.

Coupled Thermal-Stress Analysis

Capability is available.

Element 105

Twelve-node 3-D Semi-infinite Heat Transfer Element

This is a 12-node 3-D semi-infinite heat transfer element that can be used to model an unbounded domain in one direction. This element is used in conjunction with the usual linear element. The interpolation functions are linear in the 1-2 direction, and cubic in the 2-9-3 direction. Mappings are such that the element expands to infinity. Both the conductivity and the capacity matrices are numerically integrated using 12 (2 x 3 x 2) integration points. This element cannot be used with the CONTACT option. There is no specific heat matrix associated with this element.

In addition, this element can be used for an electrostatic problem. A description of this option can be found in Marc Volume A: Theory and User Information.

Quick Reference

Type 105

Twelve-node 3-D semi-infinite heat transfer element.

Connectivity

Twelve nodes per element.

See Figure 3-163 for numbering. The 1-2-6-5 face should be connected to a standard eight-node 3-D heat transfer element and the 2-3-7-6, 5-6-7-8, 1-4-8-5 and 1-2-3-4 faces should be either connected to another 12-node 3-D semi-infinite heat transfer element or be a free surface. The 4-3-7-8 face should not be connected to any other elements.

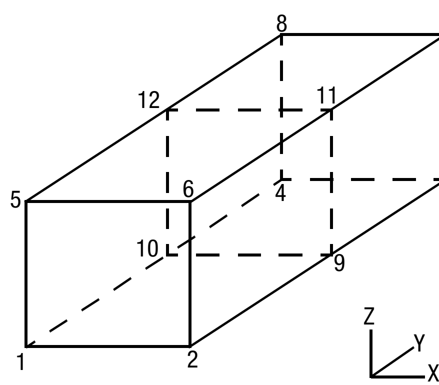


Figure 3-163 Twelve-node 3-D Semi-infinite Heat Transfer Element

Coordinates

Three coordinates in the global x-, y-, and z-directions.

Geometry

Not required.

Degrees of Freedom

1 = temperature (heat transfer)
1 = potential (electrostatic)

Fluxes

Distributed fluxes are listed below:

Flux Type	Description
0	Uniform flux per unit area on 1-2-3-4 face of the element.
1	Nonuniform flux per unit area on 1-2-3-4 face of the element; magnitude is given in the FLUX user subroutine.
2	Uniform flux per unit volume on whole element.
3	Nonuniform flux per unit volume on whole element; magnitude is given in the FLUX. user subroutine
4	Uniform flux per unit area on 5-6-7-8 face of the element.
5	Nonuniform flux per unit area on 5-6-7-8 face of the element; magnitude is given in the FLUX user subroutine.
6	Uniform flux per unit area on 1-2-6-5 face of the element.
7	Nonuniform flux per unit area on 1-2-6-5 face of the element; magnitude is given in the FLUX user subroutine.
8	Uniform flux per unit area on 2-3-7-6 face of the element.
9	Nonuniform flux per unit area on 2-3-7-6 face of the element; magnitude is given in the FLUX user subroutine.
12	Uniform flux per unit area on 1-4-8-5 face of the element.
13	Nonuniform flux per unit area on 1-4-8-5 face of the element; magnitude is given in the FLUX user subroutine.

All fluxes are positive when adding heat to the element. In addition, point fluxes can be applied at the nodes.

Films

Same specification as Fluxes .

Joule Heating

Capability is not available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Charge

Same specifications as Fluxes .

Output Points

Center or 12 Gaussian integration points (see Figure 3-164).

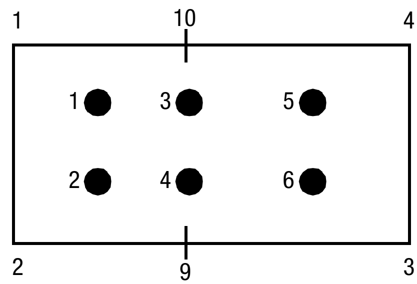


Figure 3-164 Integration Point Locations for Element 105

Tying

Use the UFORMSN user subroutine.

Coupled Thermal-Stress Analysis

Capability is available.

Element 106 Twenty-seven-node 3-D Semi-infinite Heat Transfer Element

This is a 27-node 3-D semi-infinite heat transfer element that can be used to model an unbounded domain in one direction. This element is used in conjunction with the usual quadratic element. The interpolation functions are linear in the 1-2 direction, and cubic in the 2-9-3 direction. Mappings are such that the element expands to infinity. Both the conductivity and the capacity matrices are numerically integrated using 27 (3 x 3 x 3) integration points. This element cannot be used with the CONTACT option. There is no specific heat matrix associated with this element.

In addition, this element can be used for an electrostatic problem. A description of this option can be found in Marc Volume A: Theory and User Information .

Quick Reference

Type 106

Twenty-seven-node 3-D semi-infinite heat transfer element.

Connectivity

Twenty-seven nodes per element.

See Figure 3-165 for numbering. The 1-2-6-5 face should be connected to a standard eight-node 3-D heat transfer element and the 2-3-7-6, 5-6-7-8, 1-4-8-5, and 1-2-3-4 faces should be either connected to another 27-node 3-D semi-infinite heat transfer element or be a free surface. The 4-3-7-8 face should not be connected to any other elements.

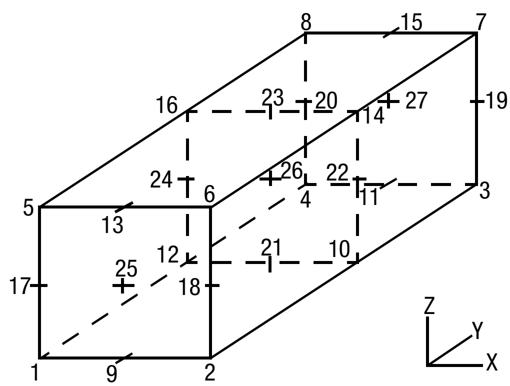


Figure 3-165 Twenty-seven-node 3-D Semi-infinite Heat Transfer Element

Coordinates

Three coordinates in the global x-, y-, and z-directions.

Geometry

Not required.

Degrees of Freedom

1 = temperature (heat transfer)
1 = potential (electrostatic)

Fluxes

Distributed fluxes are listed below:

Flux Type	Description
0	Uniform flux per unit area on 1-2-3-4 face of the element.
1	Nonuniform flux per unit area on 1-2-3-4 face of the element; magnitude is given in the FLUX user subroutine.
2	Uniform flux per unit volume on whole element.
3	Nonuniform flux per unit volume on whole element; magnitude is given in the FLUX user subroutine.
4	Uniform flux per unit area on 5-6-7-8 face of the element.
5	Nonuniform flux per unit area on 5-6-7-8 face of the element; magnitude is given in the FLUX user subroutine.
6	Uniform flux per unit area on 1-2-6-5 face of the element.
7	Nonuniform flux per unit area on 1-2-6-5 face of the element; magnitude is given in the FLUX user subroutine.
8	Uniform flux per unit area on 2-3-7-6 face of the element.
9	Nonuniform flux per unit area on 2-3-7-6 face of the element; magnitude is given in the FLUX user subroutine.
12	Uniform flux per unit area on 1-4-8-5 face of the element.
13	Nonuniform flux per unit area on 1-4-8-5 face of the element; magnitude is given in the FLUX user subroutine.

All fluxes are positive when adding heat to the element. In addition, point fluxes can be applied at the nodes.

Films

Same specification as Fluxes .

Joule Heating

Capability is not available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Charge

Same specifications as Fluxes .

Output Points

Center or 27 Gaussian integration points (see Figure 3-166).

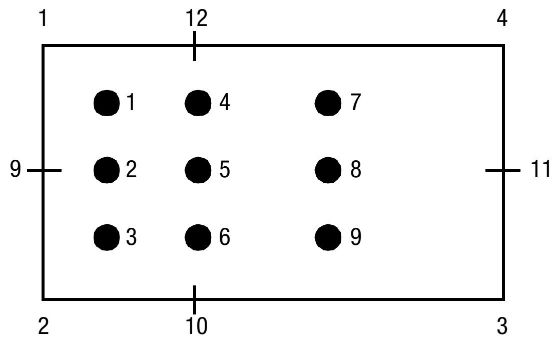


Figure 3-166 Integration Point Locations for Element 106

Tying

Use the UFORMSN user subroutine.

Coupled Thermal-Stress Analysis

Capability is available.

Element 107

Twelve-node 3-D Semi-infinite Stress Element

This is a 12-node 3-D semi-infinite stress element that can be used to model an unbounded domain in one direction. This element is used in conjunction with the usual linear element. The interpolation functions are linear in the 1-2 direction, and cubic in the 2-9-3 direction. Mappings are such that the element expands to infinity. The stiffness matrix is numerically integrated using 12 (2 x 3 x 2) integration points. This element only has linear capability. This element cannot be used with the CONTACT option. There is no mass matrix associated with this element.

Quick Reference

Type 107

Twelve-node 3-D semi-infinite stress element.

Connectivity

Twelve nodes per element.

See Figure 3-167 for numbering. The 1-2-6-5 face should be connected to a standard eight-node 3-D stress element and the 2-3-7-6, 5-6-7-8, 1-4-8-5, and 1-2-3-4 faces should be either connected to another 12-node 3-D semi-infinite stress element or be a free surface. The 4-3-7-8 face should not be connected to any other elements.

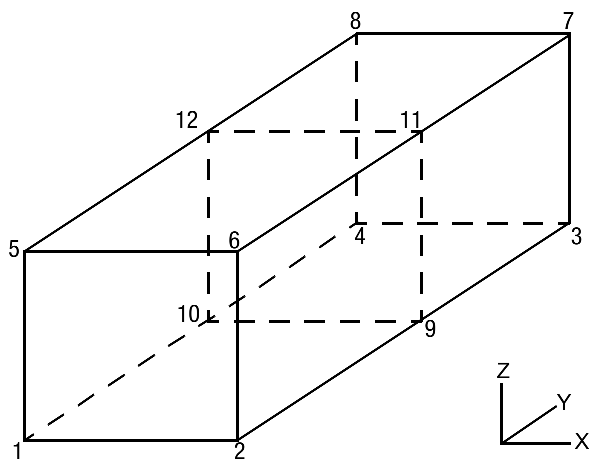


Figure 3-167 Twelve-node 3-D Semi-infinite Stress Element

Coordinates

Three coordinates in the global x-, y-, and z-directions.

Geometry

Not required.

Degrees of Freedom

1 = u displacement
2 = v displacement
3 = w displacement

Distributed Loads

Distributed loads are listed below:

Load Type	Description
0	Uniform load per unit area on 1-2-3-4 face of the element.
1	Nonuniform load per unit area on 1-2-3-4 face of the element; magnitude is given in the FORCEM user subroutine.
2	Uniform load per unit volume on whole element.
3	Nonuniform load per unit volume on whole element; magnitude and direction is given in the FORCEM user subroutine.
4	Uniform load per unit area on 5-6-7-8 face of the element.
5	Nonuniform load per unit area on 5-6-7-8 face of the element; magnitude is given in the FORCEM user subroutine.
6	Uniform load per unit area on 1-2-6-5 face of the element.
7	Nonuniform load per unit area on 1-2-6-5 face of the element; magnitude is given in the FORCEM user subroutine.
8	Uniform load per unit area on 2-3-7-6 face of the element.
9	Nonuniform load per unit area on 2-3-7-6 face of the element; magnitude is given in the FORCEM user subroutine.
12	Uniform load per unit area on 1-4-8-5 face of the element.
13	Nonuniform load per unit area on 1-4-8-5 face of the element; magnitude is given in the FORCEM user subroutine.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.

Point loads can be applied at nodes.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output Points

Center or 12 Gaussian integration points (see Figure 3-168).

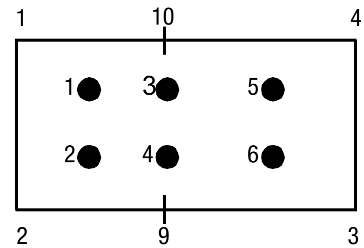


Figure 3-168 Integration Point Locations for Element 107

Output of Strains

$1 = \varepsilon_{xx}$

$2 = \varepsilon_{yy}$

$3 = \varepsilon_{zz}$

$4 = \gamma_{xy}$

$5 = \gamma_{yz}$

$6 = \gamma_{xz}$

Output of Stresses

Same as for Output of Strains .

Tying

Use the UFORMSN user subroutine.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 105. See Element 105 for a description of the conventions used for entering the flux and film data for this element.

Note:	No boundary conditions at infinity are required. Locations of nodes 9, 10, 11, and 12 express the decay of functions. See the description of element type 91 for more information.

Element 108

Twenty-seven-node 3-D Semi-infinite Stress Element

This is a 27-node 3-D semi-infinite stress element that can be used to model an unbounded domain in one direction. This element is used in conjunction with the usual quadratic element. The interpolation functions are linear in the 1-2 direction, and cubic in the 2-9-3 direction. Mappings are such that the element expands to infinity. The stiffness matrix is numerically integrated using 27 (3 x 3 x 3) integration points. This element only has linear capability. This element cannot be used with the CONTACT option. There is no mass matrix associated with this element.

Quick Reference

Type 108

Twenty-seven-node 3-D semi-infinite stress element.

Connectivity

Twenty-seven nodes per element.

See Figure 3-169 for numbering. The 1-2-6-5 face should be connected to a standard 20-node 3-D stress element and the 2-3-7-6,6-7-8, 1-4-8-5 and 1-2-3-4 faces should be either connected to another 27-node 3-D semi-infinite stress element or be a free surface. The 4-3-7-8 face should not be connected to any other elements.

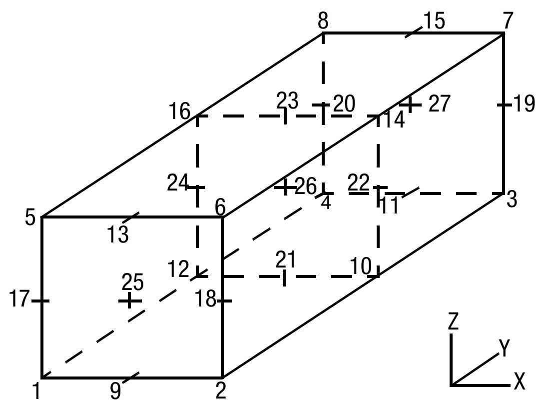


Figure 3-169 Twenty-seven-node 3-D Semi-infinite Stress Element

Geometry

Not required.

Degrees of Freedom

1 = u displacement
2 = v displacement
3 = w displacement

Distributed Loads

Distributed loads are listed below:

Load Type	Description
0	Uniform load per unit area on 1-2-3-4 face of the element.
1	Nonuniform load per unit area on 1-2-3-4 face of the element; magnitude is given in the FORCEM user subroutine.
2	Uniform load per unit volume on whole element.
3	Nonuniform load per unit volume on whole element; magnitude and direction given in the FORCEM user subroutine.
4	Uniform load per unit area on 5-6-7-8 face of the element.
5	Nonuniform load per unit area on 5-6-7-8 face of the element; magnitude is given in the FORCEM user subroutine.
6	Uniform load per unit area on 1-2-6-5 face of the element.
7	Nonuniform load per unit area on 1-2-6-5 face of the element; magnitude is given in the FORCEM user subroutine.
8	Uniform load per unit area on 2-3-7-6 face of the element.
9	Nonuniform load per unit area on 2-3-7-6 face of the element; magnitude is given in the FORCEM user subroutine.
12	Uniform load per unit area on 1-4-8-5 face of the element.
13	Nonuniform load per unit area on 1-4-8-5 face of the element; magnitude is given in the FORCEM user subroutine.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.

Point loads can be applied at nodes.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output Points

Center or 27 Gaussian integration points (see Figure 3-170).

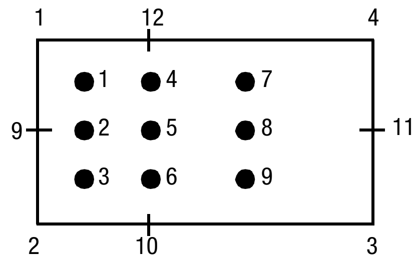


Figure 3-170 Integration Point Locations for Element 108

Output of Strains

$1 = \varepsilon_{xx}$

$2 = \varepsilon_{yy}$

$3 = \varepsilon_{zz}$

$4 = \gamma_{xy}$

$5 = \gamma_{yz}$

$6 = \gamma_{xz}$

Output of Stresses

Same as for Output of Strains .

Tying

Use the UFORMSN user subroutine.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 106. See Element 106 for a description of the conventions used for entering the flux and film data for this element.

Note:	No boundary conditions at infinity are required. Locations of nodes 10, 12, 14, 16, 21, 22, 23, 24, and 26 express the decay of functions.

Element 109

Eight-node 3-D Magnetostatic Element

This is an eight-node 3-D magnetostatic element with linear interpolation functions. It is similar to element type 43. The coefficient matrix is numerically integrated using eight (2 x 2 x 2) integration points.

Quick Reference

Type 109

Eight-node 3-D magnetostatic element.

Connectivity

Eight nodes per element.

See Figure 3-171 for numbering. Nodes 1-4 are the corners on one face, numbered in a counterclockwise direction when viewed from inside the element. Nodes 5-8 are the nodes on the other face, with node 5 opposite node 1, and so on.

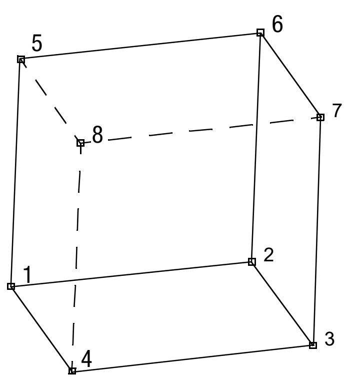


Figure 3-171 Eight-node 3-D Magnetostatic Element

Coordinates

Three coordinates in the global x-, y-, and z-directions.

Geometry

The value of the penalty factor is given in the second field EGEOM7 (default is 0.0001).

Degrees of Freedom

1 = x component of vector potential
2 = y component of vector potential
3 = z component of vector potential

Distributed Currents

Distributed currents are listed in the table below:

Current Type	Description
0	Uniform current on 1-2-3-4 face.
1	Nonuniform current on 1-2-3-4 face; magnitude is supplied through the FORCEM user subroutine.
2	Uniform body force per unit volume in -z-direction.
3	Nonuniform body force per unit volume (e.g. centrifugal force); magnitude and direction is supplied through the FORCEM user subroutine.
4	Uniform current on 6-5-8-7 face.
5	Nonuniform current on 6-5-8-7 face.
6	Uniform current on 2-1-5-6 face.
7	Nonuniform current on 2-1-5-6 face.
8	Uniform current on 3-2-6-7 face.
9	Nonuniform current on 3-2-6-7 face.
10	Uniform pressure on 4-3-7-8 face.
11	Nonuniform current on 4-3-7-8 face.
12	Uniform current on 1-4-8-5 face.
13	Nonuniform current on 1-4-8-5 face.
20	Uniform current on 1-2-3-4 face.
21	Nonuniform load on 1-2-3-4 face.
22	Uniform body force per unit volume in -z-direction.
23	Nonuniform body force per unit volume (for example, centrifugal force); magnitude and direction is supplied through the FORCEM user subroutine.
24	Uniform current on 6-5-8-7 face.
25	Nonuniform load on 6-5-8-7 face.
26	Uniform current on 2-1-5-6 face.
27	Nonuniform load on 2-1-5-6 face.
28	Uniform current on 3-2-6-7 face.
29	Nonuniform load on 3-2-6-7 face.
30	Uniform pressure on 4-3-7-8 face.
31	Nonuniform load on 4-3-7-8 face.
32	Uniform current on 1-4-8-5 face.
33	Nonuniform load on 1-4-8-5 face.
40	Uniform shear 1-2-3-4 face in 1⇒2 direction.
41	Nonuniform shear 1-2-3-4 face in 1⇒2 direction.
42	Uniform shear 1-2-3-4 face in 1⇒2 direction.

Current Type	Description
43	Nonuniform shear 1-2-3-4 face in 2⇒3 direction.
48	Uniform shear 6-5-8-7 face in 5⇒6 direction.
49	Nonuniform shear 6-5-8-7 face in 5⇒6 direction.
50	Uniform shear 6-5-8-7 face in 6⇒7 direction.
51	Nonuniform shear 6-5-8-7 face in 6⇒7 direction.
52	Uniform shear 2-1-5-6 face in 1⇒2 direction.
53	Nonuniform shear 2-1-5-6 face in 1⇒2 direction.
54	Uniform shear 2-1-5-6 face in 1⇒5 direction.
55	Nonuniform shear 2-1-5-6 face in 1⇒5 direction.
56	Uniform shear 3-2-6-7 face in 2⇒3 direction.
57	Nonuniform shear 3-2-6-7 face in 2⇒3 direction.
58	Uniform shear 3-2-6-7 face in 2⇒6 direction.
59	Nonuniform shear 3-2-6-7 face in 2⇒6 direction.
60	Uniform shear 4-3-7-8 face in 3⇒4 direction.
61	Nonuniform shear 4-3-7-8 face in 3⇒4 direction.
62	Uniform shear 4-3-7-8 face in 3⇒7 direction.
63	Nonuniform shear 4-3-7-8 face in 3⇒7 direction.
64	Uniform shear 1-4-8-5 face in 4⇒1 direction.
65	Nonuniform shear 1-4-8-5 face in 4⇒1 direction.
66	Uniform shear 1-4-8-5 face in 1⇒5 direction.
67	Nonuniform shear 1-4-8-5 face in 1⇒5 direction.

For all nonuniform currents, body forces per unit volume and loads, the magnitude and direction is supplied via the FORCEM user subroutine.

Currents are positive into element face.

Joule Heating

Capability is not available.

Electrostatic

Capability is not available.

Magnetostatic

Capability is available.

Output Points

Centroid or the eight integration points shown in Figure 3-172.

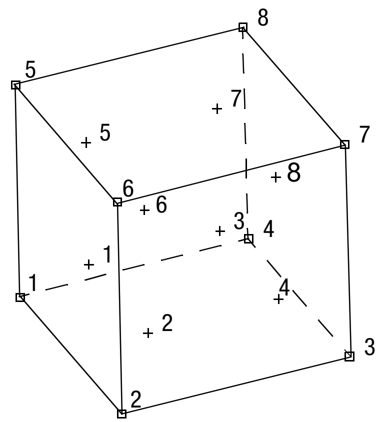


Figure 3-172 Eight-Point Gauss Integration Scheme for Element 109

Output at Integration Points

Potential

Magnetic flux density

1 = x
2 = y
3 = z

Magnetic field vector

1 = x
2 = y
3 = z

Tying

Use the UFORMSN user subroutine.

Element 110 Twelve-node 3-D Semi-infinite Magnetostatic Element

This is a 12-node 3-D semi-infinite magnetostatic element that can be used to model an unbounded domain in one direction. This element is used in conjunction with the usual linear element. The interpolation functions are linear in the 1-2 direction, and cubic in the 2-7-3 direction. Mappings are such that the element expands to infinity. The coefficient matrix is numerically integrated using 12 (2 x 3 x 2) integration points.

Quick Reference

Type 110

Twelve-node 3-D semi-infinite magnetostatic element.

Connectivity

Twelve nodes per element. See Figure 3-173 for numbering. The 1-2-6-5 face should be connected to a standard 8-node 3-D stress element and the 2-3-7-6, 5-6-7-8, 1-4-8-5 and 1-2-3-4 faces should be either connected to another 12-node 3-D semi-infinite stress element or be a free surface. The 4-3-7-8 face should not be connected to any other elements.

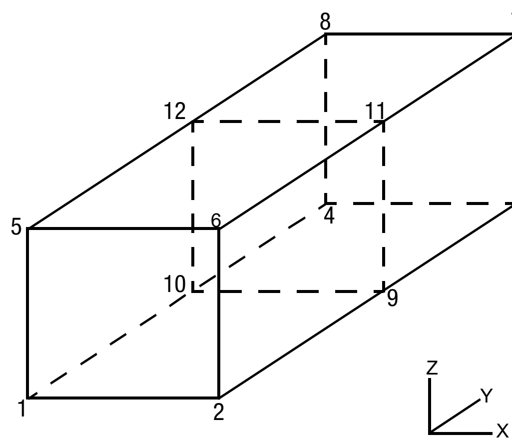


Figure 3-173 Twelve-node 3-D Semi-infinite Magnetostatic Element

Coordinates

Three coordinates in the global x-, y-, and z-directions.

Geometry

The value of the penalty factor is given in the second field `EGEOM7` (default is 0.0001).

Degrees of Freedom

1 = x component of vector potential
2 = y component of vector potential
3 = z component of vector potential

Distributed Currents

Distributed currents are listed in the table below:

Current Type	Description
0	Uniform current per unit area on 1-2-3-4 face of the element.
1	Nonuniform current per unit area on 1-2-3-4 face of the element; magnitude is given in the FORCEM. user subroutine
2	Uniform current per unit volume on whole element.
3	Nonuniform current per unit volume on whole element; magnitude is given in the FORCEM user subroutine.
4	Uniform current per unit area on 5-6-7-8 face of the element.
5	Nonuniform current per unit area on 5-6-7-8 face of the element; magnitude is given in the FORCEM user subroutine.
6	Uniform current per unit area on 1-2-6-5 face of the element.
7	Nonuniform current per unit area on 1-2-6-5 face of the element; magnitude is given in the FORCEM user subroutine.
8	Uniform current per unit area on 2-3-7-6 face of the element.
9	Nonuniform current per unit area on 2-3-7-6 face of the element; magnitude is given in the FORCEM user subroutine.
12	Uniform current per unit area on 1-4-8-5 face of the element.
13	Nonuniform current per unit area on 1-4-8-5 face of the element; magnitude is given in the FORCEM user subroutine.

Point currents can be applied at nodes.

Output Points

Center or 12 Gaussian integration points (see Figure 3-174).

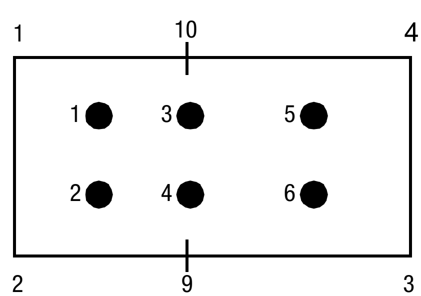


Figure 3-174 1-2 Integration Point Locations for Element 110

Joule Heating

Capability is not available.

Electrostatic

Capability is not available.

Magnetostatic

Capability is available.

Output at Integration Points

Potential

Magnetic flux density

1 = x
2 = y
3 = z

Magnetic field vector

1 = x
2 = y
3 = z

Tying

Use the UFORMSN user subroutine.

Element 111

Arbitrary Quadrilateral Planar Magnetodynamic

Element type 111 is a four-node arbitrary quadrilateral written for planar magnetodynamic applications. This element can be used for either transient or harmonic problems.

Quick Reference

Type 111

Planar quadrilateral.

Connectivity

Four nodes per element.

Node numbering follows right-handed convention (counterclockwise).

Geometry

Not applicable, the thickness is always equal to one.

Coordinates

Two coordinates in the global x- and y-directions.

Degrees Of Freedom

Global displacement degrees of freedom.

Magnetic Potential

1 = A_x
2 = A_y
3 = A_z

Electric Potential

4 = V

Distributed Current

Current types for distributed currents as follows:

0 - 11 Currents normal to element edge.
20 - 27 Currents in plane, tangential to element edge.
30 - 41 Currents out of plane.

Current Type	Description
0	Uniform normal current everywhere distributed on 1-2 face of the element.
1	Uniform body current in the x-direction.

Current Type	Description
2	Uniform body current in the y-direction.
3	Nonuniform normal current everywhere on 1-2 face of the element; magnitude is supplied through the FORCEM user subroutine.
4	Nonuniform body current in the x-direction.
5	Nonuniform body current in the y-direction
6	Uniform current on 2-3 face of the element.
7	Nonuniform current on 2-3 face of the element; magnitude is supplied through the FORCEM user subroutine.
8	Uniform current on 3-4 face of the element.
9	Nonuniform current on 3-4 face of the element; magnitude is supplied through the FORCEM user subroutine.
10	Uniform current on 4-1 face of the element.
11	Nonuniform current on 4-1 face of the element; magnitude is supplied through the FORCEM user subroutine.
20	Uniform shear current on side 1 - 2 (positive from 1 to 2).
21	Nonuniform shear current on side 1 - 2; magnitude is supplied through the FORCEM user subroutine.
22	Uniform shear current on side 2 - 3 (positive from 2 to 3).
23	Nonuniform shear current on side 2 - 3; magnitude is supplied through the FORCEM user subroutine.
24	Uniform shear current on side 3 - 4 (positive from 3 to 4).
25	Nonuniform shear current on side 3 - 4; magnitude is supplied through the FORCEM user subroutine.
26	Uniform shear current on side 4 - 1 (positive from 4 to 1).
27	Nonuniform shear current on side 4 - 1, magnitude is supplied through the FORCEM user subroutine.
30	Uniform current per unit area of 1-2 face of the element, normal to the plane.
31	Nonuniform current per unit area on 1-2 face of the element, normal to the plane; magnitude is given in the FORCEM user subroutine.
36	Uniform current per unit area on 2-3 face of the element, normal to the plane.
37	Nonuniform current per unit area on 2-3 face of the element, normal to the plane; magnitude is given in the FORCEM user subroutine.
38	Uniform current per unit area on 3-4 face of the element normal to the plane.
39	Nonuniform current per unit area on 3-4 face of the element, normal to the plane; magnitude is given in the FORCEM user subroutine.
40	Uniform current per unit area on 4-1 face of the element, normal to the plane.
41	Nonuniform current per unit area on 4-1 face of the element, normal to the plane; magnitude is given in the FORCEM user subroutine.

All currents are positive when directed into the element. In addition, point currents and charges can be applied at the nodes.

Distributed Charges

Charge types for distributed charges are as follows:

Charge Type	Description
50	Uniform charge per unit area 1-2 face of the element.
51	Uniform charge per unit volume on whole element.
52	Uniform charge per unit volume on whole element.
53	Nonuniform charge per unit area on 1-2 face of the element.
54	Nonuniform charge per unit volume on whole element.
55	Nonuniform charge per unit volume on whole element.
56	Uniform charge per unit area on 2-3 face of the element.
57	Nonuniform charge per unit area on 2-3 face of the element.
58	Uniform charge per unit area on 3-4 face of the element.
59	Nonuniform charge per unit area on 3-4 face of the element.
60	Uniform charge per unit area on 4-1 face of the element.
61	Nonuniform charge per unit area on 4-1 face of the element.

For all nonuniform charges, the magnitude is supplied through the FORCEM user subroutine.

All charges are positive when adding charge to the element. In addition, point charges can be applied at the nodes.

Output

Three components of:

Electric field intensity	E
Electric charge density	D
Magnetic field intensity	H
Magnetic charge density	B
Current density	J

Transformation

Two global degrees of freedom (A_x, A_y) can be transformed into local coordinates.

Output Points

Output is available at the four Gaussian points shown in Figure 3-175.

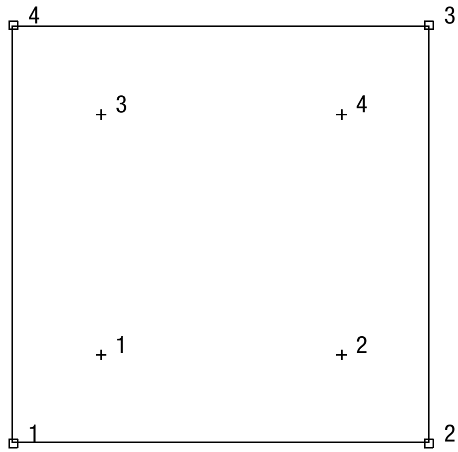


Figure 3-175 Gaussian Integration Points for Element Type 111

Element 112 Arbitrary Quadrilateral Axisymmetric Magnetodynamic Ring

Element type 112 is a four-node, isoparametric, arbitrary quadrilateral written for axisymmetric magnetodynamic applications. This element can be used for either transient or harmonic analyses.

Quick Reference

Type 112

Axisymmetric, arbitrary ring with a quadrilateral cross-section.

Connectivity

Four nodes per element. Node numbering follows right-handed convention (counterclockwise).

Geometry

Not applicable for this element.

Coordinates

Two coordinates in the global z and r directions.

Degrees of Freedom

Vector Potential

$1 = A_z$

$2 = A_r$

$3 = A_\theta$

Scalar Potential

$4 = V$

Distributed Currents

Current types for distributed currents are listed below:

Current Type	Description
0	Uniform normal current distributed on 1-2 face of the element.
1	Uniform body current in the z-direction.
2	Uniform body current in the r-direction.
3	Nonuniform normal current on 1-2 face of the element; magnitude is supplied through the FORCEM. user subroutine
4	Nonuniform body current in the z-direction.
5	Nonuniform body current in the r-direction.

Current Type	Description
6	Uniform normal current on 2-3 face of the element.
7	Nonuniform normal current on 2-3 face of the element; magnitude is supplied through the FORCEM user subroutine.
8	Uniform normal current on 3-4 face of the element.
9	Nonuniform normal current on 3-4 face of the element; magnitude is supplied through the FORCEM user subroutine.
10	Uniform normal current on 4-1 face of the element.
11	Nonuniform normal current on 4-1 face of the element; magnitude is supplied through the FORCEM user subroutine.
20	Uniform shear current on side 1 - 2 (positive from 1 to 2).
21	Nonuniform shear current on side 1 - 2; magnitude is supplied through the FORCEM user subroutine.
22	Uniform shear current on side 2 - 3 (positive from 2 to 3).
23	Nonuniform shear current on side 2 - 3; magnitude is supplied through the FORCEM user subroutine.
24	Uniform shear current on side 3 - 4 (positive from 3 to 4).
25	Nonuniform shear current on side 3 - 4; magnitude is supplied through the FORCEM user subroutine.
26	Uniform shear current on side 4 - 1 (positive from 4 to 1).
27	Nonuniform shear current on side 4 - 1; magnitude is supplied through the FORCEM user subroutine.
30	Uniform current per unit area 1-2 face of the element in the theta direction.
33	Nonuniform current per unit area on 1-2 face of the element in the theta direction; magnitude is given in the FORCEM user subroutine.
36	Uniform current per unit area on 2-3 face of the element in the theta direction.
37	Nonuniform current per unit area on 2-3 face of the element in the theta direction; magnitude is given in the FORCEM user subroutine.
38	Uniform current per unit area on 3-4 face of the element in the theta direction.
39	Nonuniform current per unit area on 3-4 face of the element in the theta direction; magnitude is given in the FORCEM user subroutine.
40	Uniform current per unit area on 4-1 face of the element in the theta direction.
41	Nonuniform current per unit area on 4-1 face of the element in the theta direction; magnitude is given in the FORCEM user subroutine.

All currents are positive when directed into the element. In addition, point currents can be applied at the nodes. The magnitude of point current must correspond to the current integrated around the circumference.

Distributed Charges

Charge types for distributed charges are as follows:

Charge Type	Description
50	Uniform charge per unit area 1-2 face of the element.
51	Uniform charge per unit volume on whole element.
52	Uniform charge per unit volume on whole element.
53	Nonuniform charge per unit area on 1-2 face of the element; magnitude is given in the FORCEM user subroutine.
54	Nonuniform charge per unit volume on whole element; magnitude is given in the FORCEM user subroutine.
55	Nonuniform charge per unit volume on whole element; magnitude is given in the FORCEM user subroutine.
56	Uniform charge per unit area on 2-3 face of the element.
57	Nonuniform charge per unit area on 2-3 face of the element; magnitude is given in the FORCEM user subroutine.
58	Uniform charge per unit area on 3-4 face of the element.
59	Nonuniform charge per unit area on 3-4 face of the element; magnitude is given in the FORCEM user subroutine.
60	Uniform charge per unit area on 4-1 face of the element.
61	Nonuniform charge per unit area on 4-1 face of the element; magnitude is given in the FORCEM user subroutine.

All charges are positive when adding charge to the element. In addition, point charges can be applied at the nodes. The magnitude of the point charge must correspond to the charge integrated around the circumference.

Output

Three components of:

Electric field intensity	E
Electric flux density	D
Magnetic field intensity	H
Magnetic flux density	B
Current density	J

Transformation

Two global degrees (A_z, A_r) of freedom can be transformed into local coordinates.

Output Points

Output is available at the four Gaussian points shown in Figure 3-176.

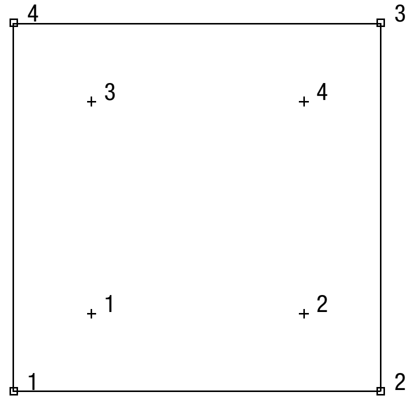


Figure 3-176 Integration Points for Element 112

Element 113 Three-dimensional Magnetodynamic Arbitrarily Distorted Brick

Element 113 is an eight-node isoparametric brick element and can be used for either transient or harmonic analysis. This element uses trilinear interpolation functions for the vector and scalar potential. The stiffness of this element is formed using eight-point Gaussian integration.

Note:	As in all three-dimensional analyses, a large nodal bandwidth results in long computing times. Optimize the nodal bandwidth.

Quick Reference

Type 113

Eight-node 3-D first-order isoparametric element (arbitrarily distorted brick).

Connectivity

Eight nodes per element (see Figure 3-177). Node numbering must follow the scheme below:

> Nodes 1, 2, 3, and 4 are corners of one face, given in counterclockwise order when viewed from inside the element. Node 5 has the same edge as node 1, node 6 as node 2, node 7 as node 3, and node 8 as node 4.

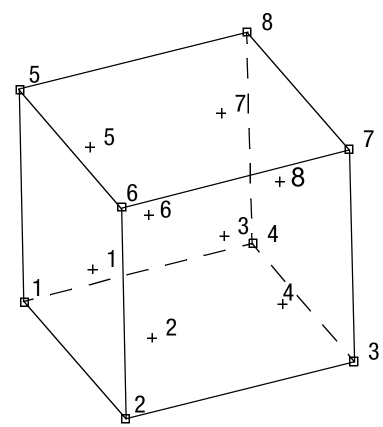


Figure 3-177 Nodes and Integration Point for Element 113

Geometry

Not applicable for this element.

Coordinates

Three coordinates in the global x-, y-, and z-directions.

Degrees of Freedom

Four global degrees of freedom A_x, A_y, A_z, V per node.

Distributed Currents

Distributed currents chosen by value of `IBODY` as follows:

Current Type	Description
0	Uniform normal current on 1-2-3-4 face.
1	Nonuniform normal current on 1-2-3-4 face.
2	Uniform volume current in the z-direction.
3	Nonuniform volume current.
4	Uniform normal current on 6-5-8-7 face.
5	Nonuniform normal current on 6-5-8-7 face.
6	Uniform normal current on 2-1-5-6 face.
7	Nonuniform normal current on 2-1-5-6 face.
8	Uniform normal current on 3-2-6-7 face.
9	Nonuniform normal current on 3-2-6-7 face.
10	Uniform normal current on 4-3-7-8 face.
11	Nonuniform normal current on 4-3-7-8 face.
12	Uniform normal current on 1-4-8-5 face.
13	Nonuniform normal current on 1-4-8-5 face.
20	Uniform normal current on 1-2-3-4 face.
21	Nonuniform current on 1-2-3-4 face.
22	Uniform volume current in the z-direction.
23	Nonuniform volume current.
24	Uniform normal current on 6-5-8-7 face.
25	Nonuniform current on 6-5-8-7 face.
26	Uniform normal current on 2-1-5-6 face.
27	Nonuniform current on 2-1-5-6 face.
28	Uniform normal current on 3-2-6-7 face.
29	Nonuniform current on 3-2-6-7 face.
30	Uniform normal current on 4-3-7-8 face.
31	Nonuniform current on 4-3-7-8 face.
32	Uniform normal current on 1-4-8-5 face.
33	Nonuniform current on 1-4-8-5 face.

Current Type	Description
40	Uniform shear current 1-2-3-4 face in 1⇒2 direction.
41	Nonuniform shear current 1-2-3-4 face in 1⇒2 direction.
42	Uniform shear current 1-2-3-4 face in 2⇒3 direction.
43	Nonuniform shear current 1-2-3-4 face in 2⇒3 direction.
48	Uniform shear current 6-5-8-7 face in 5⇒6 direction.
49	Nonuniform shear current 6-5-8-7 face in 5⇒6 direction.
50	Uniform shear current 6-5-8-7 face in 6⇒7 direction.
51	Nonuniform shear current 6-5-8-7 face in 6⇒7 direction.
52	Uniform shear current 2-1-5-6 face in 1⇒2 direction.
53	Nonuniform shear current 2-1-5-6 face in 1⇒2 direction.
54	Uniform shear current 2-1-5-6 face in 1⇒5 direction.
55	Nonuniform shear current 2-1-5-6 face in 1⇒5 direction.
56	Uniform shear current 3-2-6-7 face in 2⇒3 direction.
57	Nonuniform shear current 3-2-6-7 face in 2⇒3 direction.
58	Uniform shear current 3-2-6-7 face in 2⇒6 direction.
59	Nonuniform shear current 2-3-6-7 face in 2⇒6 direction.
60	Uniform shear current 4-3-7-8 face in 3⇒4 direction.
61	Nonuniform shear current 4-3-7-8 face in 3⇒4 direction.
62	Uniform shear current 4-3-7-8 face in 3⇒7 direction.
63	Nonuniform shear current 4-3-7-8 face in 3⇒7 direction.
64	Uniform shear current 1-4-8-5 face in 4⇒1 direction.
65	Nonuniform shear current 1-4-8-5 face in 4⇒1 direction.
66	Uniform shear current 1-4-8-5 face in 1⇒5 direction.
67	Nonuniform shear current 1-4-8-5 face in 1⇒5 direction.

For all nonuniform normal and shear currents, the magnitude is supplied through the FORCEM user subroutine.

Currents are positive into element face.

Distributed Charges

Charge types for distributed charges are listed below.

Charge Type	Description
70	Uniform current on 1-2-3-4 face.
71	Nonuniform charge on 1-2-3-4 face.

Charge Type	Description
74	Uniform charge on 5-6-7-8 face.
75	Nonuniform charge on 5-6-7-8 face.
76	Uniform charge on 1-2-6-5 face.
77	Nonuniform charge on 1-2-6-5 face.
78	Uniform charge on 2-3-7-6 face.
79	Nonuniform charge on 2-3-7-6 face
80	Uniform charge on 3-4-8-7 face.
81	Nonuniform charge on 3-4-8-7 face.
82	Uniform charge on 1-4-8-5 face.
83	Nonuniform charge on 1-4-8-5 face.

For all nonuniform charges, the magnitude is supplied through the FORCEM user subroutine.

Charges are positive into the element face.

Output

Three components of:

Electric field intensity	E
Electric flux density	D
Magnetic field intensity	H
Magnetic flux density	B
Current density	J

Transformation

Standard transformation of three global degrees of freedom to local degrees of freedom.

Output Points

Eight integration points as shown in Figure 3-177.

Element 114

Plane Stress Quadrilateral, Reduced Integration

Element type 114 is a four-node isoparametric arbitrary quadrilateral written for plane stress applications using reduced integration. This element uses an assumed strain formulation written in natural coordinates which insures good representation of the shear strains in the element.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using a single integration point at the centroid of the element. An additional variationally consistent stiffness term is included to eliminate the hourglass modes that are normally associated with reduced integration. The mass matrix of this element is formed using four-point Gaussian integration.

All constitutive models can be used with this element. As there is only a single integration point in the element, additional elements can be necessary to capture material nonlinearity such as plasticity.

Quick Reference

Type 114

Plane stress quadrilateral, using reduced integration.

Connectivity

Node numbering must follow right-handed convention (counterclockwise). See Figure 3-178.

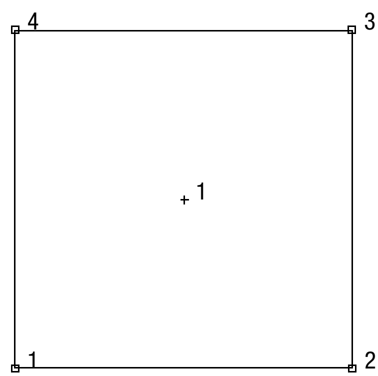


Figure 3-178 Plane Stress Quadrilateral

Geometry

The thickness is stored in the first data field (EGEOM1).

Default thickness is 1.

The second field is not used.

Coordinates

Two global coordinates, x and y directions.

Degrees of Freedom

1 = u (displacement in the global x-direction)
2 = v (displacement in the global y-direction)

Distributed Loads

Load types for distributed loads are as follows:

Load Type	Description
* 0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force by unit volume in second coordinate direction.
* 3	Nonuniform pressure on 1-2 face of the element; magnitude is supplied through the FORCEM user subroutine.
4	Nonuniform body force per unit volume in first coordinate direction; magnitude is supplied through the FORCEM user subroutine.
5	Nonuniform body force per unit volume in second coordinate direction; magnitude is supplied through the FORCEM user subroutine.
* 6	Uniform pressure on 2-3 face of the element.
* 7	Nonuniform pressure on 2-3 face of the element; magnitude is supplied through the FORCEM user subroutine.
* 8	Uniform pressure on 3-4 face of the element.
* 9	Nonuniform pressure on 3-4 face of the element; magnitude is supplied through the FORCEM user subroutine.
* 10	Uniform pressure on 4-1 face of the element.
* 11	Nonuniform pressure on 4-1 face of the element; magnitude is supplied through the FORCEM user subroutine.
* 20	Uniform shear force on side 1 - 2 (positive from 1 to 2).
21	Nonuniform shear force on side 1 - 2; magnitude is supplied through the FORCEM user subroutine.
* 22	Uniform shear force on side 2 - 3 (positive from 2 to 3).
* 23	Nonuniform shear force on side 2 - 3; magnitude is supplied through the FORCEM user subroutine.
* 24	Uniform shear force on side 3 - 4 (positive from 3 to 4).
* 25	Nonuniform shear force on side 3 - 4; magnitude is supplied through the FORCEM user subroutine.
* 26	Uniform shear force on side 4 - 1 (positive from 4 to 1).
* 27	Nonuniform shear force on side 4 - 1; magnitude is supplied through the FORCEM user subroutine.

Load Type	Description
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter two magnitudes of gravity acceleration in respectively global x- and y-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

All pressures are positive when directed into the element. Load types shown with an asterisk (*) require the magnitude of the load to be entered as force per unit area. To prescribe these loads in force per unit length, add 50 to the load type. This is often useful in design optimization where the thickness changes, but it is desired that the applied force remain the same. In addition, point loads can be applied at the nodes.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid of the element are:

ε_{xx}
ε_{yy}
γ_{xy}

Note:	Although $\varepsilon_{zz} = \frac{-\nu}{E}(\sigma_{xx} + \sigma_{yy})$, it is not printed and is posted as 0 for isotropic materials. For Mooney or Ogden (TL formulation) Marc post code 49 provides the thickness strain for plane stress elements. See Marc Volume A: Theory and User Information , Chapter 12 Output Results, Element Information for von Mises intensity calculation for strain.

Output of Stresses

Same as for Output of Strains .

Output Points

Output is available at the centroid.

Transformation

The two global degrees of freedom at the corner nodes can be transformed into local coordinates.

Tying

Use the UFORMSN user subroutine.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – output of true stress and logarithmic strain in global coordinate directions. Thickness is updated if the LARGE STRAIN parameter is specified.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 121. See Element 121 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux due to dissipation of plastic work specified with type 101.

Design Variables

The thickness can be considered a design variable.

Element 115

Arbitrary Quadrilateral Plane Strain, Reduced Integration

Element type 115 is a four-node isoparametric arbitrary quadrilateral written for plane strain applications using reduced integration. This element uses an assumed strain formulation written in natural coordinates which insures good representation of the shear strains in the element.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using a single integration point at the centroid of the element. An additional variationally consistent stiffness term is included to eliminate the hourglass modes that are normally associated with reduced integration. The mass matrix of this element is formed using four-point Gaussian integration.

This element can be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 118 instead. Element type 118 is also preferable for small strain incompressible elasticity. As there is only a single integration point in the element, additional elements may be necessary to capture material nonlinearity such as plasticity.

Quick Reference

Type 115

Plane stress quadrilateral, using reduced integration.

Connectivity

Four nodes per element. Node numbering follows right-handed convention (counterclockwise). See Figure 3-179.

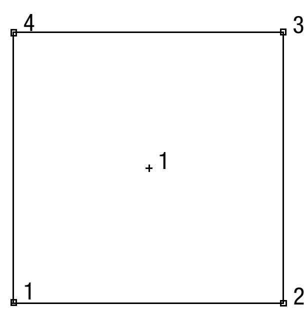


Figure 3-179 Nodes and Integration Point for Element 115

Geometry

The thickness is stored in the first data field (`EGEOM1`).

Default thickness is 1.

Coordinates

Two coordinates in the global -x and y-directions.

Degrees of Freedom

Global displacement degrees of freedom:

1 = u (displacement in the global x-direction)
2 = v (displacement in the global y-direction)

Distributed Loads

Load types for distributed loads as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force by unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element.
4	Nonuniform body force per unit volume in first coordinate direction.
5	Nonuniform body force per unit volume in second coordinate direction.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element.
8	Uniform pressure on 3-4 face of the element.
9	Nonuniform pressure on 3-4 face of the element.
10	Uniform pressure on 4-1 face of the element.
11	Nonuniform pressure on 4-1 face of the element.
20	Uniform shear force on side 1 - 2 (positive from 1 to 2).
21	Nonuniform shear force on side 1 - 2.
22	Uniform shear force on side 2 - 3 (positive from 2 to 3).
23	Nonuniform shear force on side 2 - 3.
24	Uniform shear force on side 3 - 4 (positive from 3 to 4).
25	Nonuniform shear force on side 3 - 4.
26	Uniform shear force on side 4 - 1 (positive from 4 to 1).
27	Nonuniform shear force on side 4 - 1.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.

Load Type	Description
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform pressures, body and shear forces, magnitude is supplied through the FORCEM user subroutine.

All pressures are positive when directed into the element. In addition, point loads can be applied at the nodes.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid of the element are:

$1 = \varepsilon_{xx}$

$2 = \varepsilon_{yy}$

$3 = \varepsilon_{zz} = 0$

$4 = \gamma_{xy}$

Output of Stresses

Same as for Output of Strains .

Transformation

The two global degrees of freedom at the corner nodes can be transformed into local coordinates.

Tying

Use the UFORMSN user subroutine.

Output Points

Output is available at the centroid.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain in global coordinate directions. This element does not lock for nearly incompressible materials.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 121. See Element 121 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux due to dissipation of plastic work specified with type 101.

Element 116

Arbitrary Quadrilateral Axisymmetric Ring, Reduced Integration

Element type 116 is a four-node isoparametric arbitrary quadrilateral written for axisymmetric applications using reduced integration. This element uses an assumed strain formulation written in natural coordinates which insures good representation of the shear strains in the element.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using a single integration point at the centroid of the element. An additional variationally consistent stiffness term is included to eliminate the hourglass modes that are normally associated with reduced integration. The mass matrix of this element is formed using four-point Gaussian integration.

This element can be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 80 instead. Element type 80 is also preferable for small strain incompressible elasticity. As there is only a single integration point in the element, additional elements may be necessary to capture material nonlinearity such as plasticity.

Quick Reference

Type 116

Axisymmetric, arbitrary ring with a quadrilateral cross section, using reduced integration.

Connectivity

Four nodes per element (see Figure 3-180). Node numbering for the corner nodes follows right-handed convention (counterclockwise).

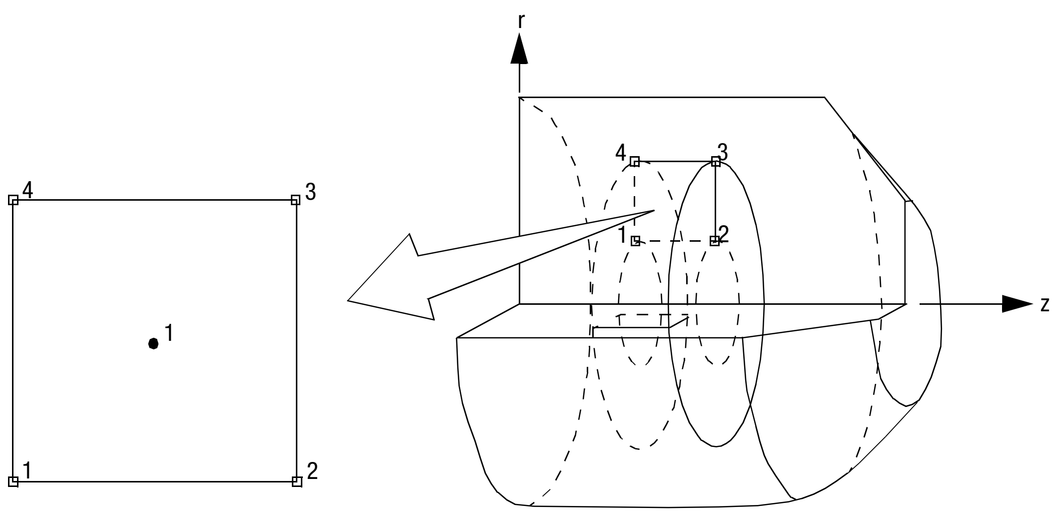


Figure 3-180 Integration Point for Element 116

Coordinates

Two coordinates in the global z and r directions.

Degrees of Freedom

Global displacement degrees of freedom at the corner nodes:

1 = u (displacement in the global z direction)
2 = v (displacement in the global r direction)

Distributed Loads

Load types for distributed loads as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force by unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element.
4	Nonuniform body force per unit volume in first coordinate direction.
5	Nonuniform body force per unit volume in second coordinate direction.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element.
8	Uniform pressure on 3-4 face of the element.
9	Nonuniform pressure on 3-4 face of the element.
10	Uniform pressure on 4-1 face of the element.
11	Nonuniform pressure on 4-1 face of the element.
20	Uniform shear force on 1-2 face in the 1⇒2 direction.
21	Nonuniform shear force on side 1 - 2.
22	Uniform shear force on side 2 - 3 (positive from 2 to 3).
23	Nonuniform shear force on side 2 - 3.
24	Uniform shear force on side 3 - 4 (positive from 3 to 4).
25	Nonuniform shear force on side 3 - 4
26	Uniform shear force on side 4 - 1 (positive from 4 to 1).
27	Nonuniform shear force on side 4 - 1.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Load Type	Description
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform pressures and shear forces, the magnitude is supplied via the FORCEM user subroutine.

All pressures are positive when directed into the element. In addition, point loads can be applied at the nodes.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid of the element in global coordinates is:

$1 = \varepsilon_{zz}$

$2 = \varepsilon_{rr}$

$3 = \varepsilon_{\theta\theta}$

$4 = \gamma_{rz}$

Output of Stresses

Same as for Output of Strains .

Transformation

The global degrees of freedom at the corner nodes can be transformed into local coordinates.

Tying

Can be tied to axisymmetric shell type 1 using standard tying type 23.

Output Points

Output is available at the centroid.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain in global coordinate directions. This element does not lock for nearly incompressible materials.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 122. See Element 122 for a description of the conventions used for entering the flux and film data for this element.

Element 117 Three-dimensional Arbitrarily Distorted Brick, Reduced Integration

Element type 117 is an eight-node isoparametric arbitrary hexahedral for general three-dimensional applications using reduced integration.This element uses an assumed strain formulation written in natural coordinates which insures good representation of the shear strains in the element.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using a single integration point at the centroid of the element. An additional variationally consistent stiffness term is included to eliminate the hourglass modes that are normally associated with reduced integration. The mass matrix of this element is formed using eight-point Gaussian integration.

This element can be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 120 instead. Element type 120 is also preferable for small strain incompressible elasticity. As there is only a single integration point in the element, additional elements may be necessary to capture material nonlinearity such as plasticity.

Note:	As in all three-dimensional analyses, a large nodal bandwidth results in long computing times. Optimize the nodal bandwidth.

Quick Reference

Type 117

Eight-node 3-D first-order isoparametric element (arbitrarily distorted cube) using reduced integration.

Connectivity

Eight nodes per element (see Figure 3-181). Node numbering must follow the scheme below:

> Nodes 1, 2, 3, and 4 are corners of one face, given in counterclockwise order when viewed from inside the element. Node 5 is the same edge as node 1, node 6 as node 2, node 7 as node 3, and node 8 as node 4.

Geometry

If the automatic brick to shell constraints are to be used, the first field must contain the transition thickness.

Coordinates

Three coordinates in the global x-, y-, and z-directions.

Degrees of Freedom

Three global degrees of freedom u, v, and w per node.

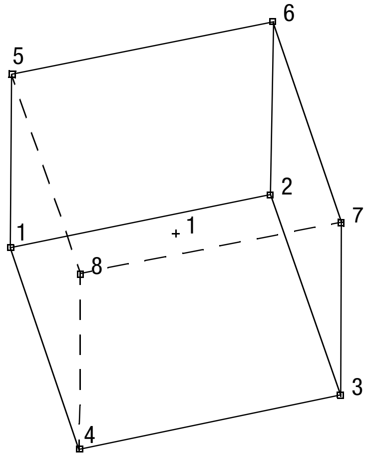


Figure 3-181 Nodes and Integration Point for Element 117

Distributed Loads

Distributed loads chosen by value of `IBODY` are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3-4 face.
1	Nonuniform pressure on 1-2-3-4 face.
2	Uniform body force per unit volume in -z direction.
3	Nonuniform body force per unit volume (e.g., centrifugal force); magnitude and direction is given in the FORCEM user subroutine.
4	Uniform pressure on 6-5-8-7 face.
5	Nonuniform pressure on 6-5-8-7 face.
6	Uniform pressure on 2-1-5-6 face.
7	Nonuniform pressure on 2-1-5-6 face.
8	Uniform pressure on 3-2-6-7 face.
9	Nonuniform pressure on 3-2-6-7 face.
10	Uniform pressure on 4-3-7-8 face.
11	Nonuniform pressure on 4-3-7-8 face.
12	Uniform pressure on 1-4-8-5 face.
13	Nonuniform pressure on 1-4-8-5 face.
20	Uniform pressure on 1-2-3-4 face.
21	Nonuniform load on 1-2-3-4 face.
22	Uniform body force per unit volume in -z direction.

Load Type	Description
23	Nonuniform body force per unit volume (for example, centrifugal force)
24	Uniform pressure on 6-5-8-7 face.
25	Nonuniform load on 6-5-8-7 face.
26	Uniform pressure on 2-1-5-6 face.
27	Nonuniform load on 2-1-5-6 face.
28	Uniform pressure on 3-2-6-7 face.
29	Nonuniform load on 3-2-6-7 face.
30	Uniform pressure on 4-3-7-8 face.
31	Nonuniform load on 4-3-7-8 face.
32	Uniform pressure on 1-4-8-5 face.
33	Nonuniform load on 1-4-8-5 face.
40	Uniform shear 1-2-3-4 face in 1⇒2 direction.
41	Nonuniform shear 1-2-3-4 face in 1⇒2 direction.
42	Uniform shear 1-2-3-4 face in 2⇒3 direction.
43	Nonuniform shear 1-2-3-4 face in 2⇒3 direction.
48	Uniform shear 6-5-8-7 face in 5⇒6 direction.
49	Nonuniform shear 6-5-8-7 face in 5⇒6 direction.
50	Uniform shear 6-5-8-7 face in 6⇒7 direction.
51	Nonuniform shear 6-5-8-7 face in 6⇒7 direction.
52	Uniform shear 2-1-5-6 face in 1⇒2 direction.
53	Nonuniform shear 2-1-5-6 face in 1⇒2 direction.
54	Uniform shear 2-1-5-6 face in 1⇒5 direction.
55	Nonuniform shear 2-1-5-6 face in 1⇒5 direction.
56	Uniform shear 3-2-6-7 face in 2⇒3 direction.
57	Nonuniform shear 3-2-6-7 face in 2⇒3 direction.
58	Uniform shear 3-2-6-7 face in 2⇒6 direction.
59	Nonuniform shear 2-3-6-7 face in 2⇒6 direction.
60	Uniform shear 4-3-7-8 face in 3⇒4 direction.
61	Nonuniform shear 4-3-7-8 face in 3⇒4 direction.
62	Uniform shear 4-3-7-8 face in 3⇒7 direction.
63	Nonuniform shear 4-3-7-8 face in 3⇒7 direction.
64	Uniform shear 1-4-8-5 face in 4⇒1 direction.

Load Type	Description
65	Nonuniform shear 1-4-8-5 face in 4⇒1 direction.
66	Uniform shear 1-4-8-5 face in 1⇒5 direction.
67	Nonuniform shear 1-4-8-5 face in 1⇒5 direction.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Pressure forces are positive into element face.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

$1 = \varepsilon_{xx}$ = global xx strain
$2 = \varepsilon_{yy}$ = global yy strain
$3 = \varepsilon_{zz}$ = global zz strain
$4 = \gamma_{xy}$ = global xy strain
$5 = \gamma_{yz}$ = global yz strain
$6 = \gamma_{zx}$ = global xz strain

Output of Stresses

Output of stress is the same as for Output of Strains .

Transformation

Standard transformation of three global degrees of freedom to local degrees of freedom at the corner nodes.

Tying

No special tying available. An automatic constraint is available for brick to shell transition meshes (see Geometry).

Output Points

Centroid.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 123. See Element 123 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux generated by dissipated plastic energy is specified with type 101.

Notes:	The element can be collapsed to a tetrahedron.
	By collapsing one plane of the element to a line (see Figure 3-182) a transition element for connecting bricks with a four-node shell element type 75 is generated. Thickness of the shell is specified in the geometry field.

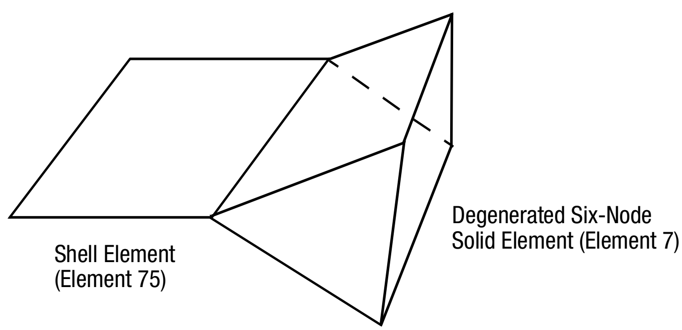


Figure 3-182 Shell-to-solid Automatic Constraint

Element 118

Arbitrary Quadrilateral Plane Strain, Incompressible Formulation with Reduced Integration

Element type 118 is a four-node isoparametric arbitrary quadrilateral written for incompressible plane strain applications using reduced integration. This element uses an assumed strain formulation written in natural coordinates which insures good representation of the shear strains in the element. The displacement formulation has been modified using the Herrmann variational principle, the pressure field is constant in this element.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using a single integration point at the centroid of the element. An additional variationally consistent stiffness term is included to eliminate the hourglass modes that are normally associated with reduced integration. The mass matrix of this element is formed using four-point Gaussian integration.

This element is designed to be used for incompressible elasticity only. It can be used for either small strain behavior or large strain behavior using the Mooney or Ogden models.This element can be used in conjunction with other elements such as type 115 when other material behavior, such as plasticity, must be represented.

Quick Reference

Type 118

Plane strain quadrilateral, Herrmann formulation, using reduced integration.

Connectivity

Five nodes per element (see Figure 3-183). Node numbering for the corners follows right-handed convention (counterclockwise). The fifth node only has a pressure degree of freedom and is not to be shared with other elements.

Note:	Avoid reducing this element into a triangle.

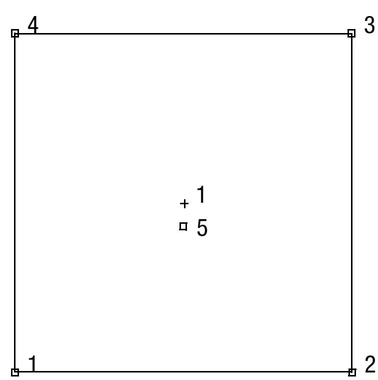


Figure 3-183 Nodes and Integration Point for Element Type 118

Geometry

The thickness is entered in the first data field (`EGEOM1`). Defaults to unit thickness.

Coordinates

Two coordinates in the global x- and y-directions for the corner nodes. No coordinates are necessary for the fifth hydrostatic pressure node.

Degrees of Freedom

Global displacement degrees of freedom at the corner nodes:

1 = u displacement (x-direction)
2 = v displacement (y-direction)

One degree of freedom (negative hydrostatic pressure) at the fifth node.

1	$= \sigma_{kk}/E$	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1 + \frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2 + \frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3 \right] \frac{9}{E_1 + E_2 + E_3}$$	
	= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)
	= p / K	(for Herrmann elements using additive decomposition; K here is the effective bulk modulus)

Distributed Loads

Load types for distributed loads are as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force by unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element.
4	Nonuniform body force per unit volume in first coordinate direction.
5	Nonuniform body force per unit volume in second coordinate direction.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element.
8	Uniform pressure on 3-4 face of the element.
9	Nonuniform pressure on 3-4 face of the element.
10	Uniform pressure on 4-1 face of the element.

Load Type	Description
11	Nonuniform pressure on 4-1 face of the element.
20	Uniform shear force on 1-2 face in the 1⇒2 direction.
21	Nonuniform shear force on side 1-2.
22	Uniform shear force on side 2-3 (positive from 2 to 3).
23	Nonuniform shear force on side 2-3.
24	Uniform shear force on side 3-4 (positive from 3 to 4).
25	Nonuniform shear force on side 3-4.
26	Uniform shear force on side 4-1 (positive from 4 to 1).
27	Nonuniform shear force on side 4-1.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

All pressures are positive when directed into the element. In addition, point loads can be applied at the nodes.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid of the element in global coordinates are:

1	$= \varepsilon_{xx}$	
2	$= \varepsilon_{yy}$	
3	$= \varepsilon_{zz} = 0$	

4	$= \gamma_{xy}$	
5	$= \sigma_{kk}/E$	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	= -p	= negative pressure (for Mooney, Ogden, or Soil)

$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1 + \frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2 + \frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3 \right] \frac{9}{E_1 + E_2 + E_3}$$

The last component is not printed in the output file, but is accessible via the ELEVAR and PLOTV user subroutines.

Output of Stresses

Same as for Output of Strains .

Transformation

The two global degrees of freedom at the corner nodes can be transformed into local coordinates.

Output Points

Output is available at the centroid.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 121. See Element 121 for a description of the conventions used for entering the flux and film data for this element.

Element 119 Arbitrary Quadrilateral Axisymmetric Ring, Incompressible Formulation with Reduced Integration

Element type 119 is a four-node isoparametric arbitrary quadrilateral written for incompressible axisymmetric applications using reduced integration. This element uses an assumed strain formulation written in natural coordinates which insures good representation of the shear strains in the element. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is constant in this element.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using a single integration point at the centroid of the element. An additional variationally consistent stiffness term is included to eliminate the hourglass modes that are normally associated with reduced integration. The mass matrix of this element is formed using four-point Gaussian integration.

This element is designed to be used for incompressible elasticity only. It can be used for either small strain behavior or large strain behavior using the Mooney or Ogden models.This element can be used in conjunction with other elements such as type 116 when other material behavior, such as plasticity, must be represented.

Quick Reference

Type 119

Axisymmetric, arbitrary ring with a quadrilateral cross-section, Herrmann formulation, using reduced integration.

Connectivity

Five nodes per element (see Figure 3-184). Node numbering for the corner nodes follows right-handed convention (counterclockwise). The fifth node only has a pressure degree of freedom and is not to be shared with other elements.

Note:	Avoid reducing the element into a triangle.

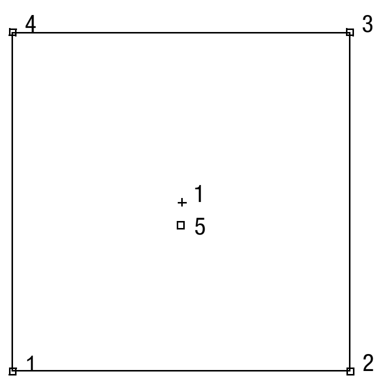


Figure 3-184 Integration Point for Element 119

Coordinates

Two coordinates in the global z- and r-directions for the corner nodes. No coordinates are necessary for the hydrostatic pressure node.

Degrees of Freedom

Global displacement degrees of freedom at the corner nodes:

1 = axial displacement (in z-direction)
2 = radial displacement (in r-direction)

One degree of freedom (negative hydrostatic pressure) at the fifth node:

1	$= \sigma_{kk}/E$	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1 + \frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2 + \frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3 \right] \frac{9}{E_1+E_2+E_3}$$	
	= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)
	= p / K	(for Herrmann elements using additive decomposition; K here is the effective bulk modulus)

Distributed Loads

Load types for distributed loads are as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force by unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element.
4	Nonuniform body force per unit volume in first coordinate direction.
5	Nonuniform body force per unit volume in second coordinate direction.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element.
8	Uniform pressure on 3-4 face of the element.
9	Nonuniform pressure on 3-4 face of the element.
10	Uniform pressure on 4-1 face of the element.
11	Nonuniform pressure on 4-1 face of the element.
20	Uniform shear force on 1-2 face in the 1⇒2 direction.
21	Nonuniform shear force on side 1-2.

Load Type	Description
22	Uniform shear force on side 2-3 (positive from 2 to 3).
23	Nonuniform shear force on side 2-3.
24	Uniform shear force on side 3-4 (positive from 3 to 4).
25	Nonuniform shear force on side 3-4
26	Uniform shear force on side 4-1 (positive from 4 to 1).
27	Nonuniform shear force on side 4-1.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

All pressures are positive when directed into the element. In addition, point loads can be applied at the nodes.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid of the element in global coordinates is:

1	$= \varepsilon_{zz}$	
2	$= \varepsilon_{rr}$	
3	$= \varepsilon_{\theta\theta}$	
4	$= \gamma_{rz}$	

5	$= \sigma_{kk}/E$	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1 + \frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2 + \frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3 \right] \frac{9}{E_1 + E_2 + E_3}$$	
	= -p	= negative pressure (for Mooney, Ogden, or Soil)

The last component is not printed in the output file, but is accessible via the ELEVAR and PLOTV user subroutines.

Output of Stresses

Same as for Output of Strains .

Transformation

The global degrees of freedom at the corner nodes can be transformed into local coordinates. No transformation for the pressure node.

Tying

Can be tied to axisymmetric shell type 1 using standard tying type 23.

Output Points

Output is available at the centroid.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 121. See Element 121 for a description of the conventions used for entering the flux and film data for this element.

Element 120

Three-dimensional Arbitrarily Distorted Brick, Incompressible Reduced Integration

Element type 120 is an eight-node isoparametric arbitrary hexahedral written for general three dimensional incompressible applications using reduced integration. This element uses an assumed strain formulation written in natural coordinates which insures good representation of the shear strains in the element. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is constant in this element.

This element is preferred over higher-order elements when used in a contact analysis.

The stiffness of this element is formed using a single integration point at the centroid of the element. An additional variationally consistent stiffness term is included to eliminate the hourglass modes that are normally associated with reduced integration. The mass matrix of this element is formed using eight-point Gaussian integration.

This element is designed to be used for incompressible elasticity only. It can be used for either small strain behavior or large strain behavior using the Mooney or Ogden models.This element can be used in conjunction with other elements such as type 117 when other material behavior, such as plasticity, must be represented.

Quick Reference

Type 120

Nine-node 3-D first-order isoparametric element (arbitrarily distorted cube) with mixed formulation, using reduced integration.

Connectivity

Nine nodes per element (see Figure 3-185).

Node numbering must follow the scheme below:

> Nodes 1, 2, 3, and 4 are corners of one face, given in counterclockwise order when viewed from inside the element. Node 5 is the same edge as node 1, node 6 as node 2, node 7 as node 3, and node 8 as node 4.

The node with the pressure degree of freedom is the last node in the connectivity list, and should not be shared with other elements.

Note:	Avoid reducing this element into a tetrahedron or a wedge.

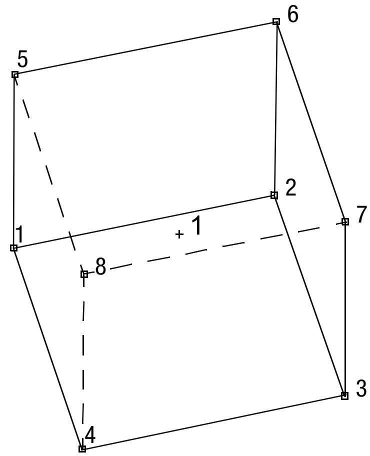


Figure 3-185 Nodes and Integration Point for Element 120

Coordinates

Three coordinates in the global x-, y-, and z-directions for the first eight nodes. No coordinates are necessary for the pressure node.

Degrees of Freedom

Three global degrees of freedom u, v, and w at the first eight nodes. One degree of freedom (negative hydrostatic pressure) at the last node.

1	$= \sigma_{kk}/E$	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1+\frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2+\frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3\right]\frac{9}{E_1+E_2+E_3}$$	
	= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)
	= p / K	(for Herrmann elements using additive decomposition; K here is the effective bulk modulus)

Distributed Loads

Distributed loads chosen by value of IBODY are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3-4 face.
1	Nonuniform pressure on 1-2-3-4 face.
2	Uniform body force per unit volume in -z direction.

Load Type	Description
3	Nonuniform body force per unit volume (for example, centrifugal force); magnitude and direction given in the FORCEM user subroutine.
4	Uniform pressure on 6-5-8-7 face.
5	Nonuniform pressure on 6-5-8-7 face.
6	Uniform pressure on 2-1-5-6 face.
7	Nonuniform pressure on 2-1-5-6 face.
8	Uniform pressure on 3-2-6-7 face.
9	Nonuniform pressure on 3-2-6-7 face.
10	Uniform pressure on 4-3-7-8 face.
11	Nonuniform pressure on 4-3-7-8 face.
12	Uniform pressure on 1-4-8-5 face.
13	Nonuniform pressure on 1-4-8-5 face.
20	Uniform pressure on 1-2-3-4 face.
21	Nonuniform load on 1-2-3-4 face.
22	Uniform body force per unit volume in -z direction.
23	Nonuniform body force per unit volume (for example, centrifugal force).
24	Uniform pressure on 6-5-8-7 face.
25	Nonuniform load on 6-5-8-7 face.
26	Uniform pressure on 2-1-5-6 face.
27	Nonuniform load on 2-1-5-6 face.
28	Uniform pressure on 3-2-6-7 face.
29	Nonuniform load on 3-2-6-7 face.
30	Uniform pressure on 4-3-7-8 face.
31	Nonuniform load on 4-3-7-8 face.
32	Uniform pressure on 1-4-8-5 face.
33	Nonuniform load on 1-4-8-5 face.
40	Uniform shear 1-2-3-4 face in $1 \Rightarrow 2$ direction.
41	Nonuniform shear 1-2-3-4 face in $1 \Rightarrow 2$ direction.
42	Uniform shear 1-2-3-4 face in $2 \Rightarrow 3$ direction.
43	Nonuniform shear 1-2-3-4 face in $2 \Rightarrow 3$ direction.
48	Uniform shear 6-5-8-7 face in $5 \Rightarrow 6$ direction.
49	Nonuniform shear 6-5-8-7 face in $5 \Rightarrow 6$ direction.
50	Uniform shear 6-5-8-7 face in $6 \Rightarrow 7$ direction.

Load Type	Description
51	Nonuniform shear 6-5-8-7 face in 6⇒7 direction.
52	Uniform shear 2-1-5-6 face in 1⇒2 direction.
53	Nonuniform shear 2-1-5-6 face in 1⇒2 direction.
54	Uniform shear 2-1-5-6 face in 1⇒5 direction.
55	Nonuniform shear 2-1-5-6 face in 1⇒5 direction.
56	Uniform shear 3-2-6-7 face in 2⇒3 direction.
57	Nonuniform shear 3-2-6-7 face in 2⇒3 direction.
58	Uniform shear 3-2-6-7 face in 2⇒
59	Nonuniform shear 2-3-6-7 face in 2⇒6 direction.
60	Uniform shear 4-3-7-8 face in 3⇒4 direction.
61	Nonuniform shear 4-3-7-8 face in 3⇒4 direction.
62	Uniform shear 4-3-7-8 face in 3⇒7 direction.
63	Nonuniform shear 4-3-7-8 face in 3⇒7 direction.
64	Uniform shear 1-4-8-5 face in 4⇒1 direction.
65	Nonuniform shear 1-4-8-5 face in 4⇒1 direction.
66	Uniform shear 1-4-8-5 face in 1⇒5 direction.
67	Nonuniform shear 1-4-8-5 face in 1⇒5 direction.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global, x-, y-, z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform pressures, loads, and shear forces, magnitude is supplied via the FORCEM user subroutine.

Pressure forces are positive into element face.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

1	= ε_{xx}	= global xx strain
2	= ε_{yy}	= global yy strain
3	= ε_{zz}	= global zz strain
4	= γ_{xy}	= global xy strain
5	= γ_{yz}	= global yz strain
6	= γ_{zx}	= global zx strain
7	= σ_{kk}/E	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)

$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1+\frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\Delta_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2+\frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3\right]\frac{9}{E_1+E_2+E_3}$$

The last component is not printed in the output file, but is accessible via the ELEVAR and PLOTV user subroutines.

Output of Stresses

Output of stress is the same as for Output of Strains .

Transformation

Standard transformation of three global degrees of freedom to local degrees of freedom at the corner nodes. No transformation for the pressure node.

Tying

No special tying available.

Output Points

Centroid.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 123. See Element 123 for a description of the conventions used for entering the flux and film data for this element.

Element 121 Planar Bilinear Quadrilateral, Reduced Integration (Heat Transfer Element)

Element type 121 is a four-node isoparametric arbitrary quadrilateral written for planar heat transfer applications using reduced integration. This element can also be used for electrostatic or magnetostatic applications.

As this element uses bilinear interpolation functions, the thermal gradients tend to be constant throughout the element.

This element can also be used as a thermal contact or a fluid channel element. The CONRAD GAP and CHANNEL model definition options must be used for thermal contact and fluid channel options, respectively. A description of the thermal contact and fluid channel capabilities is included in Marc Volume A: Theory and User Information . Note that in thermal contact and fluid channel options, the gap face and fluid channel face identifications must be entered for each gap/channel. Face identifications for this element are given in the Face Identifications (Thermal Contact Gap and Fluid Channel Options) .

In general, you need more of these lower-order elements than the higher-order elements such as types 41 or 69. Hence, use a fine mesh.

The conductivity of this element is formed using a single integration point at the centroid of the element. An additional variationally consistent conductivity term is included to eliminate the hourglass modes that are normally associated with reduced integration. The specific heat capacity matrix of this element is formed using four-point Gaussian integration.

Quick Reference

Type 121

Arbitrary, planar, heat transfer quadrilateral, using reduced integration.

Connectivity

Node numbering must follow right-hand convention (see Figure 3-186).

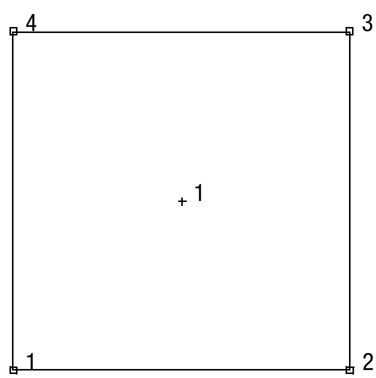


Figure 3-186 Nodes and Integration Point for Element 121

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)

Fluxes

Flux types for distributed fluxes are as follows:

Flux Type	Description
0	Uniform flux per unit area 1-2 face of the element.
1	Uniform flux per unit volume on whole element.
2	Uniform flux per unit volume on whole element.
3	Nonuniform flux per unit area on 1-2 face of the element.
4	Nonuniform flux per unit volume on whole element.
5	Nonuniform flux per unit volume on whole element.
6	Uniform flux per unit area on 2-3 face of the element.
7	Nonuniform flux per unit area on 2-3 face of the element.
8	Uniform flux per unit area on 3-4 face of the element.
9	Nonuniform flux per unit area on 3-4 face of the element.
10	Uniform flux per unit area on 4-1 face of the element.
11	Nonuniform flux per unit area on 4-1 face of the element.

For all nonuniform fluxes, the magnitude is given in the FLUX user subroutine.

All fluxes are positive when adding heat to the element. In addition, point fluxes can be applied at the nodes. Edge fluxes are evaluated using a two-point integration scheme where the integration points have the same location as the nodal points.

Films

Same specification as Fluxes .

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Current

Same specifications as Fluxes .

Charge

Same specifications as Fluxes .

Output Points

Centroid.

Tying

Use the UFORMSN user subroutine.

Face Identifications (Thermal Contact Gap and Fluid Channel Options)

Gap Face Identification	Radiative/Convective Heat Transfer Takes Place Between Edges
1	(1 - 2) and (3 - 4)
2	(1 - 4) and (2 - 3)

Fluid Channel Face Identification	Flow Direction From Edge to Edge
1	(1 - 2) to (3 - 4)
2	(1 - 4) to (2 - 3)

Element 122 Axisymmetric Bilinear Quadrilateral, Reduced Integration (Heat Transfer Element)

Element type 122 is a four-node isoparametric arbitrary quadrilateral written for axisymmetric heat transfer applications using reduced integration. This element can also be used for electrostatic or magnetostatic applications.

As this element uses bilinear interpolation functions, the thermal gradients tend to be constant throughout the element.

This element can also be used as a thermal contact or a fluid channel element. The CONRAD GAP and CHANNEL model definition options must be used for thermal contact and fluid channel options, respectively. A description of the thermal contact and fluid channel capabilities is included in Marc Volume A: Theory and User Information . Note that in thermal contact and fluid channel options, the gap face and fluid channel face identifications must be entered for each gap/channel. Face identifications for this element are given in the Face Identifications (Thermal Contact Gap and Fluid Channel Options) . The view factors calculation for radiation boundary conditions is available for this axisymmetric heat transfer element.

In general, you need more of these lower-order elements than the higher-order elements such as types 42 or 70. Hence, use a fine mesh.

The conductivity of this element is formed using a single integration point at the centroid of the element. An additional variationally consistent conductivity term is included to eliminate the hourglass modes that are normally associated with reduced integration. The specific heat capacity matrix of this element is formed using four-point Gaussian integration.

Quick Reference

Type 122

Arbitrarily distorted axisymmetric heat transfer quadrilateral, using reduced integration.

Connectivity

Node numbering must follow right-hand convention (see Figure 3-187).

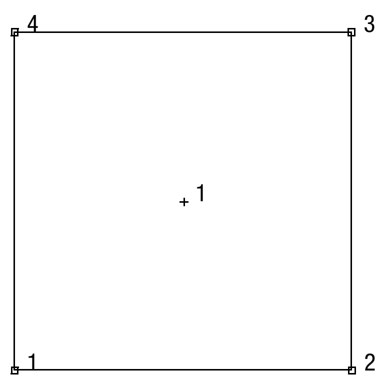


Figure 3-187 Nodes and Integration Point for Element 122

Geometry

Not applicable.

Coordinates

Two global coordinates, z and r.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)

Fluxes

Flux types for distributed fluxes are as follows:

Flux Type	Description
0	Uniform flux per unit area 1-2 face of the element.
1	Uniform flux per unit volume on whole element.
2	Uniform flux per unit volume on whole element.
3	Nonuniform flux per unit area on 1-2 face of the element.
4	Nonuniform flux per unit volume on whole element.
5	Nonuniform flux per unit volume on whole element.
6	Uniform flux per unit area on 2-3 face of the element.
7	Nonuniform flux per unit area on 2-3 face of the element.
8	Uniform flux per unit area on 3-4 face of the element.
9	Nonuniform flux per unit area on 3-4 face of the element.
10	Uniform flux per unit area on 4-1 face of the element.
11	Nonuniform flux per unit area on 4-1 face of the element.

For all nonuniform fluxes, the magnitude is given in the FLUX user subroutine.

All fluxes are positive when adding heat to the element. In addition, point fluxes can be applied at the nodes. Edge fluxes are evaluated using a two-point integration scheme where the integration points have the same location as the nodal points.

Films

Same specification as Fluxes .

Tying

Use the UFORMSN user subroutine.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Current

Same specifications as Fluxes .

Charge

Same specifications as Fluxes .

Output Points

Centroid.

Face Identifications (Thermal Contact Gap and Fluid Channel Options)

Gap Face Identification	Radiative/Convective Heat Transfer Takes Place Between Edges
1	(1 - 2) and (3 - 4)
2	(1 - 4) and (2 - 3)

Fluid Channel Face Identification	Flow Direction From Edge to Edge
1	(1 - 2) to (3 - 4)
2	(1 - 4) to (2 - 3)

View Factors Calculation for Radiation

Capability is available.

Element 123 Three-dimensional Eight-node Brick, Reduced Integration (Heat Transfer Element)

Element type 123 is an eight-node isoparametric arbitrary hexahedral written for three dimensional heat transfer applications using reduced integration. This element can also be used for electrostatic applications.

Element type 123 can also be used as a thermal contact or a fluid channel element. The CONRAD GAP and CHANNEL model definition options must be used for thermal contact and fluid channel options, respectively. A description of the thermal contact and fluid channel capabilities is included in Marc Volume A: Theory and User Information . Note that in thermal contact and fluid channel options, the gap face and fluid channel face identifications must be entered for each gap/channel. Face identifications for this element are given in the Face Identifications (Thermal Contact Gap and Fluid Channel Options) .

As this element uses trilinear interpolation functions, the thermal gradients tend to be constant throughout the element.

In general, you need more of these lower-order elements than the higher-order elements such as types 44 or 71. Hence, use a fine mesh.

The conductivity of this element is formed using a single integration point at the centroid of the element. An additional variationally consistent stiffness term is included to eliminate the hourglass modes that are normally associated with reduced integration. The specific heat capacity matrix of this element is formed using eight-point Gaussian integration.

Quick Reference

Type 123

Eight-node, 3-D, first-order, isoparametric heat transfer element using reduced integration.

Connectivity

Eight nodes per element, labeled as follows:

> Nodes 1-4 are the corners on one face, numbered in a counterclockwise direction when viewed from inside the element. Nodes 5-8 are the nodes on the other face, with node 5 opposite node 1, and so on (see Figure 3-188).

Geometry

Not applicable for this element.

Coordinates

Three coordinates in the global x-, y-, and z-directions.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)

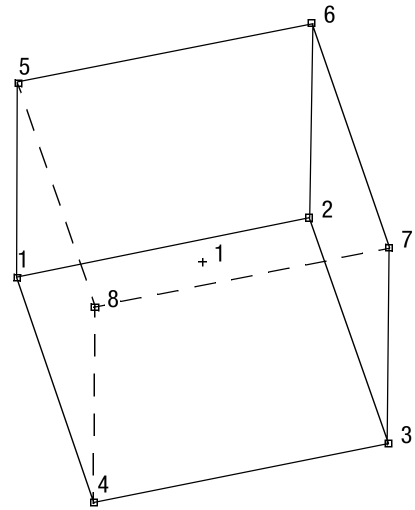


Figure 3-188 Nodes and Integration Point for Element 123

Fluxes

Fluxes are distributed according to the appropriate selection of a value of `IBODY`. Surface fluxes are assumed positive when directed into the element and are evaluated using a 4-point integration scheme, where the integration points have the same location as the nodal points.

Load Type (IBODY)	Description
0	Uniform flux on 1-2-3-4 face.
1	Nonuniform surface flux on 1-2-3-4 face.
2	Uniform volumetric flux.
3	Nonuniform volumetric flux.
4	Uniform flux on 5-6-7-8 face.
5	Nonuniform surface flux on 5-6-7-8 face.
6	Uniform flux on 1-2-6-5 face.
7	Nonuniform flux on 1-2-6-5 face.
8	Uniform flux on 2-3-7-6 face.
9	Nonuniform flux on 2-3-7-6 face.
10	Uniform flux on 3-4-8-7 face.
11	Nonuniform flux on 3-4-8-7 face.
12	Uniform flux on 1-4-8-5 face.
13	Nonuniform flux on 1-4-8-5 face.

For all nonuniform fluxes, the magnitude is supplied via the FLUX user subroutine.

For IBODY= 3, P is the magnitude of volumetric flux at volumetric integration point NN of element N. For IBODY odd but not equal to 3, P is the magnitude of surface flux for surface integration point NN of element N.

Films

Same specification as Fluxes .

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Current

Same specification as Fluxes .

Charge

Same specifications as Fluxes .

Output Points

Centroid.

Note:	As in all three-dimensional analysis a large nodal bandwidth results in long computing times. Use the optimizers as much as possible.

Face Identifications (Thermal Contact Gap and Fluid Channel Options)

Gap Face Identification	Radiative/Convective Heat Transfer Takes Place Between Faces
1	(1 - 2 - 6 - 5) and (3 - 4 - 8 - 7)
2	(2 - 3 - 7 - 6) and (1 - 4 - 8 - 5)
3	(1 - 2 - 3 - 4) and (5 - 6 - 7 - 8)

Fluid Channel Face Identification	Flow Direction From Edge to Edge
1	(1 - 2 - 6 - 5) to (3 - 4 - 8 - 7)
2	(2 - 3 - 7 - 6) to (1 - 4 - 8 - 5)
3	(1 - 2 - 3 - 4) to (5 - 6 - 7 - 8)

Element 124

Plane Stress, Six-node Distorted Triangle

This is a second-order isoparametric two-dimensional plane stress triangular element. Displacement and position (coordinates) within the element are interpolated from six sets of nodal values, the three corners and the three midsides (see Figure 3-189). The interpolation function is such that each edge has parabolic variation along itself. This allows for an accurate representation of the strain field in elastic analyses.

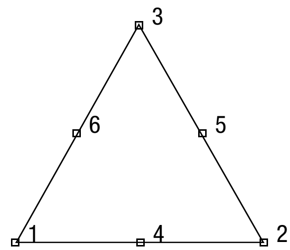


Figure 3-189 Nodes of Six-node, 2-D Element

The stiffness of this element is formed using three-point integration.

All constitutive relations can be used with this element.

The connectivity ordering is shown in Figure 3-189. Note that the basic numbering is counterclockwise in the (x-y) plane.

The second-order elements are usually preferred to a more refined mesh of first-order elements for most problems, especially when bending behavior in the plane of the elements is expected, since the basic strain variation in the second-order elements is linear in any direction.

Quick Reference

Type 124

Second order, isoparametric, distorted triangle. Plane stress.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention). Then the fourth node between first and second; the fifth between second and third, etc. See Figure 3-189.

Geometry

Thickness stored in first data field (`EGEOM1`). Default thickness is unity. Other fields are not used.

Coordinates

Two global coordinates, x and y, at each node.

Degrees of Freedom

Two at each node:

1 = u = global x-direction displacement
2 = v = global y-direction displacement

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type (IBODY)	Description
* 0	Uniform pressure on 1-4-2 face.
* 1	Nonuniform pressure on 1-4-2 face.
* 2	Uniform shear on 1-4-2 face.
* 3	Nonuniform shear on 1-4-2 face.
* 4	Uniform pressure on 2-5-3 face.
* 5	Nonuniform pressure on 2-5-3 face.
* 6	Uniform shear on 2-5-3 face.
* 7	Nonuniform shear on 2-5-3 face.
* 8	Uniform pressure on 3-6-1 face.
* 9	Nonuniform pressure on 3-6-1 face.
* 10	Uniform shear on 3-6-1 face.
* 11	Nonuniform shear on 3-6-1 face.
* 12	Uniform body force in x-direction.
* 13	Nonuniform body force in x-direction.
14	Uniform body force in y-direction.
15	Nonuniform body force in y direction.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter two magnitudes of gravity acceleration in respectively global x, y direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine. Load types shown with an asterisk (*) require the magnitude of the load to be entered as force per unit area. To prescribe these loads in force per unit length, add 50 to the load type. This is often useful in design optimization where the thickness changes, but it is desired that the applied force remain the same.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid or element integration points (see Figure 3-190 and Output Points) in the following order:

1 = ε_{xx}, direct

2 = ε_{yy}, direct

3 = γ_{xy}, shear

Note:	Although $\varepsilon_{zz} = \frac{-\nu}{E}(\sigma_{xx} + \sigma_{yy})$, it is not printed and is posted as 0 for isotropic materials. For Mooney or Ogden (TL formulation) Marc post code 49 provides the thickness strain for plane stress elements. See Marc Volume A: Theory and User Information , Chapter 12 Output Results, Element Information for von Mises intensity calculation for strain.

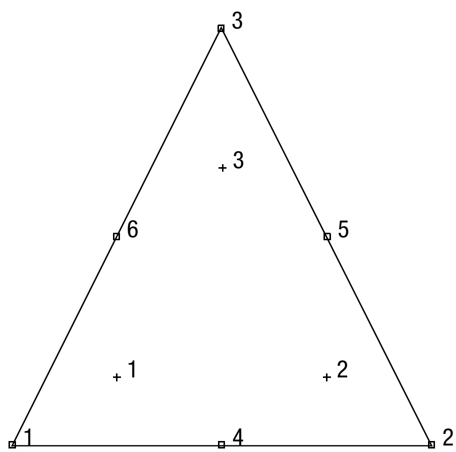


Figure 3-190 Integration Points of Six-node, 2-D Element

Output of Stresses

Output of stresses is the same as Output of Strains .

Transformation

Only in x-y plane.

Tying

Use the UFORMSN user subroutine.

Output Points

If the CENTROID parameter is used, output occurs at the centroid of the element.

If the ALL POINTS parameter is used, three output points are given, as shown in Figure 3-190. This is the usual option for a second-order element.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – output of stress and strain in global coordinates. Thickness is updated.

Note:	Distortion of element during solution can cause poor results. Element type 3 is preferred.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 131. See Element 131 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered as a design variable.

Element 125

Plane Strain, Six-node Distorted Triangle

This is a second-order isoparametric two-dimensional plane strain triangular element. Displacement and position (coordinates) within the element are interpolated from six sets of nodal values, the three corners and the three midsides (see Figure 3-191). The interpolation function is such that each edge has parabolic variation along itself. This allows for an accurate representation of the strain field in elastic analyses.

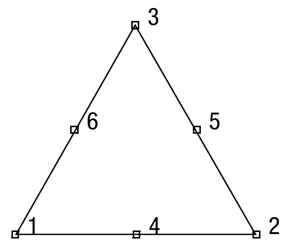


Figure 3-191 Nodes of Six-node, 2-D Element

The stiffness of this element is formed using three-point integration.

This element can be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 128. Element type 128 is also preferable for small strain incompressible elasticity.

The connectivity ordering is shown in Figure 3-191. Note that the basic numbering is counterclockwise in the (x-y) plane.

The second-order elements are usually preferred to a more refined mesh of first-order elements for most problems, especially when bending behavior in the plane of the elements is expected, since the basic strain variation in the second-order elements is linear in any direction.

Quick Reference

Type 125

Second order isoparametric distorted triangle. Plane strain.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention). Then the fourth node between first and second; the fifth node between second and third, etc. See Figure 3-191.

Geometry

Thickness stored in first data field (`EGEOM1`). Default thickness is unity. Other fields are not used.

Coordinates

Two global coordinates, x and y, at each node.

Degrees of Freedom

Two at each node:

1 = u = global x-direction displacement
2 = v = global y-direction displacement

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-4-2 face.
1	Nonuniform pressure on 1-4-2 face.
2	Uniform shear on 1-4-2 face.
3	Nonuniform shear on 1-4-2 face.
4	Uniform pressure on 2-5-3 face.
5	Nonuniform pressure on 2-5-3 face.
6	Uniform shear on 2-5-3 face.
7	Nonuniform shear on 2-5-3 face.
8	Uniform pressure on 3-6-1 face.
9	Nonuniform pressure on 3-6-1 face.
10	Uniform shear on 3-6-1 face.
11	Nonuniform shear on 3-6-1 face.
12	Uniform body force in x-direction.
13	Nonuniform body force in x-direction.
14	Uniform body force in y-direction.
15	Nonuniform body force in y-direction.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter two magnitudes of gravity acceleration in respectively global x, y direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Load Type (IBODY)	Description
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid or element integration points (see Figure 3-192 and Output Points) in the following order:

1 = ε_{xx}, direct

2 = ε_{yy}, direct

3 = ε_{zz}, direct

4 = γ_{xy}, shear

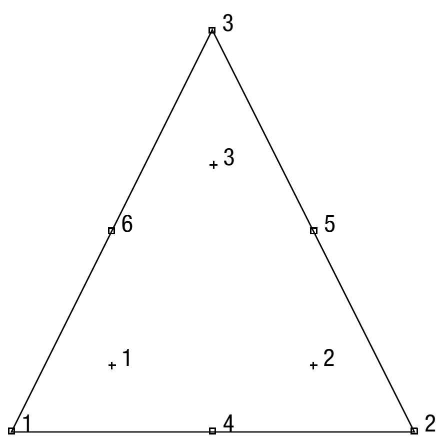


Figure 3-192 Integration Points of Six-node, 2-D Element

Output of Stresses

Output of stresses is the same as Output of Strains .

Transformation

Only in x-y plane.

Tying

Use the UFORMSN user subroutine.

Output Points

If the CENTROID parameter is used, the output occurs at the centroid of the element.

If the ALL POINTS parameter is used, three output points are given, as shown in Figure 3-192. This is the usual option for a second order element.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – output of stress and strain in global coordinates. Thickness is updated.

Note:	Distortion of element during solution can cause poor results. Element type 11 is preferred.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 131. See Element 131 for a description of the conventions used for entering the flux and film data for this element.

Element 126

Axisymmetric, Six-node Distorted Triangle

This is a second-order isoparametric two-dimensional axisymmetric triangular element. Displacement and position (coordinates) within the element are interpolated from six sets of nodal values, the three corners and the three midsides (see Figure 3-193). The interpolation function is such that each edge has parabolic variation along itself. This allows for an accurate representation of the strain field in elastic analyses.

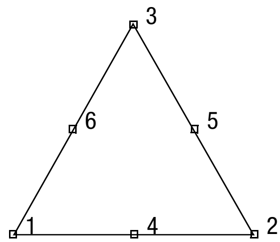


Figure 3-193 Nodes of Six-node, Axisymmetric Element

The stiffness of this element is formed using three-point integration.

This element can be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 129. Element type 129 is also preferable for small strain incompressible elasticity.

The second-order elements are usually preferred to a more refined mesh of first-order elements for most problems, especially when bending behavior in the plane of the elements is expected, since the basic strain variation in the second-order elements is linear in any direction.

Quick Reference

Type 126

Second-order isoparametric distorted triangle. Axisymmetric.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention). Then the fourth node between first and second; the fifth node between second and third, etc. See Figure 3-193.

Geometry

Geometry input is not necessary for this element.

Coordinates

Two global coordinates, z and r, at each node.

Degrees of Freedom

Two at each node:

1 = u = global z-direction displacement
2 = v = global r-direction displacement

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-4-2 face.
1	Nonuniform pressure on 1-4-2 face.
2	Uniform shear on 1-4-2 face.
3	Nonuniform shear on 1-4-2 face.
4	Uniform pressure on 2-5-3 face.
5	Nonuniform pressure on 2-5-3 face.
6	Uniform shear on 2-5-3 face.
7	Nonuniform shear on 2-5-3 face.
8	Uniform pressure on 3-6-1 face.
9	Nonuniform pressure on 3-6-1 face.
10	Uniform shear on 3-6-1 face.
11	Nonuniform shear on 3-6-1 face.
12	Uniform body force in z-direction.
13	Nonuniform body force in z-direction.
14	Uniform body force in r-direction.
15	Nonuniform body force in r-direction.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter two magnitudes of gravity acceleration in respectively global x, r direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Load Type (IBODY)	Description
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid or element integration points (see Figure 3-194 and Output Points) in the following order:

1 = ε_{zz}, direct

2 = ε_{rr}, direct

3 = $\varepsilon_{\theta\theta}$, hoop direct

4 = γ_{zr}, shear in the section

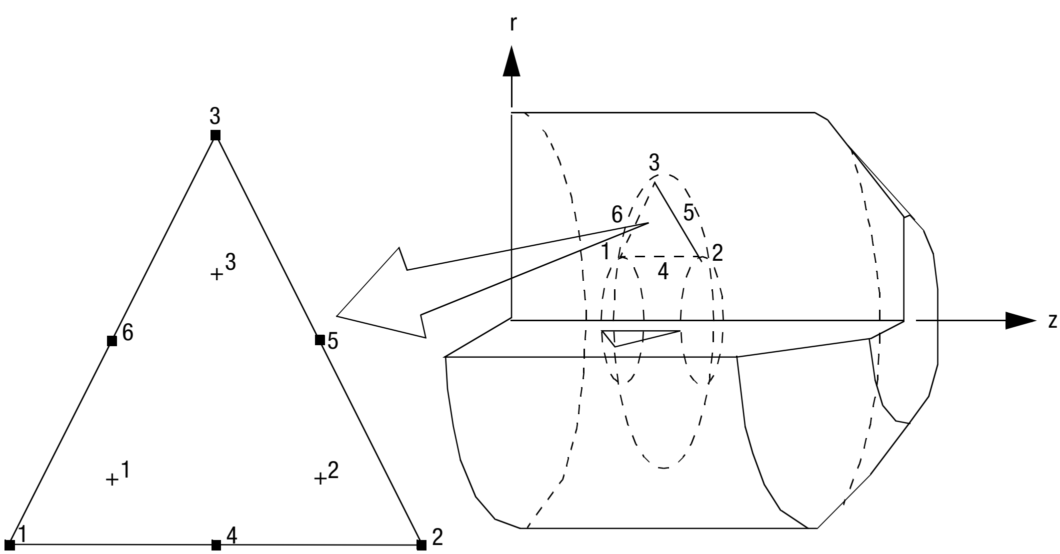


Figure 3-194 Integration Points of Six-node, Axisymmetric Element

Output of Stresses

Output of stresses is the same as Output of Strains .

Transformation

Only in z-r plane.

Tying

Use the UFORMSN user subroutine.

Output Points

If the CENTROID parameter is used, the output occurs at the centroid of the element.

If the ALL POINTS parameter is used, three output points are given, as shown in Figure 3-194. This is the usual option for a second-order element.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available - output of stress and strain in global coordinates. Thickness is updated.

Note:	Distortion of element during solution can cause poor results. Element type 10 is preferred.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 132. See Element 132 for a description of the conventions used for entering the flux and film data for this element.

Element 127

Three-dimensional Ten-node Tetrahedron

This element is a second-order isoparametric three-dimensional tetrahedron. Each edge forms a parabola so that four nodes define the corners of the element and a further six nodes define the position of the "midpoint" of each edge (Figure 3-195). This allows for an accurate representation of the strain field in elastic analyses.

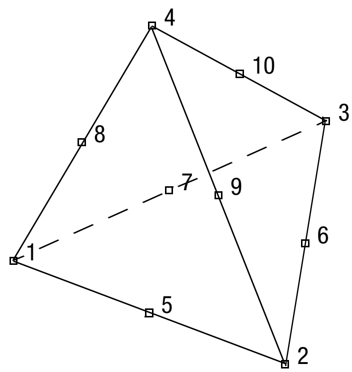


Figure 3-195 Form of Element 127

The stiffness of this element is formed using four-point integration. The mass matrix of this element is formed using sixteen-point Gaussian integration.

This element can be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 130. Element type 130 is also preferable for small strain incompressible elasticity.

Geometry

The geometry of the element is interpolated from the Cartesian coordinates of ten nodes.

Connectivity

The convention for the ordering of the connectivity array is as follows:

> Nodes 1, 2, 3 are the corners of the first face, given in counterclockwise order when viewed from inside the element. Node 4 is on the opposing vertex. Nodes 5, 6, 7 are on the first face between nodes 1 and 2, 2 and 3, 3 and 1, respectively. Nodes 8, 9, 10 are along the edges between the first face and node 4, between nodes 1 and 4, 2 and 4, 3 and 4, respectively.

Note that in most normal cases, the elements will be generated automatically via a preprocessor (such as Mentat (Mentat)) so that you need not be concerned with the node numbering scheme.

Integration

The element is integrated numerically using four points (Gaussian quadrature). The first plane of such points is closest to the 1, 2, 3 face of the element with the first point closest to the first node of the element (see Figure 3-196).

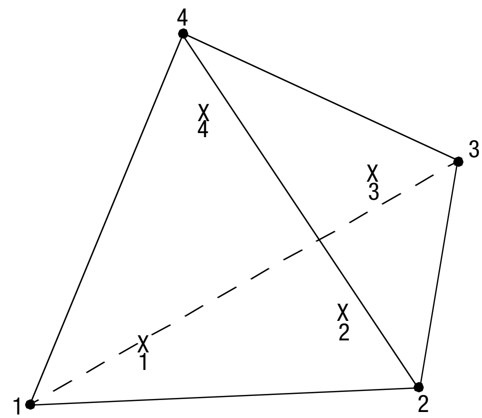


Figure 3-196 Element 127 Integration Plane

Quick Reference

Type 127

Ten nodes, isoparametric arbitrary distorted tetrahedron.

ConnectivitY

Ten nodes numbered as described in the connectivity write-up for this element and as shown in Figure 3-195.

Geometry

Not required.

Coordinates

Three global coordinates in the x-, y-, and z-directions.

Degrees of Freedom

Three global degrees of freedom, u, v, and w.

Distributed Loads

Distributed loads chosen by value of `IBODY` are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3 face.
1	Nonuniform pressure on 1-2-3 face.
2	Uniform pressure on 1-2-4 face.
3	Nonuniform pressure on 1-2-4 face.
4	Uniform pressure on 2-3-4 face.

Load Type	Description
5	Nonuniform pressure on 2-3-4 face.
6	Uniform pressure on 1-3-4 face.
7	Nonuniform pressure on 1-3-4 face.
8	Uniform body force per unit volume in x-direction.
9	Nonuniform body force per unit volume in x-direction.
10	Uniform body force per unit volume in y-direction.
11	Nonuniform body force per unit volume in y-direction.
12	Uniform body force per unit volume in z-direction.
13	Nonuniform body force per unit volume in z-direction.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

The FORCEM user subroutine is called once per integration point when flagged. The magnitude of load defined by DIST LOADS is ignored and the FORCEM value is used instead.

For nonuniform body force, force values must be provided for the four integration points.

For nonuniform surface pressure, force values need only be supplied for the three integration points on the face of application.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

$1 = \varepsilon_{xx}$

$2 = \varepsilon_{yy}$

$3 = \varepsilon_{zz}$

$4 = \varepsilon_{xy}$

$5 = \varepsilon_{yz}$

$6 = \varepsilon_{zx}$

Output of Stresses

Same as for Output of Strains .

Transformation

Three global degrees of freedom can be transformed to local degrees of freedom.

Tying

Use the UFORMSN user subroutine.

Output Points

Centroid or four Gaussian integration points (see Figure 3-196).

You should invoke the appropriate OPTIMIZE option in order to minimize the matrix solution time.

Note:	A large bandwidth results in a lengthy central processing time.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinate directions.

Note:	Distortion of element during analysis can cause bad solutions. Element type 7 is preferred.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 133. See Element 133 for a description of the conventions used or entering the flux and film data for this element. Volumetric flux due to dissipation of plastic work specified with type 101.

Element 128

Plane Strain, Six-node Distorted Triangle, Herrmann Formulation

This is a second-order isoparametric triangular element written for incompressible plane strain applications. This element uses biquadratic interpolation functions to represent coordinates and displacements. This allows for an accurate representation of the strain field. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is represented using bilinear interpolation functions based upon the extra degree of freedom at the corner nodes.

The stiffness of this element is formed using three integration points.

This element is designed to be used for incompressible materials. It can be used for either small strain behavior or large strain behavior.

The connectivity ordering is shown in Figure 3-197. Note that the basic numbering is counterclockwise in the (x-y) plane.

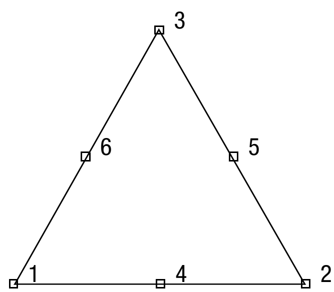


Figure 3-197 Nodes of Six-node, 2-D Element

Quick Reference

Type 128

Second-order isoparametric distorted triangle. Plane strain. Hybrid formulation for incompressible and nearly incompressible materials.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention). Then the fourth node between first and second; the fifth node between second and third, etc. See Figure 3-197.

Geometry

Thickness stored in first data field (`EGEOM1`). Default thickness is unity. Other fields are not used.

Coordinates

Two global coordinates, x and y, at each node.

Degrees of Freedom

At each corner node:

1 = u = global x-direction displacement
2 = v = global y-direction displacement

At corner nodes only,

3 = σ_{kk}/E	= mean pressure variable (isotropic)
= formula below (orthotropic)	
$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1 + \frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2 + \frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3 \right] \frac{9}{E_1 + E_2 + E_3}$$	
= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)
= p / K	(for Herrmann elements using additive decomposition; K here is the effective bulk modulus)

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-4-2 face.
1	Nonuniform pressure on 1-4-2 face.
2	Uniform shear on 1-4-2 face.
3	Nonuniform shear on 1-4-2 face.
4	Uniform pressure on 2-5-3 face.
5	Nonuniform pressure on 2-5-3 face.
6	Uniform shear on 2-5-3 face.
7	Nonuniform shear on 2-5-3 face.
8	Uniform pressure on 3-6-1 face.
9	Nonuniform pressure on 3-6-1 face.
10	Uniform shear on 3-6-1 face.
11	Nonuniform shear on 3-6-1 face.
12	Uniform body force in x-direction.
13	Nonuniform body force in x-direction.
14	Uniform body force in y-direction.
15	Nonuniform body force in y direction.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Load Type (IBODY)	Description
102	Gravity loading in global direction. Enter two magnitudes of gravity acceleration in respectively global x, y direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid or element integration points (see Figure 3-198 and Output Points) in the following order:

1	$= \varepsilon_{xx}$	direct
2	$= \varepsilon_{yy}$	direct
3	$= \varepsilon_{zz}$	thickness direction, direct
4	$= \gamma_{xy}$	shear
5	$= \sigma_{kk}/E$	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	= -p	= negative pressure (for Mooney, Ogden, or Soil)

$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1+\frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2+\frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3\right]\frac{9}{E_1+E_2+E_3}$$

The last component is not printed in the output file, but is accessible via the ELEVAR and PLOTV user subroutines.

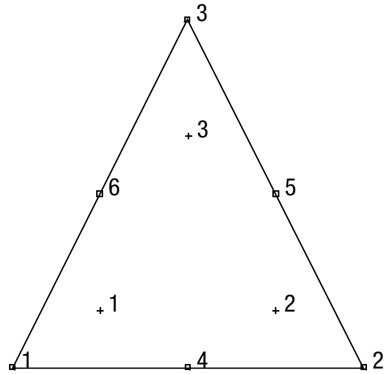


Figure 3-198 Integration Points of Six-node, 2-D Element

Output of Stresses

Output of stresses is the same as Output of Strains .

Transformation

Only in x-y plane.

Tying

Use the UFORMSN user subroutine.

Output Points

If the CENTROID parameter is used, the output occurs at the centroid of the element.

If the ALL POINTS parameter is used, three output points are given, as shown in Figure 3-198. This is the usual option for a second-order element.

Special Considerations

The mean pressure degree of freedom must be allowed to be discontinuous across changes in material properties. Use tying for this case.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available – finite elastic strains can be calculated with the MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 131. See Element 131 for a description of the conventions used for entering the flux and film data for this element.

Element 129 Axisymmetric, Six-node Distorted Triangle, Herrmann Formulation

This is a second-order isoparametric triangular element written for incompressible axisymmetric applications. This element uses biquadratic interpolation functions to represent coordinates and displacements. This allows for an accurate representation of the strain field. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is represented using bilinear interpolation functions based upon the extra degree of freedom at the corner nodes.

The stiffness of this element is formed using three integration points.

This element is designed to be used for incompressible elasticity only. It can be used for either small strain behavior or large strain behavior using the Mooney or Ogden models. This element can be used in conjunction with other elements such as type 126 when other material behavior, such as plasticity, must be represented.

The connectivity ordering is shown in Figure 3-199. Note that the basic numbering is counterclockwise in the (z-r) plane.

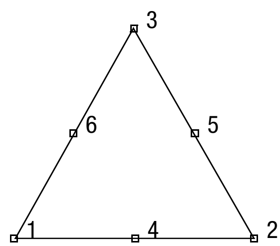


Figure 3-199 Nodes of Six-node, 2-D Element

Quick Reference

Type 129

Second order, isoparametric, distorted triangle. Axisymmetric. Hybrid formulation for incompressible and nearly incompressible elastic materials.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention). Then the fourth node between first and second; the fifth node between second and third, etc. See Figure 3-199.

Geometry

Not required.

Coordinates

Two global coordinates, x and y, at each node.

Degrees of Freedom

At each corner node:

1 = u = global z-direction displacement
2 = v = global r-direction displacement

At corner nodes only:

3 = σ_{kk}/E = mean pressure variable (isotropic)

= formula below (orthotropic)

$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}} \sigma_1 + \frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}} \sigma_2 + \frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}} \sigma_3 \right] \frac{9}{E_1 + E_2 + E_3}$$

= -p = negative hydrostatic pressure (for Mooney, Ogden, or Soil)

Tractions

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-4-2 face.
1	Nonuniform pressure on 1-4-2 face.
2	Uniform shear on 1-4-2 face.
3	Nonuniform shear on 1-4-2 face.
4	Uniform pressure on 2-5-3 face.
5	Nonuniform pressure on 2-5-3 face.
6	Uniform shear on 2-5-3 face.
7	Nonuniform shear on 2-5-3 face.
8	Uniform pressure on 3-6-1 face.
9	Nonuniform pressure on 3-6-1 face.
10	Uniform shear on 3-6-1 face.
11	Nonuniform shear on 3-6-1 face.
12	Uniform body force in x-direction.
13	Nonuniform body force in x-direction.
14	Uniform body force in y-direction.
15	Nonuniform body force in y direction.

Load Type (IBODY)	Description
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter two magnitudes of gravity acceleration in respectively global x, r direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid or element integration points (see Figure 3-200 and Output Points on the following page) in the following order:

1	$= \varepsilon_{zz}$	direct
2	$= \varepsilon_{rr}$	direct
3	$= \varepsilon_{\theta\theta}$	hoop direction, direct
4	$= \gamma_{xy}$	shear
5	$= \sigma_{kk}/E$	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	= -p	= negative pressure (for Mooney, Ogden, or Soil)

$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1+\frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2+\frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3\right]\frac{9}{E_1+E_2+E_3}$$

The last component is not printed in the output file, but is accessible via the ELEVAR and PLOTV user subroutines.

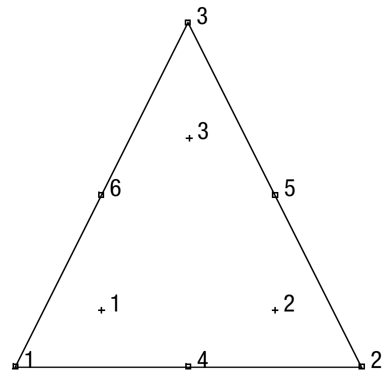


Figure 3-200 Integration Points of Six-node, 2-D Element

Output Of Stresses

Output of stresses is the same as Output of Strains .

Transformation

Only in z-r plane.

Tying

Use the UFORMSN user subroutine.

Output Points

If the CENTROID parameter is used, the output occurs at the centroid of the element.

If the ALL POINTS parameter is used, three output points are given, as shown in Figure 3-200. This is the usual option for a second-order element.

Special Considerations

The mean pressure degree of freedom must be allowed to be discontinuous across changes in material properties. Use tying for this case.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available – finite elastic strains can be calculated with the MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 132. See Element 132 for a description of the conventions used for entering the flux and film data for this element.

Element 130

Three-dimensional Ten-node Tetrahedron, Herrmann Formulation

This is a second-order isoparametric tetrahedron element written for incompressible three-dimensional applications. This element uses biquadratic interpolation functions to represent coordinates and displacements. This allows for an accurate representation of the strain field. The displacement formulation has been modified using the Herrmann variational principle. The pressure field is represented using bilinear interpolation functions based upon the extra degree of freedom at the corner nodes.

The stiffness of this element is formed using four integration points. The mass matrix of this element is formed using sixteen-point Gaussian integration.

This element is designed to be used for incompressible elasticity only. It can be used for either small strain behavior or large strain behavior using the Mooney or Ogden models. This element can be used in conjunction with other elements such as type 127 when other material behavior, such as plasticity, must be represented.

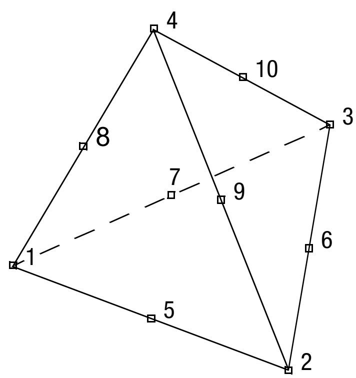


Figure 3-201 Form of Element 130

Geometry

The geometry of the element is interpolated from the Cartesian coordinates of ten nodes.

Connectivity

The convention for the ordering of the connectivity array is as follows:

> Nodes 1, 2, 3 are the corners of the first face, given in counter clockwise order when viewed from inside the element. Node 4 is on the opposing vertex. Nodes 5, 6, 7 are on the first face between nodes 1 and 2, 2 and 3, 3 and 1, respectively. Nodes 8, 9, 10 are along the edges between the first face and node 4, between nodes 1 and 4, 2 and 4, 3 and 4, respectively.

Note that in most normal cases, the elements are generated automatically via a preprocessor (such as Mentat) so that you need not be concerned with the node numbering scheme.

Integration

The element is integrated numerically using four points (Gaussian quadrature). The first plane of such points is closest to the 1-2-3 face of the element, with the first point closest to the first node of the element (see Figure 3-202).

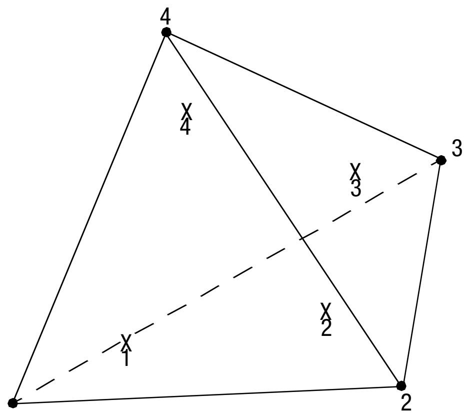


Figure 3-202 Element 130 Integration Plane

Quick Reference

Type 130

Ten-nodes, isoparametric arbitrary distorted tetrahedron.

Hybrid formulation for incompressible and nearly incompressible elastic materials.

Connectivity

Ten nodes numbered as described in the connectivity write-up for this element and as shown in Figure 3-201.

Geometry

Not required.

Coordinates

Three global coordinates in the x-, y-, and z-directions.

Degrees of Freedom

At each corner:

1 = u = global x-direction displacement
2 = v = global y-direction displacement
3 = w = global z-direction displacement

At corner nodes only:

4	= σ_{kk}/E	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1+\frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2+\frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3\right]\frac{9}{E_1+E_2+E_3}$$	
	= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)
	= p / K	(for Herrmann elements using additive decomposition; K here is the effective bulk modulus)

Distributed Loads

Distributed loads chosen by value of `IBODY` are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3 face.
1	Nonuniform pressure on 1-2-3 face.
2	Uniform pressure on 1-2-4 face.
3	Nonuniform pressure on 1-2-4 face.
4	Uniform pressure on 2-3-4 face.
5	Nonuniform pressure on 2-3-4 face.
6	Uniform pressure on 1-3-4 face.
7	Nonuniform pressure on 1-3-4 face.
8	Uniform body force per unit volume in x direction.
9	Nonuniform body force per unit volume in x direction.
10	Uniform body force per unit volume in y direction.
11	Nonuniform body force per unit volume in y direction.
12	Uniform body force per unit volume in z direction.
13	Nonuniform body force per unit volume in z direction.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Load Type	Description
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

The FORCEM user subroutine is called once per integration point when flagged. The magnitude of load defined by DIST LOADS is ignored and the FORCEM value is used instead.

For nonuniform body force, force values must be provided for the four integration points.

For nonuniform surface pressure, force values need only be supplied for the three integration points on the face of application.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

1	$= \varepsilon_{xx}$	
2	$= \varepsilon_{yy}$	
3	$= \varepsilon_{zz}$	
4	$= \gamma_{xy}$	
5	$= \gamma_{yz}$	
6	$= \gamma_{zx}$	
7	$= \sigma_{kk}/E$	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	$= -p$	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)

$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1 + \frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2 + \frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3 \right] \frac{9}{E_1 + E_2 + E_3}$$

The last component is not printed in the output file, but is accessible via the ELEVAR and PLOTV user subroutines.

Output of Stresses

Same as for Output of Strains .

Transformation

Three global degrees of freedom can be transformed to local degrees of freedom.

Tying

Use the UFORMSN user subroutine.

Output Points

Centroid or four Gaussian integration points (see Figure 3-202).

Notes:	A large bandwidth results in a lengthy central processing time.
	You should invoke the appropriate OPTIMIZE option in order to minimize the matrix solution time.

Special Considerations

The mean pressure degree of freedom must be allowed to be discontinuous cross changes in material properties. Use tying for this case.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available – finite elastic strains can be calculated with the MOONEY or OGDEN option.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 133. See Element 133 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux due to dissipation of inelastic work specified with type 101.

Element 131

Planar, Six-node Distorted Triangle (Heat Transfer Element)

This is a second-order isoparametric two-dimensional heat transfer triangular element. Temperatures and position (coordinates) within the element are interpolated from six sets of nodal values, the three corners and the three midsides (see Figure 3-203). The interpolation function is such that each edge has parabolic variation along itself. The second-order elements are usually preferred to a more refined mesh of first-order elements for most problems. The conductivity of this element is formed using three-point integration.

The connectivity ordering is shown in Figure 3-203. Note that the basic numbering is counterclockwise in the (x-y) plane.

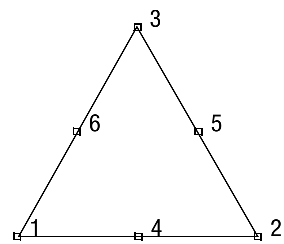


Figure 3-203 Nodes of Six-node, 2-D Element

Quick Reference

Type 131

Second-order isoparametric distorted heat transfer triangle.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention). Then the fourth node between first and second; the fifth node between second and third, etc. See Figure 3-203.

Geometry

Thickness stored in first data field (`EGEOM1`). Default thickness is unity. Other fields are not used.

Coordinates

Two global coordinates, x and y, at each node.

Degrees of Freedom

Two at each node:

1 = temperature
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatics)

Fluxes

Surface fluxes are available for this element as follows:

Load Type (IBODY)	Description
0	Uniform flux on 1-4-2 face.
1	Nonuniform flux on 1-4-2 face.
4	Uniform flux on 2-5-3 face.
5	Nonuniform flux on 2-5-3 face.
8	Uniform flux on 3-6-1 face.
9	Nonuniform flux on 3-6-1 face.
12	Uniform body flux per unit volume.
13	Nonuniform body flux per unit volume.

For all nonuniform fluxes, the load magnitude is supplied via the FLUX user subroutine.

Films

Same specification as Fluxes .

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Current

Same specifications as Fluxes .

Charge

Same specifications as Fluxes .

Output Points

If the CENTROID parameter is used, the output occurs at the centroid of the element.

If the ALL POINTS parameter is used, three output points are given, as shown in Figure 3-204. This is the usual option for a second-order element.

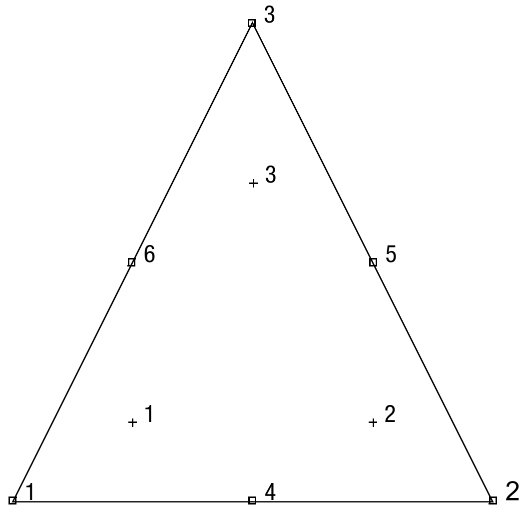


Figure 3-204 Integration Points of Six-node, 2-D Element

Element 132 Axisymmetric, Six-node Distorted Triangle (Heat Transfer Element)

This is a second-order isoparametric two-dimensional axisymmetric triangular element. Temperatures and position (coordinates) within the heat transfer element are interpolated from six sets of nodal values, the three corners and the three midsides (see Figure 3-205). The interpolation function is such that each edge has parabolic variation along itself. The second-order elements are usually preferred to a more refined mesh of first-order elements for most problems. The conductivity of this element is formed using three-point integration.

The connectivity ordering is shown in Figure 3-205. Note that the basic numbering is counterclockwise in the (z-r) plane.

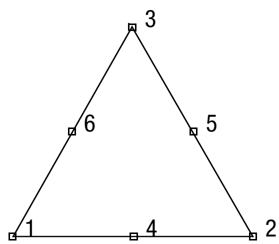


Figure 3-205 Nodes of Six-node, Axisymmetric Element

Quick Reference

Type 132

Second-order isoparametric distorted axisymmetric heat transfer triangle.

Connectivity

Corners numbered first, in counterclockwise order (right-handed convention). Then the fourth node between first and second; the fifth node between second and third, etc. See Figure 3-205.

Geometry

Not required.

Coordinates

Two global coordinates, z and r, at each node.

Degrees of Freedom

Two at each node:

1 = temperature
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatics)

Fluxes

Surface fluxes are available for this element as follows:

Load Type (IBODY)	Description
0	Uniform flux on 1-4-2 face.
1	Nonuniform flux on 1-4-2 face.
4	Uniform flux on 2-5-3 face.
5	Nonuniform flux on 2-5-3 face.
8	Uniform flux on 3-6-1 face.
9	Nonuniform flux on 3-6-1 face.
12	Uniform body flux per unit volume.
13	Nonuniform body flux per unit volume.

For all nonuniform loads, the load magnitude is supplied via the FLUX user subroutine.

Films

Same specification as Fluxes .

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Current

Same specification as Fluxes .

Charge

Same specification as Fluxes .

Output Points

If the CENTROID parameter is used, the output occurs at the centroid of the element.

If the ALL POINTS parameter is used, three output points are given, as shown in Figure 3-206. This is the usual option for a second-order element.

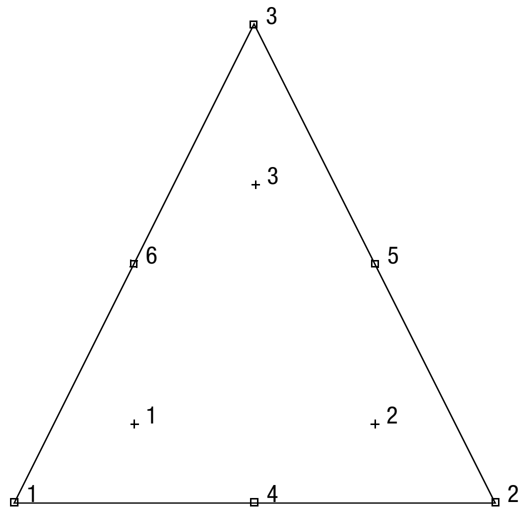


Figure 3-206 Integration Points of Six-node, Axisymmetric Element

Element 133

Three-dimensional Ten-node Tetrahedron (Heat Transfer Element)

This element is a second-order isoparametric three-dimensional heat transfer tetrahedron. Each edge forms a parabola, so that four nodes define the corners of the element and a further six nodes define the position of the "midpoint" of each edge (see Figure 3-207). The second-order elements are usually preferred to a more refined mesh of first-order elements for most problems. The conductivity of this element is formed using four-point integration. The specific heat capacity matrix of this element is formed using sixteen-point Gaussian integration.

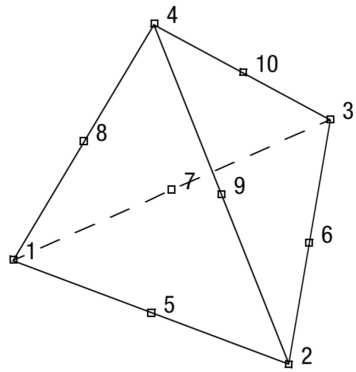


Figure 3-207 Form of Element 133

Geometry

The geometry of the element is interpolated from the Cartesian coordinates of ten nodes.

Connectivity

The convention for the ordering of the connectivity array is as follows:

> Nodes 1, 2, 3 are the corners of the first face, given in counterclockwise order when viewed from inside the element. Node 4 is on the opposing vertex. Nodes 5, 6, 7 are on the first face between nodes 1 and 2, 2 and 3, 3 and 1, respectively. Nodes 8, 9, 10 are along the edges between the first face and node 4, between nodes 1 and 4, 2 and 4, 3 and 4, respectively.

Note that in most normal cases, the elements are generated automatically via a preprocessor (such as Mentat), so that you need not be concerned with the node numbering scheme.

Integration

The conductivity matrix of the element is integrated numerically using four points (Gaussian quadrature). The first plane of such points is closest to the 1, 2, 3 face of the element, with the first point closest to the first node of the element (see Figure 3-208). Similarly, the heat capacitance matrix of the element is integrated numerically using sixteen integration points. The GEOMETRY option can be used to switch to a reduced (four point) integration scheme for the heat capacitance matrix.

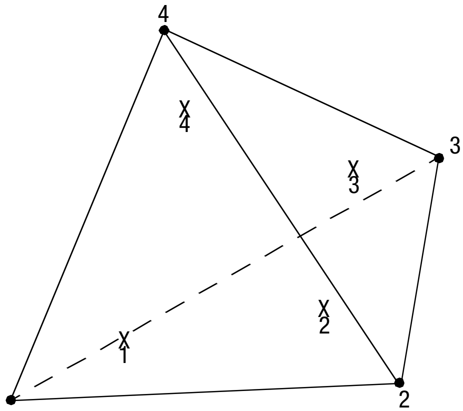


Figure 3-208 1-2 Element 133 Integration Plane

Quick Reference

Type 133

Ten-nodes, isoparametric arbitrary distorted heat transfer tetrahedron.

Connectivity

Ten nodes numbered as described in the connectivity write-up for this element and as shown in Figure 3-207.

Geometry

Not required. Optionally, the `EGEOM5` field can be set to one to use a reduced integration scheme for the heat capacitance matrix.

Coordinates

Three global coordinates in the x-, y-, and z-directions.

Degrees of Freedom

1 = temperature
1 = voltage, temperature (Joule Heating)
1 = potential, (electrostatic)

Distributed Fluxes

Distributed fluxes chosen by value of `IBODY` are as follows:

Load Type	Description
0	Uniform flux on 1-2-3 face.
1	Nonuniform flux on 1-2-3 face.
2	Uniform flux on 1-2-4 face.

Load Type	Description
3	Nonuniform flux on 1-2-4 face.
4	Uniform flux on 2-3-4 face.
5	Nonuniform flux on 2-3-4 face.
6	Uniform flux on 1-3-4 face.
7	Nonuniform flux on 1-3-4 face.
8	Uniform body flux per unit volume.
9	Nonuniform body flux per unit volume.

The FLUX user subroutine is called once per integration point when flagged. The magnitude of load defined by DIST FLUXES is ignored and the FLUX value is used instead.

For nonuniform body flux, flux values must be provided for the four integration points.

For nonuniform surface pressure, flux values need only be supplied for the three integration points on the face of application.

Films

Same specification as Distributed Fluxes .

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Current

Same specification as Distributed Fluxes .

Charges

Same specifications as Distributed Fluxes .

Output Points

Centroid or four Gaussian integration points (see Figure 3-208).

Notes:	A large bandwidth results in a lengthy central processing time.
	You should invoke the appropriate OPTIMIZE option in order to minimize the matrix solution time.

Element 134

Three-dimensional Four-node Tetrahedron

This element is a linear isoparametric three-dimensional tetrahedron (see Figure 3-209). As this element uses linear interpolation functions, the strains are constant throughout the element. The element is integrated numerically using one point at the centroid of the element. This results in a poor representation of shear behavior. A fine mesh is required to obtain an accurate solution. This element should only be used for linear elasticity. The higher-order element 127 is more accurate, especially for nonlinear problems.

This element gives poor results for incompressible materials. This includes rubber elasticity, large strain plasticity, and creep, For such problems, use element type 157.

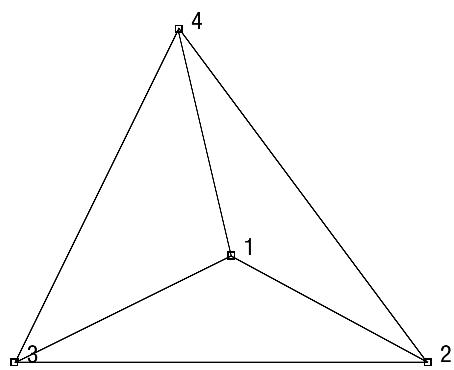


Figure 3-209 Form of Element 134

Geometry

The geometry of the element is interpolated from the Cartesian coordinates of four nodes.

Connectivity

The convention for the ordering of the connectivity array is as follows:

> Nodes 1, 2, 3 are the corners of the first face, given in counterclockwise order when viewed from inside the element. Node 4 is on the opposing vertex. Note that in most normal cases, the elements are generated automatically via a preprocessor (such as Mentat or a CAD program) so that you need not be concerned with the node numbering scheme.

Quick Reference

Type 134

Four-nodes, isoparametric arbitrary distorted tetrahedron.

Connectivity

Four nodes numbered as described in the connectivity write-up for this element and as shown in Figure 3-209.

Geometry

Not required.

Coordinates

Three global coordinates in the x-, y-, and z-directions.

Degrees of Freedom

Three global degrees of freedom, u, v, and w.

Distributed Loads

Distributed loads chosen by value of IBODY are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3 face.
1	Nonuniform pressure on 1-2-3 face.
2	Uniform pressure on 1-2-4 face.
3	Nonuniform pressure on 1-2-4 face.
4	Uniform pressure on 2-3-4 face.
5	Nonuniform pressure on 2-3-4 face.
6	Uniform pressure on 1-3-4 face.
7	Nonuniform pressure on 1-3-4 face.
8	Uniform body force per unit volume in x direction.
9	Nonuniform body force per unit volume in x direction.
10	Uniform body force per unit volume in y direction.
11	Nonuniform body force per unit volume in y direction.
12	Uniform body force per unit volume in z direction.
13	Nonuniform body force per unit volume in z direction.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Load Type	Description
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

The FORCEM user subroutine is called once per integration point when flagged. The magnitude of load defined by DIST LOADS is ignored and the FORCEM value is used instead.

For nonuniform body force, force values must be provided for the one integration point.

For nonuniform surface pressure, force values need only be supplied for the one integration point on the face of application.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

$1 = \varepsilon_{xx}$

$2 = \varepsilon_{yy}$

$3 = \varepsilon_{zz}$

$4 = \gamma_{xy}$

$5 = \gamma_{yz}$

$6 = \gamma_{zx}$

Output of Stresses

Same as for Output of Strains .

Transformation

Three global degrees of freedom can be transformed to local degrees of freedom.

Tying

Use the UFORMSN user subroutine.

Output Points

Centroid.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinate directions.

Notes:	A large bandwidth results in a lengthy central processing time.
	You should invoke the appropriate OPTIMIZE option in order to minimize the matrix solution time.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 135. See Element 135 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux due to dissipation of plastic work specified with type 101.

Element 135

Three-dimensional Four-node Tetrahedron (Heat Transfer Element)

This element is a linear isoparametric three-dimensional tetrahedron for heat transfer applications (see Figure 3-210). As this element uses linear interpolation functions, the thermal gradients are constant throughout the element. A fine mesh is required to obtain an accurate solution. The element is integrated numerically using one point at the centroid of the element. The specific heat capacity matrix of this element is formed using four-point Gaussian integration.

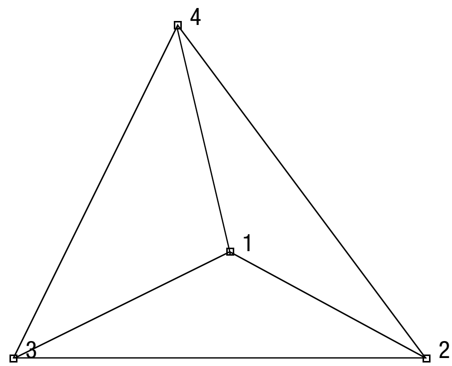


Figure 3-210 Form of Element 135

Geometry

The geometry of the element is interpolated from the Cartesian coordinates of four nodes.

Connectivity

The convention for the ordering of the connectivity array is as follows:

> Nodes 1, 2, 3 are the corners of the first face, given in counterclockwise order when viewed from inside the element. Node 4 is on the opposing vertex. Note that in most normal cases, the elements are generated automatically via a preprocessor (such as Mentat or a CAD program) so that you need not be concerned with the node numbering scheme.

Quick Reference

Type 135

Four-nodes, isoparametric arbitrary heat transfer tetrahedron.

Connectivity

Four nodes numbered as described in the connectivity write-up for this element and as shown in Figure 3-210.

Geometry

Not required.

Coordinates

Three global coordinates in the x-, y-, and z-directions.

Degrees of Freedom

1 = temperature
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)

Distributed Fluxes

Distributed fluxes chosen by value of `IBODY` are as follows:

Load Type	Description
0	Uniform flux on 1-2-3 face.
1	Nonuniform flux on 1-2-3 face.
2	Uniform flux on 1-2-4 face.
3	Nonuniform flux on 1-2-4 face.
4	Uniform flux on 2-3-4 face.
5	Nonuniform flux on 2-3-4 face.
6	Uniform flux on 1-3-4 face.
7	Nonuniform flux on 1-3-4 face.
8	Uniform body flux per unit volume.
9	Nonuniform body flux per unit volume.

The FLUX user subroutine is called once per integration point when flagged. The magnitude of load defined by DIST FLUXES is ignored and the FLUX value is used instead.

For nonuniform body flux, flux values must be provided for the one integration point.

For nonuniform surface fluxes, flux values need to be supplied for the 3-integration points on the face of application, where the integration points have the same location as the nodal points.

Films

Same specification as Distributed Fluxes .

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Current

Same specification as Distributed Fluxes .

Charges

Same specification as Distributed Fluxes .

Output Points

Centroid.

Notes:	A large bandwidth results in a lengthy central processing time.
	You should invoke the appropriate OPTIMIZE option in order to minimize the matrix solution time.

Element 136 — Three-dimensional Arbitrarily Distorted Pentahedral

Element type 136 is a six-node, isoparametric, arbitrary pentahedral. As this element uses trilinear interpolation functions, the strains tend to be constant throughout the element. This results in a poor representation of shear behavior.

The stiffness of this element is formed using six-point Gaussian integration.

For nearly incompressible behavior, including plasticity or creep, it is advantageous to use an alternative integration procedure. This constant dilatation method which eliminates potential element locking is flagged through the GEOMETRY option.

This element can be used for all constitutive relations. When using incompressible rubber materials (for example, Mooney and Ogden), the element must be used within the Updated Lagrange framework.

Quick Reference

Type 136

Three-dimensional, six-node, first-order, isoparametric element (arbitrarily distorted pentahedral).

Connectivity

Six nodes per element. Node numbering must follow the scheme below (see Figure 3-211):

> Nodes 1, 2, and 3 are corners of one face, given in counterclockwise order when viewed from inside the element. Node 4 has the same edge as node 1. Node 5 has the same edge as node 2. Node 6 has the same edge as node 3.

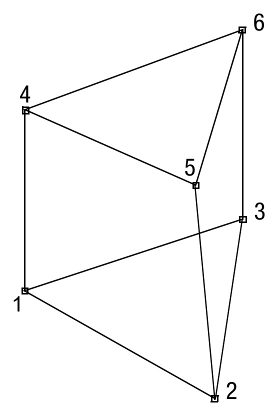


Figure 3-211 Arbitrarily Distorted Pentahedral

Geometry

If a nonzero value is entered in the second data field (`EGEOM2`), the volumetric strain is constant throughout the element. This is particularly useful for analysis of approximately incompressible materials, and for analysis of structures in the fully plastic range. It is also recommended for creep problems in which it is attempted to obtain the steady state solution.

Coordinates

Three coordinates in the global x-, y-, and z-directions.

Degrees of Freedom

Three global degrees of freedom u, v, and w per node.

Distributed Loads

Distributed loads chosen by value of `IBODY` are as follows:

Load Type	Description
1	Uniform pressure on 1-2-5-4 face.
2	Nonuniform pressure on 1-2-5-4 face; magnitude and direction supplied through the FORCEM user subroutine.
3	Uniform pressure on 2-3-6-5 face.
4	Nonuniform pressure on 2-3-6-5 face; magnitude and direction supplied through the FORCEM user subroutine.
5	Uniform pressure on 3-1-4-6 face.
6	Nonuniform pressure on 3-1-4-6 face; magnitude and direction supplied through the FORCEM user subroutine.
7	Uniform pressure on 1-3-2 face.
8	Nonuniform pressure on 1-3-2 face; magnitude and direction supplied through the FORCEM user subroutine.
9	Uniform pressure on 4-5-6 face.
10	Nonuniform pressure on 4-5-6 face; magnitude and direction supplied through the FORCEM user subroutine.
11	Uniform volumetric load in x-direction.
12	Nonuniform volumetric load in x-direction; magnitude and direction supplied through the FORCEM user subroutine.
13	Uniform volumetric load in y-direction.
14	Nonuniform volumetric load in y-direction; magnitude and direction supplied through the FORCEM user subroutine.
15	Uniform volumetric load in z-direction.
16	Nonuniform volumetric load in z-direction; magnitude and direction supplied through the FORCEM user subroutine.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Load Type	Description
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Pressure forces are positive into element face.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

1 = ε_{xx}
2 = ε_{yy}
3 = ε_{zz}
4 = γ_{xy}
5 = γ_{yz}
6 = γ_{zx}

Output of Stresses

Output of stresses is the same as for Output of Strains .

Transformation

Standard transformation of three global degrees of freedom to local degrees of freedom.

Tying

No special tying available.

Output Points

Centroid or the six integration points as shown in Figure 3-212.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available.

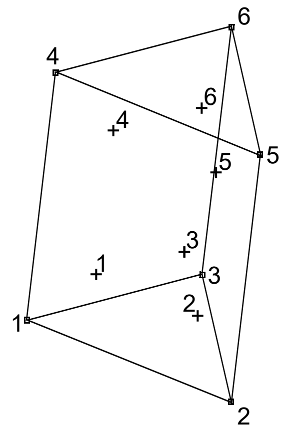


Figure 3-212 Six-Point Gauss Integration Scheme for Element 136

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 137. See Element 137 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux generated by dissipated plastic energy is specified with type 101.

Element 137 — Three-dimensional Six-node Pentahedral (Heat Transfer Element)

Element type 137 is a six-node, isoparametric, arbitrary pentahedral written for three-dimensional heat transfer applications. This element can also be used for electrostatic applications.

As this element uses trilinear interpolation functions, the thermal gradients tend to be constant throughout the element.

The conductivity of this element is formed using six-point Gaussian integration.

Quick Reference

Type 137

Six-node, three-dimensional, first-order isoparametric heat transfer element.

Connectivity

Eight nodes per element.

Nodes 1, 2, and 3 are the corners on one face, numbered in a counterclockwise direction when viewed from inside the element. Nodes 4, 5, and 6 are the nodes on the other face, with node 4 opposite node 1, and so on (see Figure 3-213).

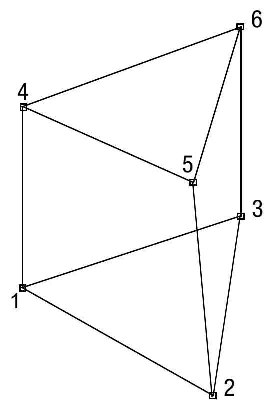


Figure 3-213 Arbitrarily Distorted Heat Transfer Pentahedral

Geometry

If a nonzero value is entered in the fourth data field (EGEOM4), the temperatures at the integration points obtained from interpolation of nodal temperatures are constant throughout the element.

Coordinates

Three coordinates in the global x, y, and z directions.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)

Fluxes

Fluxes are distributed according to the appropriate selection of a value of IBODY. Surface fluxes are assumed positive when directed into the element and are evaluated using a 4-point integration scheme, where the integration points have the same location as the nodal points.

Load Type (IBODY)	Description
1	Uniform flux on 1-2-5-4 face.
2	Nonuniform flux on 1-2-5-4 face; magnitude and direction supplied through the FORCEM user subroutine.
3	Uniform flux on 2-3-6-5 face.
4	Nonuniform flux on 2-3-6-5 face; magnitude and direction supplied through the FORCEM user subroutine.
5	Uniform flux on 3-1-4-6 face.
6	Nonuniform flux on 3-1-4-5 face; magnitude and direction supplied through the FORCEM user subroutine.
7	Uniform flux on 1-3-2 face.
8	Nonuniform flux on 1-3-2 face; magnitude and direction supplied through the FORCEM user subroutine.
9	Uniform flux on 4-5-6 face.
10	Nonuniform flux on 4-5-6 face; magnitude and direction supplied through the FORCEM user subroutine.
11	Uniform volumetric flux.
12	Nonuniform volumetric flux; magnitude and direction supplied through the FORCEM user subroutine.

Films

Same specification as Fluxes .

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Current

Same specification as Fluxes .

Charge

Same specifications as Fluxes .

Output Points

Centroid or six Gaussian integration points (see Figure 3-214).

Note:	As in all three-dimensional analysis, a large nodal bandwidth results in long computing times. Use the optimizers as much as possible.

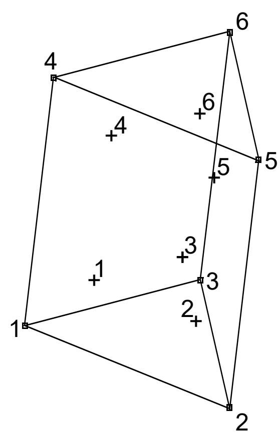


Figure 3-214 Integration Points for Element 137

Element 138

Bilinear Thin-triangular Shell Element

This is a three-node, thin-triangular shell element with global displacements and rotations as degrees of freedom. Bilinear interpolation is used for the coordinates, displacements and the rotations. The membrane strains are obtained from the displacement field; the curvatures from the rotation field. The element can be used in curved shell analysis as well as in the analysis of complicated plate structures. For the latter case, the element is easy to use since connections between intersecting plates can be modeled without tying.

Due to its simple formulation when compared to the standard higher order shell elements, it is less expensive and, therefore, very attractive in nonlinear analysis. The element is not very sensitive to distortion. All constitutive relations can be used with this element.

Geometric Basis

The first two element base vectors lie in the plane of the three corner nodes. The first one (V_1) points from node 1 to node 2; the second one (V_2) is perpendicular to V_1 and lies on the same side as node 3. The third one (V_3) is determined such that V_1, V_2, and V_3 form a right-hand system (Figure 3-215).

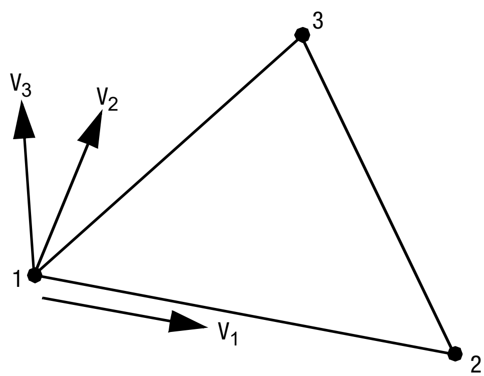


Figure 3-215 Form of Element 138

There are three integration points in the plane of the element (Figure 3-216).

Displacements

The six nodal displacement variables are as follows:

u, v, w	Displacement components defined in global Cartesian x,y,z coordinate system.
ϕ_x, ϕ_y, ϕ_z	Rotation components about global x, y and z axis respectively.

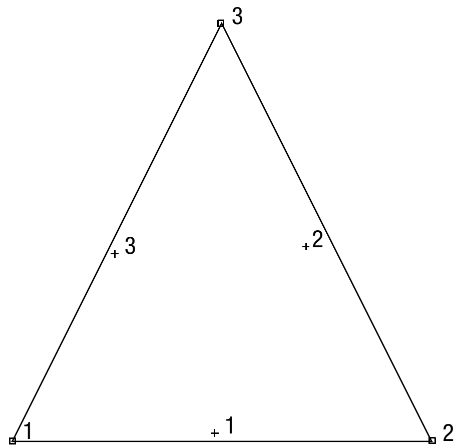


Figure 3-216 Integration Points of Three-node Shell Element

Quick Reference

Type 138

Bilinear, three-node thin shell element.

Connectivity

Three nodes per element.

Geometry

Bilinear thickness variation is allowed in the plane of the element. Thicknesses at first, second, and third nodes of the element are stored for each element in the first (`EGEOM1`), second (`EGEOM2`), and third (`EGEOM3`), geometry data fields, respectively. If `EGEOM2 = EGEOM3 = 0`, then a constant thickness (`EGEOM1`) is assumed for the element.

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

If the element needs to be offset from the user-specified position, then the eighth data field (`EGEOM8`) is set to -2.0. In this case, an extra line containing the offset information is provided (see the GEOMETRY option in Marc Volume C: Program Input). The offset magnitudes along the element normal for the three corner nodes are provided in the first, second and third data fields of the extra line. A uniform offset for all nodes can be set by providing the offset magnitude in the first data field and then setting the constant offset flag (sixth data field of the extra line) to 1.

Coordinates

Three coordinates per node in the global x-, y-, and z-directions.

Degrees of Freedom

Six degrees of freedom per node:

1 = u = global (Cartesian) x-displacement
2 = v = global (Cartesian) y-displacement
3 = w = global (Cartesian) z-displacement
4 = ϕ_x = rotation about global x-axis
5 = ϕ_y = rotation about global y-axis
6 = ϕ_z = rotation about global z-axis

Distributed Loads

A table of distributed loads is listed below:

Load Type	Description
1	Uniform gravity load per surface area in -z-direction.
2	Uniform pressure with positive magnitude in -V_3-direction.
3	Nonuniform gravity load per surface area in -z-direction.
4	Nonuniform pressure with positive magnitude in -V_3-direction, magnitude is given in the FORCEM user subroutine.
5	Nonuniform load per surface area in arbitrary direction, magnitude is given in the FORCEM user subroutine.
11	Uniform edge load in the plane of the basic triangle on the 1-2 edge and perpendicular to the edge.
12	Uniform edge load in the plane of the basic triangle on the 1-2 edge and tangential to the edge.
13	Nonuniform edge load in the plane of the basic triangle on the 1-2 edge and perpendicular to the edge; magnitude is given in the FORCEM user subroutine.
14	Nonuniform edge load in the plane of the basic triangle on the 1-2 edge and tangential to the edge; magnitude is given in the FORCEM user subroutine.
15	Nonuniform edge load in the plane of the basic triangle on the 1-2 edge; magnitude and direction is given in the FORCEM user subroutine.
16	Uniform load on the 1-2 edge of the shell. Perpendicular to the shell; i.e., -v_3 direction
21	Uniform edge load in the plane of the basic triangle on the 2-3 edge and perpendicular to the edge.
22	Uniform edge load in the plane of the basic triangle on the 2-3 edge and tangential to the edge.
23	Nonuniform edge load in the plane of the basic triangle on the 2-3 edge and perpendicular to the edge; magnitude is given in the FORCEM user subroutine.
24	Nonuniform edge load in the plane of the basic triangle on the 2-3 edge and tangential to the edge; magnitude is given in the FORCEM user subroutine.
25	Nonuniform edge load in the plane of the basic triangle on the 2-3 edge; magnitude and direction is given in the FORCEM user subroutine.
26	Uniform load on the 2-3 edge of the shell. Perpendicular to the shell; i.e., -v_3 direction

Load Type	Description
31	Uniform edge load in the plane of the basic triangle on the 3-1 edge and perpendicular to the edge.
32	Uniform edge load in the plane of the basic triangle on the 3-1 edge and tangential to the edge.
33	Nonuniform edge load in the plane of the basic triangle on the 3-1 edge and perpendicular to the edge; magnitude is given in the FORCEM user subroutine.
34	Nonuniform edge load in the plane of the basic triangle on the 3-1 edge and tangential to the edge; magnitude is given in the FORCEM user subroutine.
35	Nonuniform edge load in the plane of the basic triangle on the 3-1 edge; magnitude and direction is given in the FORCEM user subroutine.
36	Uniform load on the 3-1 edge of the shell. Perpendicular to the shell; i.e., $-v_3$ direction
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity load in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.

All edge loads require the input as force per unit length.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Point Loads

Point loads and moments can also be applied at the nodes.

Output Of Strains

Generalized strain components are:

Middle surface stretches:	ε_{11} ε_{22} ε_{12}
Middle surface curvatures:	κ_{11} κ_{22} κ_{12}

in local $(\underline{V}_1, \underline{V}_2, \underline{V}_3)$ system.

Output Of Stresses

$\sigma_{11}, \sigma_{22}, \sigma_{12}$ in local $(\underline{V}_1, \underline{V}_2, \underline{V}_3)$ system given at equally spaced layers though thickness. First layer is on positive V_3 direction surface.

Transformation

Displacement and rotation at corner nodes can be transformed to local direction.

Tying

Use the UFORMSN user subroutine.

Updated Lagrange Procedure and Finite Strain Plasticity

Updated Lagrange capability is available. Note, however, that since the curvature calculation is linearized, you have to select your load steps such that the rotation remains small within a load step.

Section Stress - Integration

Integration through the shell thickness is performed numerically using Simpson's rule. Use the SHELL SECT parameter to specify the number of integration points. This number must be odd. Seven points are enough for simple plasticity or creep analysis. Eleven points are enough for complex plasticity or creep (for example, thermal plasticity). The default is 11 points.

Beam Stiffeners

The element is fully compatible with open- and closed-section beam element types 78 and 79.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 50. See Element 50 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered as a design variable.

Element 139

Bilinear Thin-shell Element

This is a four-node, thin-shell element with global displacements and rotations as degrees of freedom. Bilinear interpolation is used for the coordinates, displacements and the rotations. The membrane strains are obtained from the displacement field; the curvatures from the rotation field. The element can be used in curved shell analysis as well as in the analysis of complicated plate structures. For the latter case, the element is easy to use since connections between intersecting plates can be modeled without tying.

Due to its simple formulation when compared to the standard higher order shell elements, it is less expensive and, therefore, very attractive in nonlinear analysis. The element is not very sensitive to distortion. All constitutive relations can be used with this element.

Geometric Basis

The element is defined geometrically by the (x,y,z) coordinates of the four corner nodes. Due to the bilinear interpolation, the surface forms a hyperbolic paraboloid which is allowed to degenerate to a plate. The element thickness is specified in the GEOMETRY option.

The stress output is given in local orthogonal surface directions, V_1, V_2, and V_3, which for the centroid are defined in the following way (see Figure 3-217).

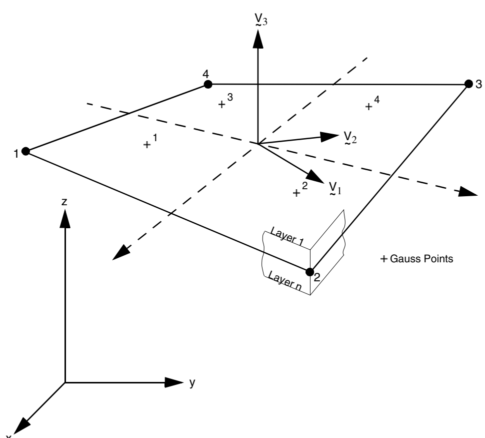


Figure 3-217 Form of Element 139

At the centroid the vectors tangent to the curves with constant isoparametric coordinates are normalized.

$$t_1 = \frac{\partial x}{\partial \xi} \Big/ \left|\frac{\partial x}{\partial \xi}\right|, \quad t_2 = \frac{\partial x}{\partial \eta} \Big/ \left|\frac{\partial x}{\partial \eta}\right|$$

Now a new basis is being defined as:

$$s = t_1 + t_2, \quad d = t_1 - t_2$$

After normalizing these vectors by:

$$\bar{s} = s / \sqrt{2}|s| \quad \bar{d} = d / \sqrt{2}|d|$$

The local orthogonal directions are then obtained as:

$$\underset{\sim}{V}_1 = \bar{s} + \bar{d}, \ \underset{\sim}{V}_2 = \bar{s} - \bar{d} \text{ , and } \underset{\sim}{V}_3 = \underset{\sim}{V}_1 \times \underset{\sim}{V}_2$$

In this way, the vectors $\frac{\partial x}{\partial \xi}$, $\frac{\partial x}{\partial \eta}$ and $\underset{\sim}{V}_1$, $\underset{\sim}{V}_2$ have the same bisecting plane.

The local directions at the Gaussian integration points are found by projection of the centroid directions. Hence, if the element is flat, the directions at the Gauss points are identical to those at the centroid.

Displacements

The six nodal displacement variables are as follows:

u, v, w	Displacement components defined in global Cartesian x,y,z coordinate system.
ϕ_x, ϕ_y, ϕ_z	Rotation components about global x, y and z axis respectively.

Quick Reference

Type 139

Bilinear, four-node thin shell element.

Connectivity

Four nodes per element. The element can be collapsed into a triangle in which case the element is identical to element type 138.

Geometry

Bilinear thickness variation is allowed in the plane of the element. Thicknesses at first, second, third, and fourth nodes of the element are stored for each element in the first (`EGEOM1`), second (`EGEOM2`), third (`EGEOM3`) and fourth (`EGEOM4`), geometry data fields, respectively. If `EGEOM2` = `EGEOM3` = `EGEOM4 = 0`, then a constant thickness (`EGEOM1`) is assumed for the element.

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

If the element needs to be offset from the user-specified position, then the 8th data field (`EGEOM8`) is set to -2.0. In this case, an extra line containing the offset information is provided (see the GEOMETRY option in Marc Volume C: Program Input). The offset magnitudes along the element normal for the four corner nodes are provided in the 1st, 2nd, 3rd and 4th data fields of the extra line. A uniform offset for all nodes can be set by providing the offset magnitude in the 1st data field and then setting the constant offset flag (6th data field of the extra line) to 1.

Coordinates

Three coordinates per node in the global x-, y-, and z-directions.

Degrees of Freedom

Six degrees of freedom per node:

1 = u = global (Cartesian) x-displacement
2 = v = global (Cartesian) y-displacement
3 = w = global (Cartesian) z-displacement
4 = ϕ_x = rotation about global x-axis
5 = ϕ_y = rotation about global y-axis
6 = ϕ_z = rotation about global z-axis

Distributed Loads

A table of distributed loads is listed below:

Load Type	Description
1	Uniform gravity load per surface area in -z-direction.
2	Uniform pressure with positive magnitude in -V_3-direction.
3	Nonuniform gravity load per surface area in -z-direction.
4	Nonuniform pressure with positive magnitude in -V_3-direction, magnitude is given in the FORCEM user subroutine.
5	Nonuniform load per surface area in arbitrary direction, magnitude is given in the FORCEM user subroutine.
11	Uniform edge load in the plane of the surface on the 1-2 edge and perpendicular to this edge.
12	Nonuniform edge load magnitude is given in the FORCEM user subroutine in the plane of the surface on the 1-2 edge.
13	Nonuniform edge load magnitude and direction is given in the FORCEM user subroutine on 1-2 edge.
14	Uniform load on the 1-2 edge of the shell, tangent to, and in the direction of the 1-2 edge
15	Uniform load on the 1-2 edge of the shell. Perpendicular to the shell; i.e., -v_3 direction
21	Uniform edge load in the plane of the surface on the 2-3 edge and perpendicular to this edge.
22	Nonuniform edge load magnitude is given in the FORCEM user subroutine in the plane of the surface on the 2-3 edge.
23	Nonuniform edge load magnitude and direction is given in the FORCEM user subroutine on 2-3 edge.
24	Uniform load on the 2-3 edge of the shell, tangent to and in the direction of the 2-3 edge
25	Uniform load on the 2-3 edge of the shell. Perpendicular to the shell; i.e., -v_3 direction
31	Uniform edge load in the plane of the surface on the 3-4 edge and perpendicular to this edge.
32	Nonuniform edge load magnitude is given in the FORCEM user subroutine in the plane of the surface on the 3-4 edge.
33	Nonuniform edge load magnitude and direction is given in the FORCEM user subroutine on 3-4 edge.
34	Uniform load on the 3-4 edge of the shell, tangent to and in the direction of the 3-4 edge

Load Type	Description
35	Uniform load on the 3-4 edge of the shell. Perpendicular to the shell; i.e., $-v_3$ direction
41	Uniform edge load in the plane of the surface on the 4-1 edge and perpendicular to this edge.
42	Nonuniform edge load magnitude is given in the FORCEM user subroutine in the plane of the surface on the 4-1 edge.
43	Nonuniform edge load magnitude and direction is given in the FORCEM user subroutine on 4-1 edge.
44	Uniform load on the 4-1 edge of the shell, tangent to and in the direction of the 4-1 edge
45	Uniform load on the 4-1 edge of the shell. Perpendicular to the shell; i.e., $-v_3$ direction
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

All edge loads require the input as force per unit length.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Point Loads

Point loads and moments can also be applied at the nodes.

Output Of Strains

Generalized strain components are:

Middle surface stretches:	ε_{11} ε_{22} ε_{12}
Middle surface curvatures:	κ_{11} κ_{22} κ_{12}

in local $(\underset{\sim}{V}_1, \underset{\sim}{V}_2, \underset{\sim}{V}_3)$ system.

Output Of Stresses

σ_{11}, σ_{22}, σ_{12} in local $(\underset{\sim}{V}_1, \underset{\sim}{V}_2, \underset{\sim}{V}_3)$ system given at equally spaced layers though thickness. First layer is on positive $\underset{\sim}{V}_3$ direction surface.

Transformation

Displacement and rotation at corner nodes can be transformed to local direction.

Tying

Use the UFORMSN user subroutine.

Updated Lagrange Procedure and Finite Strain Plasticity

Updated Lagrange capability is available. Note, however, that since the curvature calculation is linearized, you have to select your load steps such that the rotation remains small within a load step.

Section Stress - Integration

Integration through the shell thickness is performed numerically using Simpson's rule. Use the SHELL SECT parameter to specify the number of integration points. This number must be odd. Seven points are enough for simple plasticity or creep analysis. Eleven points are enough for complex plasticity or creep (for example, thermal plasticity). The default is 11 points.

Beam Stiffeners

The element is fully compatible with open- and closed-section beam element types 78 and 79.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 85. See Element 85 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered as a design variable.

Element 140

Bilinear Thick-shell Element with One-point Quadrature

This is a four-node, thick-shell element with global displacements and rotations as degrees of freedom. Bilinear interpolation is used for the coordinates, the displacements, and the rotations. This shell element adopts the degenerated shell geometry and reduced integration scheme, where only one integration point, in conjunction with physical stabilization approach, is utilized. The Assumed Natural Strain (ANS) method is employed to prevent shear locking. The strain field is derived from the Lagrangian strain tensor based on the convective coordinate system. In order to consider warped shell geometry, the nodal fiber coordinate system at each shell node is defined and updated with step-by-step procedures. Meanwhile, the rigid body motion is extracted by applying the rigid body projection method. For details, refer to: Cardoso, R.P.R., Yoon, J.W. et al., *Comp Methods Appl. Mech. Engr*., 191/45, pp. 5177-5204 (2002).

Furthermore, parameters are added to enhance the membrane and shear strains to improve the membrane and transverse shear behavior of this element. The membrane enhancement includes four parameters to improve the membrane behavior. The membrane and transverse shear enhancement includes two extra transverse shear parameters in addition to the four parameters for the membrane enhancement. This improves both the membrane and transverse shear behavior. For details, refer to Cardoso, R.P.R., Yoon, J.W. and Valente, R.A.F., "A new approach to reduce membrane and transverse shear locking for one-point quadrature shell elements: linear formulation", *Inter. J. for Numerical Methods in Engineering*, 66/2, 214-249 (2006)).

To include the enhanced formulation in the analysis using the current version, it is necessary to include the FEATURE,5801 parameter for linear analysis or FEATURE,5802 parameter for nonlinear analysis to the input file.

Compared to the full integration scheme-based shell element, this shell element shows better computational efficiency and uses less memory without sacrificing accuracy. It also shows an excellent performance for distorted elements.

All constitutive relations can be used with this element. The mass matrix of this element is formed using four-point Gaussian integration.

This element fails in passing the membrane patch test. An improved version has been published later to resolve this problem: Rui, P. R. Cardoso and Jeong-Whan Yoon, "One point quadrature shell elements: a study on convergence and patch tests", *Computational Mechanics*, 40, 871-883 (2007). However, for warped (or curved) element geometries, it's accuracy is not as good as element 140 which is based on the convective coordinate system.

Geometric Basis

The element is defined geometrically by the (x,y,z) coordinates of the four corner nodes. Due to the bilinear interpolation, the surface forms a hyperbolic paraboloid which is allowed to degenerate to a plate. The element thickness is specified in the GEOMETRY option.

The stress output is given in local orthogonal surface directions, $\underset{\sim}{V}_1$, $\underset{\sim}{V}_2$, and $\underset{\sim}{V}_3$, which are defined for the centroid in the following way (see Figure 3-218). $a_{i\,=\,1\,\sim\,4}$ are fiber vectors defined at nodes.

At the centroid the vectors tangent to the curves with constant isoparametric coordinates are normalized.

$$t_1 = \frac{\partial x}{\partial \xi} \Big/ \left|\frac{\partial x}{\partial \xi}\right|, \quad t_2 = \frac{\partial x}{\partial \eta} \Big/ \left|\frac{\partial x}{\partial \eta}\right|$$

Now a new basis is being defined as:

$$s = t_1 + t_2, \quad d = t_1 - t_2$$

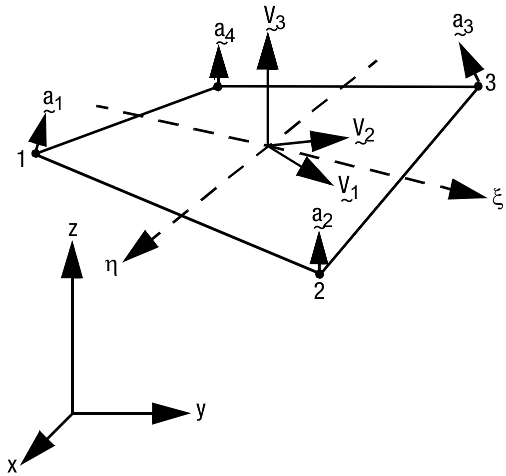


Figure 3-218 Form of Element 140

After normalizing these vectors by:

$$\bar{s} = s / \sqrt{2}|s| \quad \bar{d} = d / \sqrt{2}|d|$$

The local orthogonal directions are then obtained as:

$$\tilde{V}_1 = \bar{s} + \bar{d}, \; \tilde{V}_2 = \bar{s} - \bar{d} \text{ , and } \tilde{V}_3 = \tilde{V}_1 \times \tilde{V}_2$$

In this way, the vectors $\frac{\partial x}{\partial \xi}$, $\frac{\partial x}{\partial \eta}$ and $\tilde{V}_2$, $\tilde{V}_2$ have the same bisecting plane.

The local directions at the Gaussian integration points are found by projection of the centroid directions. Hence, if the element is flat, the directions at the Gauss points are identical to those at the centroid.

Displacements

The six nodal displacement variables are as follows:

u, v, w	Displacement components defined in global Cartesian x,y,z coordinate system.
ϕ_x, ϕ_y, ϕ_z	Rotation components about global x, y and z axis respectively.

Quick Reference

Type 140

Bilinear, four-node shell element including transverse shear effects using reduced integration with physical stabilization.

Connectivity

Four nodes per element. The element can be collapsed into a triangle.

Geometry

Bilinear thickness variation is allowed in the plane of the element. Thicknesses at first, second, third, and fourth nodes of the element are stored for each element in the first (`EGEOM1`), second (`EGEOM2`), third (`EGEOM3`) and fourth (`EGEOM4`), geometry data fields, respectively. If `EGEOM2` = `EGEOM3` = `EGEOM4` = `0`, then a constant thickness (`EGEOM1`) is assumed for the element.

Note that the NODAL THICKNESS model definition option can also be used for the input of element thickness.

If the element needs to be offset from the user-specified position, the eighth data field (`EGEOM8`) is set to -2.0. In this case, an extra line containing the offset information is provided (see the GEOMETRY option in Marc Volume C: Program Input). The offset magnitudes along the element normal for the four corner nodes are provided in the first, second, third and fourth data fields of the extra line. A uniform offset for all nodes can be set by providing the offset magnitude in the first data field and then setting the constant offset flag (sixth data field of the extra line) to 1.

Coordinates

Three coordinates per node in the global x-, y-, and z-directions.

Degrees of Freedom

Six degrees of freedom per node:

1 = u = global (Cartesian) x-displacement
2 = v = global (Cartesian) y-displacement
3 = w = global (Cartesian) z-displacement
4 = ϕ_x = rotation about global x-axis
5 = ϕ_y = rotation about global y-axis
6 = ϕ_z = rotation about global z-axis

Distributed Loads

A table of distributed loads is listed below:

Load Type	Description
1	Uniform gravity load per surface area in -z-direction.
2	Uniform pressure with positive magnitude in -V_3-direction, magnitude is given in the FORCEM user subroutine.
3	Nonuniform gravity load per surface area in -z-direction.
4	Nonuniform pressure with positive magnitude in -V_3-direction, magnitude is given in the FORCEM user subroutine.
5	Nonuniform load per surface area in arbitrary direction, magnitude is given in the FORCEM user subroutine.
11	Uniform edge load in the plane of the surface on the 1-2 edge and perpendicular to this edge.

Load Type	Description
12	Nonuniform edge load magnitude is given in the FORCEM user subroutine in the plane of the surface on the 1-2 edge.
13	Nonuniform edge load magnitude and direction is given in the FORCEM user subroutine on 1-2 edge.
14	Uniform load on the 1-2 edge of the shell, tangent to, and in the direction of the 1-2 edge
15	Uniform load on the 1-2 edge of the shell. Perpendicular to the shell; i.e., $-v_3$ direction
21	Uniform edge load in the plane of the surface on the 2-3 edge and perpendicular to this edge.
22	Nonuniform edge load magnitude is given in the FORCEM user subroutine in the plane of the surface on the 2-3 edge.
23	Nonuniform edge load magnitude and direction is given in the FORCEM user subroutine on 2-3 edge.
24	Uniform load on the 2-3 edge of the shell, tangent to and in the direction of the 2-3 edge
25	Uniform load on the 2-3 edge of the shell. Perpendicular to the shell; i.e., $-v_3$ direction
31	Uniform edge load in the plane of the surface on the 3-4 edge and perpendicular to this edge.
32	Nonuniform edge load magnitude is given in the FORCEM user subroutine in the plane of the surface on the 3-4 edge.
33	Nonuniform edge load magnitude and direction is given in the FORCEM user subroutine on 3-4 edge.
34	Uniform load on the 3-4 edge of the shell, tangent to and in the direction of the 3-4 edge
35	Uniform load on the 3-4 edge of the shell. Perpendicular to the shell; i.e.,$-v_3$ direction
41	Uniform edge load in the plane of the surface on the 4-1 edge and perpendicular to this edge.
42	Nonuniform edge load magnitude is given in the FORCEM user subroutine in the plane of the surface on the 4-1 edge.
43	Nonuniform edge load magnitude and direction is given in the FORCEM user subroutine on 4-1 edge.
44	Uniform load on the 4-1 edge of the shell, tangent to and in the direction of the 4-1 edge
45	Uniform load on the 4-1 edge of the shell. Perpendicular to the shell; i.e., $-v_3$ direction
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.

Load Type	Description
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

All edge loads require the input as force per unit length.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Point Loads

Point loads and moments can also be applied at the nodes.

Output Of Strains

Generalized strain components are:

Middle surface stretches:	ε_{11} ε_{22} ε_{12}
Middle surface curvatures:	κ_{11} κ_{22} κ_{12}
Transverse shear strains:	γ_{23} γ_{31}

in local $(\underset{\sim}{V}_1, \underset{\sim}{V}_2, \underset{\sim}{V}_3)$ system.

Output Of Stresses

σ_{11}, σ_{22}, σ_{12}, σ_{23}, σ_{31} in local $(\underset{\sim}{V}_1, \underset{\sim}{V}_2, \underset{\sim}{V}_3)$ system given at equally spaced layers though thickness. First layer is on positive $\underset{\sim}{V}_3$ direction surface.

Transformation

Displacement and rotation at corner nodes can be transformed to local direction.

Tying

Use the UFORMSN user subroutine.

Updated Lagrange Procedure and Finite Strain Plasticity

Updated Lagrange capability is available. Note, however, that since the curvature calculation is linearized, you have to select your load steps such that the rotation remains small within a load step.

Section Stress - Integration

Integration through the shell thickness is performed numerically using Simpson's rule. Use the SHELL SECT parameter to specify the number of integration points. This number must be odd. Seven points are enough for simple plasticity or creep analysis. Eleven points are enough for complex plasticity or creep (for example, thermal plasticity). The default is 11 points.

Beam Stiffeners

The element is fully compatible with open- and closed-section beam element types 78 and 79.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 85.

Design Variables

The thickness can be considered as a design variable.

Element 141

Heat Transfer Shell

Not available at this time.

Element 142 — Eight-node Axisymmetric Rebar Element with Twist

This element is similar to element 48, but is written for axisymmetric applications with torsional strains. It is a hollow, isoparametric 8-node quadrilateral in which you can place single strain members such as reinforcing rods or cords (i.e., rebars). The element is then used in conjunction with the 8-node axisymmetric continuum element with twist (for example, element 66 or 67) to represent cord reinforced composite materials. This technique allows the rebar and the filler to be represented accurately with respect to their stress distribution, so that separate constitutive theories can be used in each (for example, cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via the REBAR option or the REBAR user subroutine.

Integration

It is assumed that several "layers" of rebars are presented. The number of such "layers" is input by you via the REBAR option or, if the REBAR user subroutine is used, in the second element geometry field. A maximum number of five layers can be used within a rebar element. Each layer is assumed to be similar to a pair of opposite element edges (although the rebar direction is arbitrary), so that the "thickness" direction is from one of the element edges to its opposite one. For instance (see Figure 3-219), if the layer is similar to the 1, 2 and 3, 4 edges of the element, the "thickness" direction is from the 1, 2 edge to 3, 4 edge of the element. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains two integration points.

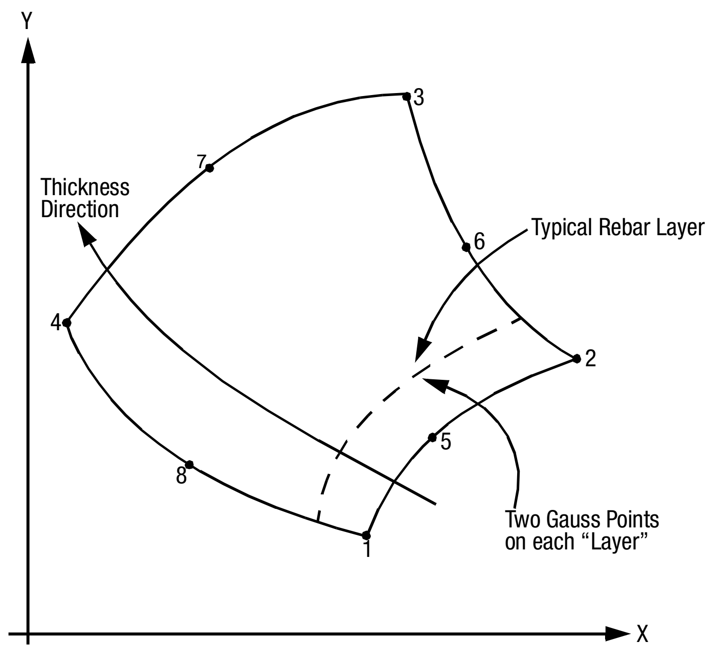


Figure 3-219 Eight-node Rebar Element Conventions

At each such integration point on each layer, you must input (via either the REBAR option or the REBAR user subroutine) the position, equivalent thickness (or, alternatively, spacing and area of cross section), and orientation of the rebars. See Marc

Volume C: Program Input for the REBAR option or Marc Volume D: User Subroutines and Special Routines for the REBAR user subroutine.

Quick Reference

Type 142

Eight-node, isoparametric rebar element with torsional strains.

Connectivity

Eight nodes per element. Node numbering of the element is same as that for element 66 or 67.

Geometry

The number of rebar layers and the isoparametric direction of the layers are defined using the REBAR model definition option.

Coordinates

Two global coordinates in z- and r-directions.

Degrees of Freedom

Displacement output in global components is as follows:

1 - z
2 - r
3 - θ

Tractions

Point loads can be applied at the nodes but no distributed loads are available. Distributed loads are applied only to corresponding 8-node axisymmetric element with twist (for example, element types 66 or 67).

Output of Stress and Strain

One stress and one strain are output at each integration point – the axial rebar value. For the case of large deformations, the stress is the second Piola-Kirchhoff stress and the strain is the Green strain.

By using post code 471 and 481 (representing second Piola-Kirchhoff stress in undeformed configuration and Cauchy stress in deformed configuration, respectively), the rebar stress can be written into the post file in the form of a stress tensor defined in the global coordinate directions. Mentat can be used to plot the principal directions of the stress tensor to show the magnitude of rebar stress, rebar orientation, and their changes based on deformation.

Transformations

Any local set (u,v) can be used in the (z-r) plane at any node.

Special Considerations

Either the REBAR option or the REBAR user subroutine is needed to input the position, size, and orientation of the rebars.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Element 143

Four-node Plane Strain Rebar Element

This element is isoparametric, plane strain, 4-node hollow quadrilateral in which you can place single strain members such as reinforcing rods or cords (that is, rebars). The element is then used in conjunction with the 4-node plane strain continuum element (for example, element 11 or 80) to represent cord reinforced composite materials. This technique allows the rebar and the filler to be represented accurately with respected to their stress distribution, so that separate constitutive theories can be used in each (for example, cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via the REBAR option or the REBAR user subroutine.

Integration

It is assumed that several "layers" of rebars are presented. The number of such "layers" is input by you via the REBAR option or, if the REBAR user subroutine is used, in the second element geometry field. A maximum number of five layers can be used within a rebar element. Each layer is assumed to be similar to a pair of opposite element edges (although the rebar direction is arbitrary), so that the "thickness" direction is from one of the element edges to its opposite one. For instance (see Figure 3-220), if the layer is similar to the 1, 2 and 3, 4 edges of the element, the "thickness" direction is from the 1, 2 edge to 3, 4 edge of the element. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains two integration points.

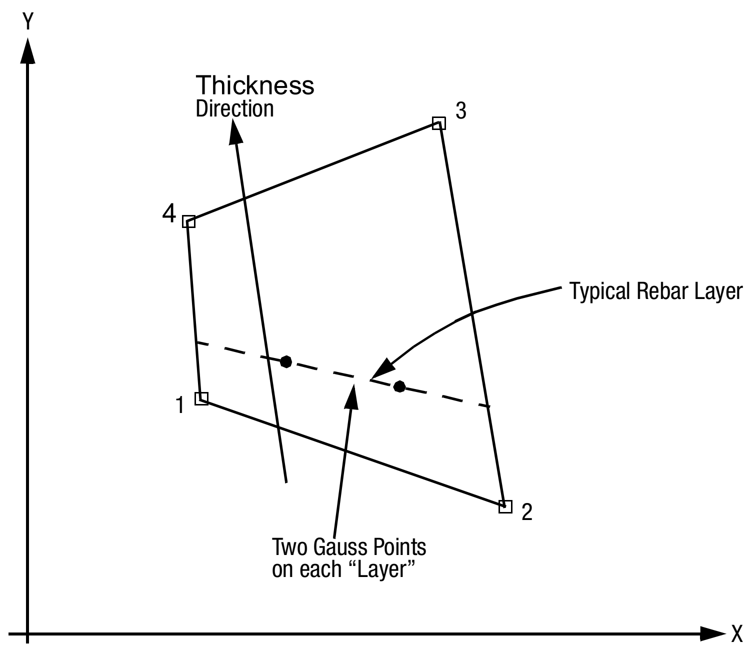


Figure 3-220 Four-node Rebar Element Conventions

At each such integration point on each layer, you must input (via either the REBAR option or the REBAR user subroutine) the position, equivalent thickness (or, alternatively, spacing and area of cross-section), and orientation of the rebars. See Marc Volume C: Program Input for the REBAR option or Marc Volume D: User Subroutines and Special Routines for the REBAR user subroutine.

Quick Reference

Type 143

Four-node, isoparametric rebar element to be used with 4-node plane strain continuum element.

Connectivity

Four nodes per element. Node numbering of the element is same as that for element 11 or 80.

Geometry

Element thickness (in z-direction) in first field. Default thickness is unity. Note, this should not be confused with the "thickness" concept associated with rebar layers.

The number of rebar layers and the isoparametric direction of the layers are defined using the REBAR model definition option.

Coordinates

Two global coordinates x- and y-directions.

Degrees of Freedom

Displacement output in global components is as follows:

1 - u
2 - v

Tractions

Point loads can be applied at the nodes, but no distributed loads are available. Distributed loads are applied only to corresponding 4-node plane strain elements (e.g., element types 11 or 80).

Output of Stress and Strain

One stress and one strain are output at each integration point – the axial rebar value. For the case of large deformations, the stress is the second Piola-Kirchhoff stress and the strain is the Green strain.

By using post code 471 and 481 (representing Second Piola Kirchhoff stress in undeformed configuration and Cauchy stress in deformed configuration, respectively), the rebar stress can be written into the post file in the form of a stress tensor defined in the global coordinate directions. Mentat can be used to plot the principal directions of the stress tensor to show the magnitude of rebar stress, rebar orientation, and their changes based on deformation.

Transformation

Any local set (u,v) can be used at any node.

Special Consideration

Either the REBAR option or the REBAR user subroutine is needed to input the position, size, and orientation of the rebars.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Element 144

Four-node Axisymmetric Rebar Element

This element is similar to element 143, but is written for axisymmetric conditions. It is a hollow, isoparametric 4-node quadrilateral in which you can place single strain members such as reinforcing rods or cords (that is, rebars). The element is then used in conjunction with the 4-node axisymmetric continuum element (for example., element 10 or 82) to represent cord reinforced composite materials. This technique allows the rebar and the filler to be represented accurately with respect to their stress distribution, so that separate constitutive theories can be used in each (for example, cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via the REBAR option or the REBAR user subroutine.

Integration

It is assumed that several "layers" of rebars are presented. The number of such "layers" is input by you via the REBAR option or, if the REBAR user subroutine is used, in the second element geometry field. A maximum number of five layers can be used within a rebar element. Each layer is assumed to be similar to a pair of opposite element edges (although the rebar direction is arbitrary), so that the "thickness" direction is from one of the element edges to its opposite one. For instance (see Figure 3-221), if the layer is similar to the 1, 2 and 3, 4 edges of the element, the "thickness" direction is from the 1, 2 edge to 3, 4 edge of the element. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains two integration points.

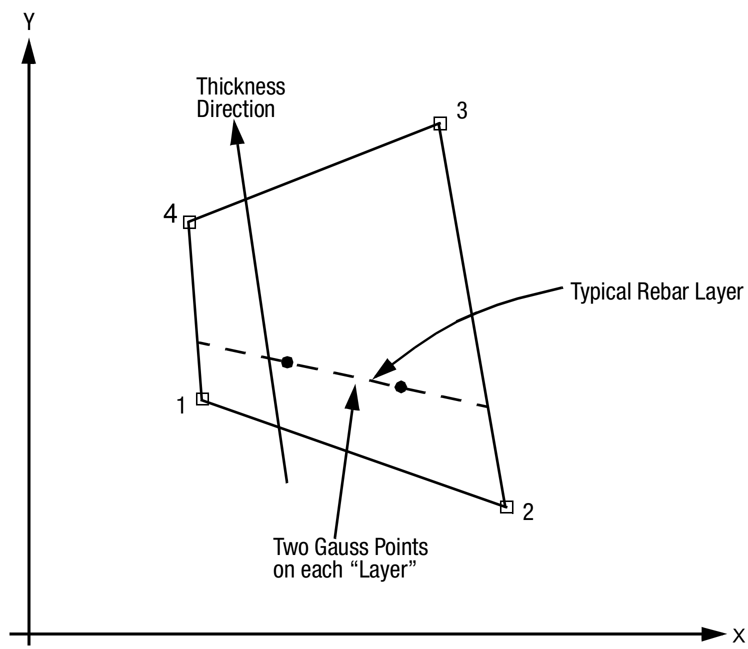


Figure 3-221 Four-node Rebar Element Conventions

At each such integration point on each layer, you must input (via either the REBAR option or the REBAR user subroutine) the position, equivalent thickness (or, alternatively, spacing and area of the cross section), and orientation of the rebars. See

Marc Volume C: Program Input for the REBAR option or Marc Volume D: User Subroutines and Special Routines for the REBAR user subroutine.

Quick Reference

Type 144

Four-node, isoparametric rebar element to be used with 4-node axisymmetric continuum element.

Connectivity

Four nodes per element. Node numbering of the element is same as that for element 10 or 82.

Geometry

The number of rebar layers and the isoparametric direction of the layers are defined using the REBAR model definition option.

Coordinates

Two global coordinates z- and r-directions.

Degrees of Freedom

Displacement output in global components is as follows:

1 - z
2 - r

Tractions

Point loads can be applied at the nodes, but no distributed loads are available. Distributed loads are applied only to corresponding 4-node axisymmetric elements (for example, element types 10 or 82).

Output of Stress and Strain

One stress and one strain are output at each integration point – the axial rebar value. For the case of large deformations, the stress is the second Piola-Kirchhoff stress and the strain is the Green strain.

By using post code 471 and 481 (representing Second Piola Kirchhoff stress in undeformed configuration and Cauchy stress in deformed configuration, respectively), the rebar stress can be written into the post file in the form of a stress tensor defined in the global coordinate directions. Mentat can be used to plot the principal directions of the stress tensor to show the magnitude of rebar stress, rebar orientation, and their changes based on deformation.

Transformation

Any local set (u,v) can be used in the (z-r) plane at any node.

Special Consideration

Either the REBAR option or the REBAR user subroutine is needed to input the position, size, and orientation of the rebars.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Element 145 Four-node Axisymmetric Rebar Element with Twist

This element is similar to element 144, but is written for axisymmetric applications with torsional strains. It is a hollow, isoparametric 4-node quadrilateral in which you can place single strain members such as reinforcing rods or cords (that is, rebars). The element is then used in conjunction with the 4-node axisymmetric continuum element with twist (for example, element 20 or 83) to represent cord reinforced composite materials. This technique allows the rebar and the filler to be represented accurately with respected to their stress distribution, so that separate constitutive theories can be used in each (for example, cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via the REBAR option or the REBAR user subroutine.

Integration

It is assumed that several "layers" of rebars are presented. The number of such "layers" is input by you via the REBAR option or, if the REBAR user subroutine is used, in the second element geometry field. A maximum number of five layers can be used within a rebar element. Each layer is assumed to be similar to a pair of opposite element edges (although the rebar direction is arbitrary), so that the "thickness" direction is from one of the element edges to its opposite one. For instance (see Figure 3-222), if the layer is similar to the 1, 2 and 3, 4 edges of the element, the "thickness" direction is from the 1, 2 edge to 3, 4 edge of the element. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains two integration points.

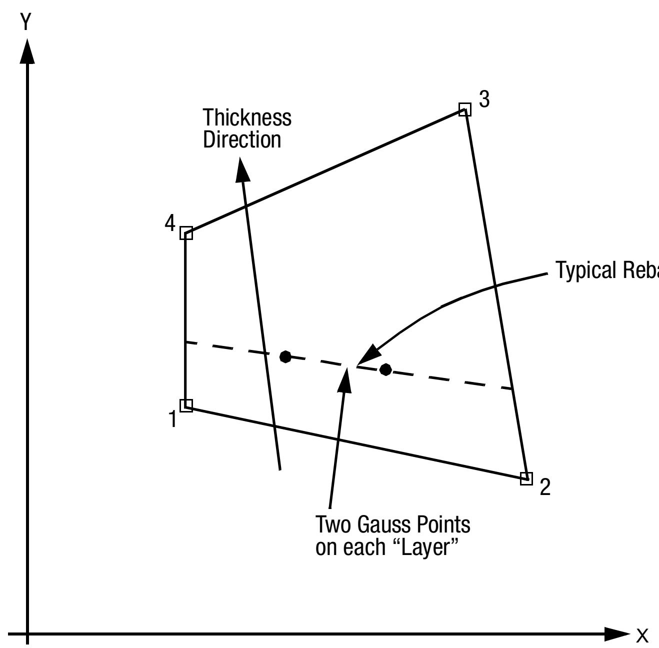


Figure 3-222 Four-node Rebar Element Conventions

At each such integration point on each layer, you must input (via either the REBAR option or the REBAR user subroutine) the position, equivalent thickness (or, alternatively, spacing and area of cross section), and orientation of the rebars. See Marc

Volume C: Program Input for the REBAR option or Marc Volume D: User Subroutines and Special Routines for the REBAR user subroutine.

Quick Reference

Type 145

Four-node, isoparametric rebar element with torsional strains.

Connectivity

Four nodes per element. Node numbering of the element is same as that for element 20 or 83.

Geometry

The number of rebar layers and the isoparametric direction of the layers are defined using the REBAR model definition option.

Coordinates

Two global coordinates z- and r-directions.

Degrees of Freedom

Displacement output in global components is as follows:

1 - z
2 - r
3 - θ

Tractions

Point loads can be applied at the nodes, but no distributed loads are available. Distributed loads are applied only to corresponding 4-node axisymmetric elements with twist (for example, element types 20 or 83).

Output of Stress and Strain

One stress and one strain are output at each integration point – the axial rebar value. For the case of large deformations, the stress is the second Piola-Kirchhoff stress and the strain is the Green strain.

By using post code 471 and 481 (representing Second Piola Kirchhoff stress in undeformed configuration and Cauchy stress in deformed configuration, respectively), the rebar stress can be written into the post file in the form of a stress tensor defined in the global coordinate directions. Mentat can be used to plot the principal directions of the stress tensor to show the magnitude of rebar stress, rebar orientation, and their changes based on deformation.

Transformation

Any local set (u,v) can be used in the (z-r) plane at any node.

Special Consideration

Either the REBAR option or the REBAR user subroutine is needed to input the position, size, and orientation of the rebars.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

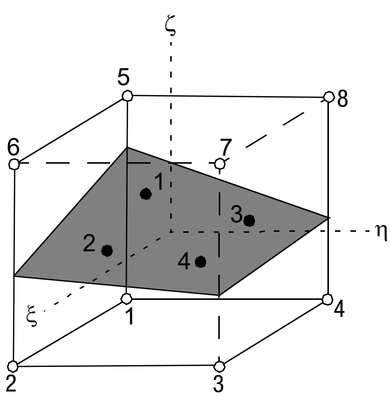
Element 146

Three-dimensional 8-node Rebar Element

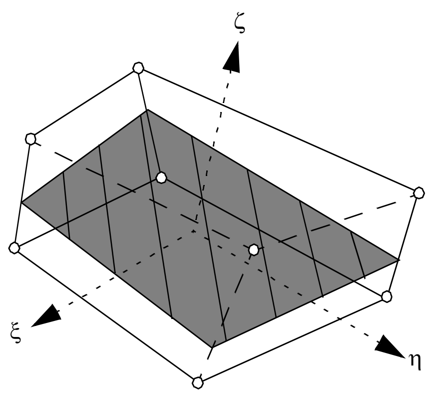
This element is an isoparametric, three-dimensional, 8-node empty brick in which you can place single strain members such as reinforcing rods or cords (that is, rebars). The element is then used in conjunction with the 8-node brick continuum element (for example, element types 7 or 84) to represent cord reinforced composite materials. This technique allows the rebar and the filler to be represented accurately with respect to their stress distribution, so that separate constitutive theories can be used in each (for example, cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via the REBAR option or the REBAR user subroutine.

Integration

It is assumed that several "layers" of rebars are presented. The number of such "layers" is input by you via the REBAR option or, if the REBAR user subroutine is used, in the second element geometry field. A maximum number of five layers can be used within a rebar element. Each layer is assumed to be similar to a pair of opposite element faces (although the rebar direction is arbitrary), so that the "thickness" direction is from one of the element faces to its opposite one. For instance (see Figure 3-223), if the layer is similar to the 1, 2, 3, 4 and 5, 6, 7, 8 faces of the element, the "thickness" direction is from the 1, 2, 3, 4 face to 5, 6, 7, 8 face of the element. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains four integration points (see Figure 3-223). At each such integration point on each layer, you must input (via either the REBAR option or the REBAR user subroutine) the position, equivalent thickness (or, alternatively, spacing and area of cross-section), and orientation of the rebars. See Marc Volume C: Program Input for the REBAR option or Marc Volume D: User Subroutines and Special Routines for the REBAR user subroutine.



(a) Parent Domain



(b) Mapping of Parent Domain

Figure 3-223 Typical Layer in 8-node 3-D Rebar Element

Quick Reference

Type 146

Eight-node, isoparametric rebar element to be used with 8-node brick continuum element.

Connectivity

Eight nodes per element. Node numbering of the element is same as that for element types 7 or 84.

Geometry

The number of rebar layers and the isoparametric direction of the layers are defined using the REBAR model definition option.

Coordinates

Three global coordinates in the x-, y-, and z-directions.

Degrees of Freedom

Displacement output in global components is as follows:

1 – u
2 – v
3 – w

Tractions

Point loads can be applied at the nodes, but no distributed loads are available. Distributed loads are applied only to corresponding 8-node brick elements (for example, element types 7 or 84).

Output Of Strains and Stresses

One stress and one strain are output at each integration point – the axial rebar value. For the case of large deformations, the stress is the second Piola-Kirchhoff stress and the strain is the Green strain.

By using post code 471 and 481 (representing Second Piola Kirchhoff stress in undeformed configuration and Cauchy stress in deformed configuration, respectively), the rebar stress can be written into the post file in the form of a stress tensor defined in the global coordinate directions. Mentat can be used to plot the principal directions of the stress tensor to show the magnitude of rebar stress, rebar orientation, and their changes based on deformation.

Transformation

Any local set (u,v,w) can be used at any node.

Special Consideration

Either the REBAR option or the REBAR user subroutine is needed to input the position, size, and orientation of the rebars.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Element 147

Four-node Rebar Membrane

This element is hollow, isoparametric 4-node membrane in which you can place single strain members such as reinforcing rods or cords (that is, rebars). The element is then used in conjunction with the 4-node membrane (element 18) to represent cord reinforced composite materials. This technique allows the rebar and the filler to be represented accurately with respected to their stress distribution, so that separate constitutive theories can be used in each (for example, cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via the REBAR option or the REBAR user subroutine.

Integration

It is assumed that several "layers" of rebars are presented. The number of such "layers" is input by you via the REBAR option or if the REBAR user subroutine is used in the second element geometry field. A maximum number of five layers can be used within a rebar element. The rebar layers are assumed to be placed on the same spatial position as that of the element (although the rebar direction is arbitrary and the "thickness" of the layers can be different). The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains four integration points. At each such integration point on each layer, you must input (via either the REBAR option or the REBAR user subroutine) the position, equivalent thickness (or, alternatively, spacing and area of cross-section), and orientation of the rebars. See Marc Volume C: Program Input for the REBAR option or Marc Volume D: User Subroutines and Special Routines for the REBAR user subroutine.

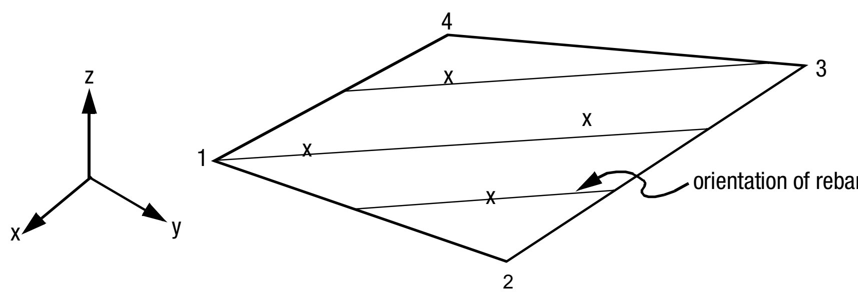


Figure 3-224 One Layer of Rebar Membrane Element

Quick Reference

Type 147

Four-node, isoparametric rebar membrane to be used with 4-node membrane (element 18).

Connectivity

Four nodes per element. Node numbering of the element is same as that for element 18.

Geometry

The number of rebar layers are defined using the REBAR model definition option.

Coordinates

Three global coordinates in the x-, y-, and z-directions.

Degrees of Freedom

Displacement output in global components is as follows:

1 – u
2 – v
3 – w

Tractions

Point loads can be applied at the nodes, but no distributed loads are available. Distributed loads are applied only to corresponding 4-node membrane (element type 18).

Output Of Strains and Stresses

One stress and one strain are output at each integration point – the axial rebar value. For the case of large deformations, the stress is the second Piola-Kirchhoff stress and the strain is the Green strain.

By using post code 471 and 481 (representing Second Piola Kirchhoff stress in undeformed configuration and Cauchy stress in deformed configuration, respectively), the rebar stress can be written into the post file in the form of a stress tensor defined in the global coordinate directions. Mentat can be used to plot the principal directions of the stress tensor to show the magnitude of rebar stress, rebar orientation, and their changes based on deformation.

Transformation

Nodal degrees of freedom can be transformed to local degrees of freedom.

Special Consideration

Either the REBAR option or the REBAR user subroutine is needed to input the position, size, and orientation of the rebars.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Element 148

Eight-node Rebar Membrane

This element is hollow, isoparametric 8-node membrane in which you can place single strain members such as reinforcing rods or cords (that is, rebars). The element is then used in conjunction with the 8-node membrane (element 30) to represent cord reinforced composite materials. This technique allows the rebar and the filler to be represented accurately with respected to their stress distribution, so that separate constitutive theories can be used in each (for example, cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via the REBAR option or the REBAR user subroutine.

Integration

It is assumed that several "layers" of rebars are presented. The number of such "layers" is input by you via the REBAR option or, if the REBAR user subroutine is used, in the second element geometry field. A maximum number of five layers can be used within a rebar element. The rebar layers are assumed to be placed on the same spatial position as that of the element (although the rebar direction is arbitrary and the "thickness" of the layers can be different). The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains four integration points. At each such integration point on each layer, you must input (via either the REBAR option or the REBAR user subroutine) the position, equivalent thickness (or, alternatively, spacing and area of cross-section), and orientation of the rebars. See Marc Volume C: Program Input for REBAR option or Marc Volume D: User Subroutines and Special Routines for the REBAR user subroutine.

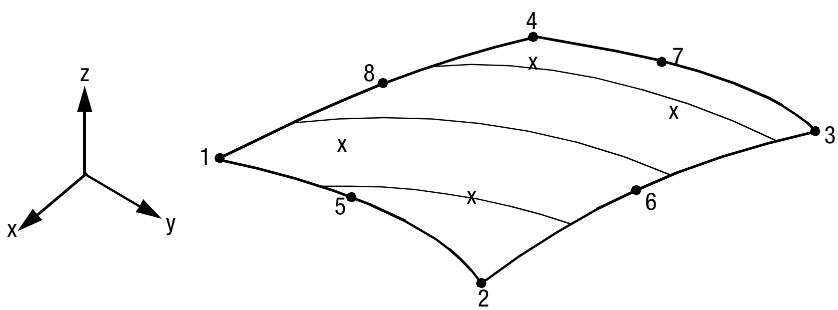


Figure 3-225 One Layer of Rebar Membrane Element

Quick Reference

Type 148

Eight-node, isoparametric rebar membrane to be used with 8-node membrane (element 30).

Connectivity

Eight nodes per element. Node numbering of the element is same as that for element 30.

Geometry

The number of rebar layers are defined using the REBAR model definition option.

Coordinates

Three global coordinates in the x-, y-, and z-directions.

Degrees of Freedom

Displacement output in global components is as follows:

1 – u
2 – v
3 – w

Tractions

Point loads can be applied at the nodes, but no distributed loads are available. Distributed loads are applied only to corresponding 8-node membrane (element type 30).

Output of Strains and Stresses

One stress and one strain are output at each integration point – the axial rebar value. For the case of large deformations, the stress is the second Piola-Kirchhoff stress and the strain is the Green strain.

By using post code 471 and 481 (representing Second Piola Kirchhoff stress in undeformed configuration and Cauchy stress in deformed configuration, respectively), the rebar stress can be written into the post file in the form of a stress tensor defined in the global coordinate directions. Mentat can be used to plot the principal directions of the stress tensor to show the magnitude of rebar stress, rebar orientation, and their changes based on deformation.

Transformation

Nodal degrees of freedom can be transformed to local degrees of freedom.

Special Consideration

Either the REBAR option or the REBAR user subroutine is needed to input the position, size, and orientation of the rebars.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Element 149

Three-dimensional, Eight-node Composite Brick Element

This element is an isoparametric, three-dimensional, 8-node composite brick. Different material properties can be used for different layers within the element (see Figure 3-226).

All constitutive equations available can be used for the element. To use the Mooney, Ogden, Arruda-Boyce, and Gent nearly incompressible rubber material models, the Updated Lagrange formulation must be used.

Integration

The number of continuum layers within an element is input via the COMPOSITE option. To ensure the stability of the element, a minimum number of two layers is required within the element. Each layer is assumed to be placed parallel to a pair of opposite element faces, so that the "thickness" direction is from one of the element faces to its opposite one. For instance, if the layer is parallel to the 1, 2, 3, 4 and 5, 6, 7, 8 faces of the element, the "thickness" direction is from element face 1, 2, 3, 4 to element face 5, 6, 7, 8 (see Figure 3-226). The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains four integration points (see Figure 3-226). On each layer, you must input, via the COMPOSITE option, the thickness (or percentage of the thickness) and the material set id. See Marc Volume C: Program Input for the COMPOSITE option. A maximum of 510 layers can be used within each element. The mass matrix of this element is formed using eight-point Gaussian integration.

Quick Reference

Type 149

Eight-node, isoparametric composite brick element.

Connectivity

Eight nodes per element. Node numbering of the element is the same as for element types 7 and 84.

Geometry

If a nonzero value is entered in the second data field (EGEOM2), the volumetric strain is constant throughout the element. This is particularly useful when materials in one or more layers are nearly incompressible.

The isoparametric direction of layers is defined in the third field (EGEOM3), enter 1, 2, or 3:

1 = Material layers are parallel to the 1, 2, 3, 4 and 5, 6, 7, 8 faces; the thickness direction is from 1, 2, 3, 4 face to the 5, 6, 7, 8, face of the element (Figure 3-226 (a)).

2 = Material layers are parallel to the 1, 4, 8, 5 and 2, 3, 7, 6 faces; the thickness direction is from 1, 4, 8, 5 face to the 2, 3, 7, 6 face of the element (Figure 3-226 (b)).

3 = Material layers are parallel to the 2, 1, 5, 6 and 3, 4, 8, 7 faces; the thickness direction is from 2, 1, 5, 6 face to the 3, 4, 8, 7 face of the element (Figure 3-226 (c)).

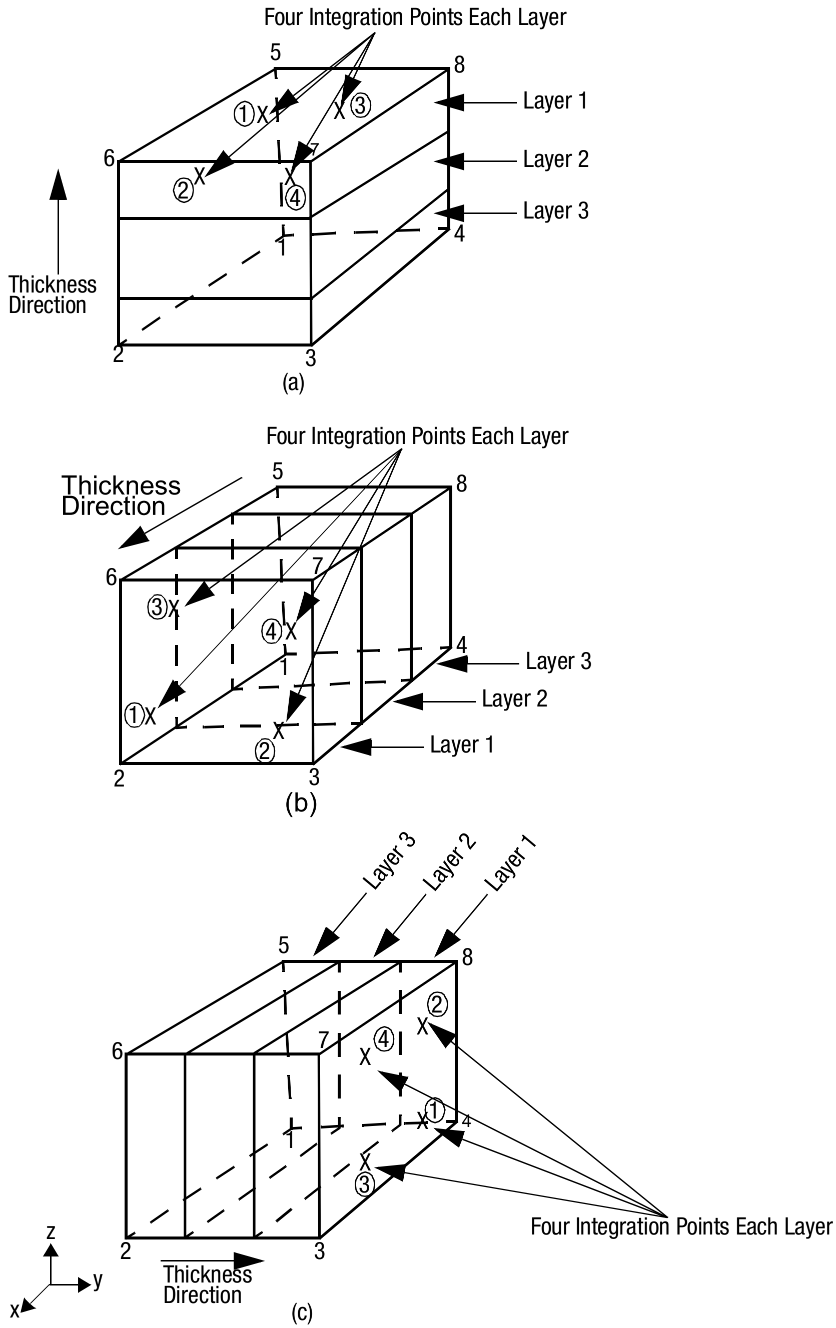


Figure 3-226 Typical 8-node 3-D Composite Element

Coordinates

Three global coordinates in x-, y-, and z-directions.

Degrees of Freedom

Displacement output in global components is as follows:

1 - u
2 - v
3 - w

Distributed Loads

Distributed loads chosen by value of IBODY are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3-4 face.
1	Nonuniform pressure on 1-2-3-4 face; magnitude supplied through the FORCEM user subroutine.
2	Uniform body force per unit volume in -z-direction.
3	Nonuniform body force per unit volume (e.g., centrifugal force); magnitude and direction supplied through the FORCEM user subroutine.
4	Uniform pressure on 6-5-8-7 face.
5	Nonuniform pressure on 6-5-8-7 face (FORCEM user subroutine).
6	Uniform pressure on 2-1-5-6 face.
7	Nonuniform pressure on 2-1-5-6 face (FORCEM user subroutine).
8	Uniform pressure on 3-2-6-7 face.
9	Nonuniform pressure on 3-2-6-7 face (FORCEM user subroutine).
10	Uniform pressure on 4-3-7-8 face.
11	Nonuniform pressure on 4-3-7-8 face (FORCEM user subroutine).
12	Uniform pressure on 1-4-8-5 face.
13	Nonuniform pressure on 1-4-8-5 face (FORCEM user subroutine).
20	Uniform pressure on 1-2-3-4 face.
21	Nonuniform load on 1-2-3-4 face; magnitude and direction supplied in the FORCEM user subroutine.
22	Uniform body force per unit volume in -z-direction.
23	Nonuniform body force per unit volume (e.g., centrifugal force); magnitude and direction supplied through the FORCEM user subroutine.
24	Uniform pressure on 6-5-8-7 face.
25	Nonuniform load on 6-5-8-7 face; magnitude and direction supplied in the FORCEM user subroutine.
26	Uniform pressure on 2-1-5-6 face.

Load Type	Description
27	Nonuniform load on 2-1-5-6 face; magnitude and direction supplied in the FORCEM user subroutine.
28	Uniform pressure on 3-2-6-7 face.
29	Nonuniform load on 3-2-6-7 face; magnitude and direction supplied in the FORCEM user subroutine.
30	Uniform pressure on 4-3-7-8 face.
31	Nonuniform load on 4-3-7-8 face; magnitude and direction supplied in the FORCEM user subroutine.
32	Uniform pressure on 1-4-8-5 face.
33	Nonuniform load on 1-4-8-5 face; magnitude and direction supplied in the FORCEM user subroutine.
40	Uniform shear 1-2-3-4 face in the 1-2 direction.
41	Nonuniform shear 1-2-3-4 face in the 1-2 direction.
42	Uniform shear 1-2-3-4 face in the 2-3 direction.
43	Nonuniform shear 1-2-3-4 face in the 2-3 direction.
48	Uniform shear 6-5-8-7 face in the 5-6 direction.
49	Nonuniform shear 6-5-8-7 face in the 5-6 direction.
50	Uniform shear 6-5-8-7 face in the 6-7 direction.
51	Nonuniform shear 6-5-8-7 face in the 6-7 direction.
52	Uniform shear 2-1-5-6 face in the 1-2 direction.
53	Nonuniform shear 2-1-5-6 face in the 1-2 direction.
54	Uniform shear 2-1-5-6 face in the 1-5 direction.
55	Nonuniform shear 2-1-5-6 face in the 1-5 direction.
56	Uniform shear 3-2-6-7 face in the 2-3 direction.
57	Nonuniform shear 3-2-6-7 face in the 2-3 direction.
58	Uniform shear 3-2-6-7 face in the 2-6 direction.
59	Nonuniform shear 2-3-6-7 face in the 2-6 direction.
60	Uniform shear 4-3-7-8 face in the 3-4 direction.
61	Nonuniform shear 4-3-7-8 face in the 3-4 direction.
62	Uniform shear 4-3-7-8 face in the 3-7 direction.
63	Nonuniform shear 4-3-7-8 face in the 3-7 direction.
64	Uniform shear 1-4-8-5 face in the 4-1 direction.
65	Nonuniform shear 1-4-8-5 face in the 4-1 direction.
66	Uniform shear 1-4-8-5 in the 1-5 direction.
67	Nonuniform shear 1-4-8-5 face in the 1-5 direction.

Load Type	Description
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Pressure forces are positive into element face.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains and Stresses

Stresses and Strains are output at each integration point. For the case of large deformations, the stresses are the second Piola-Kirchhoff stresses and the strains are the Green strains.

1 = global xx strain
2 = global yy strain
3 = global zz strain
4 = global xy strain
5 = global yz strain
6 = global xz strain

The PRINT ELEMENT or PRINT CHOICE options can be used to print out stress and strain results for specific integration points in the output file. The PRINT ELEMENT option is only valid for the first 28 integration points while the PRINT CHOICE option can be used for any integration point number (from 1 to a maximum of 2040). On the post file, stresses and strains can be obtained for the four integration points of each layer. When no layer is specified, post file results are presented for layer 1.

Interlaminar normal and shear stresses, as well as their directions, are output.

By using post code 501 or 511 (representing interlaminar normal and shear stresses, respectively), the interlaminar normal or shear stress can be written into the post file in the form of a stress tensor defined in the global coordinate directions. Mentat can be used to plot the principal directions of the stress tensor to show the magnitude and the direction of the stress, and their changes based on deformation.

Transformation

Any local set (u, v, w) can be used at any node.

There is no specific element coordinate system available for this element type. The preferred material orientation can be defined by using the ORIENTATION model definition option and the ply angle in the COMPOSITE model definition option. The first two axes of the local coordinate system, defined with ORIENTATION, must be parallel to the material layer. The third axis of the local system is, therefore, normal to the material layer and points to the thickness direction. The preferred direction is then a rotation of the local system about its third axis with the ply angle defined in COMPOSITE.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 175. See Element 175 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux generated by dissipated plastic energy is specified with type 101.

Element 150 Three-dimensional, Twenty-node Composite Brick Element

This element is isoparametric, three-dimensional, 20-node composite brick. Different material properties can be used for different layers within the element (see Figure 3-227).

All constitutive equations available can be used for the element. To use the Mooney and Ogden nearly incompressible material model, the Updated Lagrange formulation must be used.

Integration

The number of continuum layers within an element is input via the COMPOSITE option. To ensure the stability of the element, a minimum number of two layers is required within the element. Each layer is assumed to be placed parallel to a pair of opposite element faces, so that the "thickness" direction is from one of the element faces to its opposite one. For instance (see Figure 3-227), if the layer is parallel to the 1, 2, 3, 4 and 5, 6, 7, 8 faces of the element, the "thickness" direction is from element face 1, 2, 3, 4 to element face 5, 6, 7, 8. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains four integration points (see Figure 3-227). On each layer, you must input, via the COMPOSITE option, the thickness (or percentage of the thickness) and the material set id. See Marc Volume C: Program Input for the COMPOSITE option. A maximum of 510 layers can be used within each element. The mass matrix of this element is formed using twenty-seven-point Gaussian integration.

Quick Reference

Type 150

Twenty-node, isoparametric composite brick element.

Connectivity

Twenty nodes per element. Node numbering of the element is the same as for element types 21 and 61.

Geometry

The isoparametric direction of layers is defined in the third field (EGEOM3), enter 1, 2, or 3:

1 = Material layers are parallel to the 1, 2, 3, 4 and 5, 6, 7, 8 faces; the thickness direction is from the 1, 2, 3, 4 face to the 5, 6, 7, 8 face of the element (Figure 3-227 (a)).

2 = Material layers are parallel to the 1, 4, 8, 5 and 2, 3, 7, 6 faces; the thickness direction is from the 1, 4, 8, 5 face to the 2, 3, 7, 6 face of the element (Figure 3-227 (b)).

3 = Material layers are parallel to the 2, 1, 5, 6 and 3, 4, 8, 7 faces; the thickness direction is from the 2, 1, 5, 6 face to the 3, 4, 8, 7 face of the element (Figure 3-227 (c)).

Coordinates

Three global coordinates in x-, y-, and z-directions.

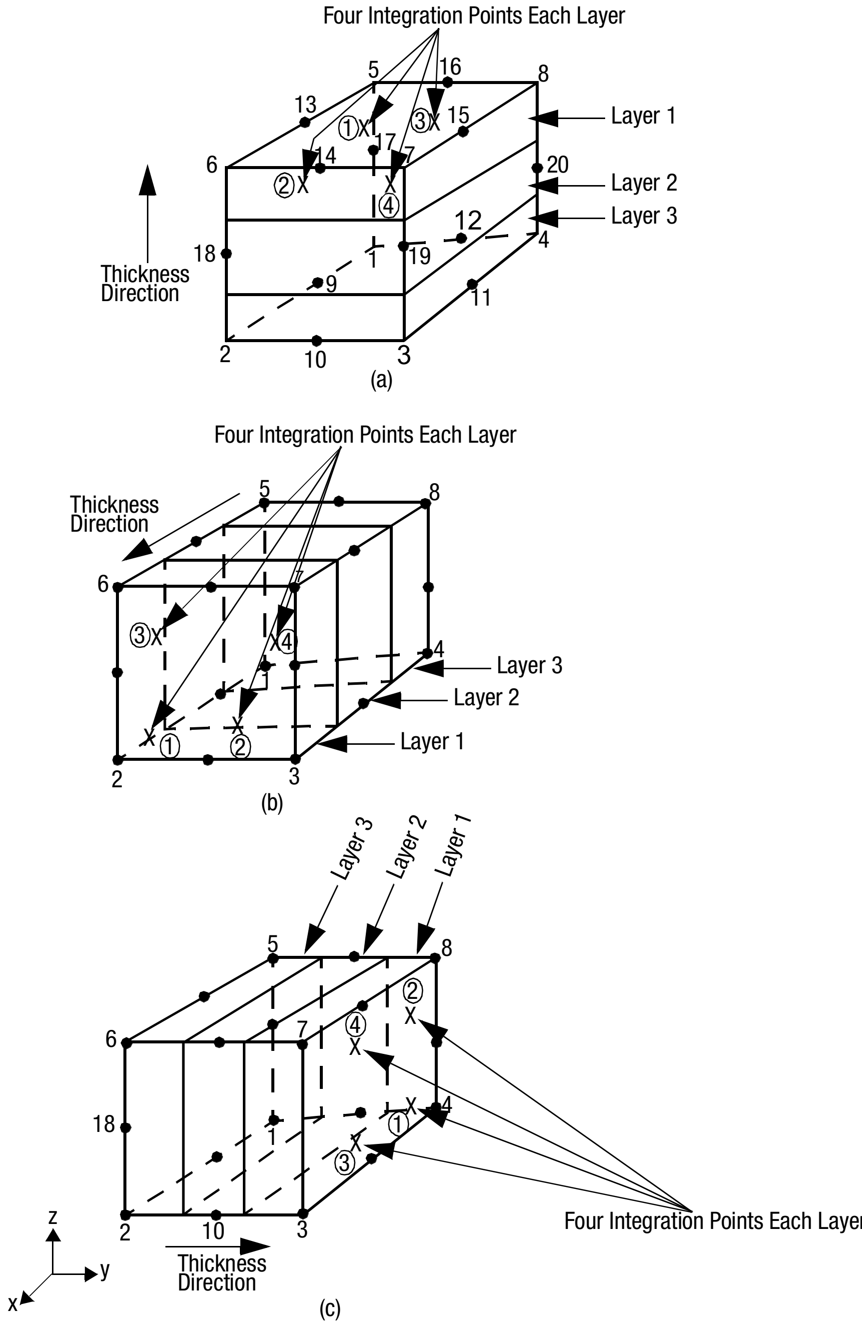


Figure 3-227 Typical 20-node 3-D Composite Element

Degrees of Freedom

Displacement output in global components is as follows:

1 - u
2 - v
3 - w

Distributed Loads

Distributed loads chosen by value of IBODY are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3-4 face.
1	Nonuniform pressure on 1-2-3-4 face; magnitude supplied through the FORCEM user subroutine.
2	Uniform body force per unit volume in -z-direction.
3	Nonuniform body force per unit volume (e.g., centrifugal force); magnitude and direction supplied through the FORCEM user subroutine.
4	Uniform pressure on 6-5-8-7 face.
5	Nonuniform pressure on 6-5-8-7 face.
6	Uniform pressure on 2-1-5-6 face.
7	Nonuniform pressure on 2-1-5-6 face.
8	Uniform pressure on 3-2-6-7 face.
9	Nonuniform pressure on 3-2-6-7 face.
10	Uniform pressure on 4-3-7-8 face.
11	Nonuniform pressure on 4-3-7-8 face.
12	Uniform pressure on 1-4-8-5 face.
13	Nonuniform pressure on 1-4-8-5 face.
20	Uniform pressure on 1-2-3-4 face.
21	Nonuniform load on 1-2-3-4 face; magnitude and direction supplied in the FORCEM user subroutine.
22	Uniform body force per unit volume in -z-direction.
23	Nonuniform body force per unit volume (e.g., centrifugal force; magnitude supplied through the FORCEM user subroutine)
24	Uniform pressure on 6-5-8-7 face.
25	Nonuniform load on 6-5-8-7 face; magnitude and direction supplied in the FORCEM user subroutine.
26	Uniform pressure on 2-1-5-6 face.
27	Nonuniform load on 2-1-5-6 face; magnitude and direction supplied in the FORCEM user subroutine.
28	Uniform pressure on 3-2-6-7 face.
29	Nonuniform load on 3-2-6-7 face; magnitude and direction supplied in the FORCEM user subroutine.

Load Type	Description
30	Uniform pressure on 4-3-7-8 face.
31	Nonuniform load on 4-3-7-8 face; magnitude and direction supplied in the FORCEM user subroutine.
32	Uniform pressure on 1-4-8-5 face.
33	Nonuniform load on 1-4-8-5 face; magnitude and direction supplied in the FORCEM user subroutine.
40	Uniform shear 1-2-3-4 face in 12 direction.
41	Nonuniform shear 1-2-3-4 face in 12 direction.
42	Uniform shear 1-2-3-4 face in 23 direction.
43	Nonuniform shear 1-2-3-4 face in 23 direction.
48	Uniform shear 6-5-8-7 face in 56 direction.
49	Nonuniform shear 6-5-8-7 face in 56 direction.
50	Uniform shear 6-5-8-7 face in 67 direction.
51	Nonuniform shear 6-5-8-7 face in 67 direction.
52	Uniform shear 2-1-5-6 face in 12 direction.
53	Nonuniform shear 2-1-5-6 face in 12 direction.
54	Uniform shear 2-1-5-6 face in 15 direction.
55	Nonuniform shear 2-1-5-6 face in 15 direction.
56	Uniform shear 3-2-6-7 face in 23 direction.
57	Nonuniform shear 3-2-6-7 face in 23 direction.
58	Uniform shear 3-2-6-7 face in 26 direction.
59	Nonuniform shear 2-3-6-7 face in 26 direction.
60	Uniform shear 4-3-7-8 face in 34 direction.
61	Nonuniform shear 4-3-7-8 face in 34 direction.
62	Uniform shear 4-3-7-8 face in 37 direction.
63	Nonuniform shear 4-3-7-8 face in 37 direction.
64	Uniform shear 1-4-8-5 face in 41 direction.
65	Nonuniform shear 1-4-8-5 face in 41 direction.
66	Uniform shear 1-4-8-5 face in 15 direction.
67	Nonuniform shear 1-4-8-5 face in 15 direction.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.

Load Type	Description
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

The FORCEM user subroutine is called once per integration point when flagged. The magnitude of load defined by DIST LOADS is ignored and the FORCEM value is used instead.

For nonuniform surface pressure, force values need only be supplied for the nine integration points on the face of application. Nodal (concentrated) loads can also be supplied.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains and Stresses

Stresses and Strains are output at each integration point. For the case of large deformations, the stresses are the second Piola-Kirchhoff stresses and the strains are the Green strains.

1 = global xx strain
2 = global yy strain
3 = global zz strain
4 = global xy strain
5 = global yz strain
6 = global xz strain

The PRINT ELEMENT or PRINT CHOICE options can be used to print out stress and strain results for specific integration points in the output file. The PRINT ELEMENT option is only valid for the first 28 integration points while the PRINT CHOICE option can be used for any integration point number (from 1 to a maximum of 2040). On the post file, stresses and strains can be obtained for the four integration points of each layer. When no layer is specified, post file results are presented for layer 1.

Interlaminar normal and shear stresses, as well as their directions, are output.

By using post code 501 or 511 (representing interlaminar normal and shear stresses, respectively), the interlaminar normal or shear stress can be written into the post file in the form of a stress tensor defined in the global coordinate directions. Mentat can be used to plot the principal directions of the stress tensor to show the magnitude and the direction of the stress, and their changes based on deformation.

Transformation

Any local set (u, v, w) can be used at any node.

There is no specific element coordinate system available for this element type. The preferred material orientation can be defined by using the ORIENTATION model definition option and the ply angle in the COMPOSITE model definition option. The first two axes of the local coordinate system, defined with ORIENTATION, must be parallel to the material layer. The third axis of the local system is, therefore, normal to the material layer and points to the thickness direction. The preferred direction is then a rotation of the local system about its third axis with the ply angle defined in COMPOSITE.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 176. See Element 176 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux generated by dissipated plastic energy is specified with type 101.

Element 151

Quadrilateral, Plane Strain, Four-node Composite Element

This is a isoparametric, plane strain, four-node composite element. Different material properties can be used for different layers within the element (see Figure 3-228).

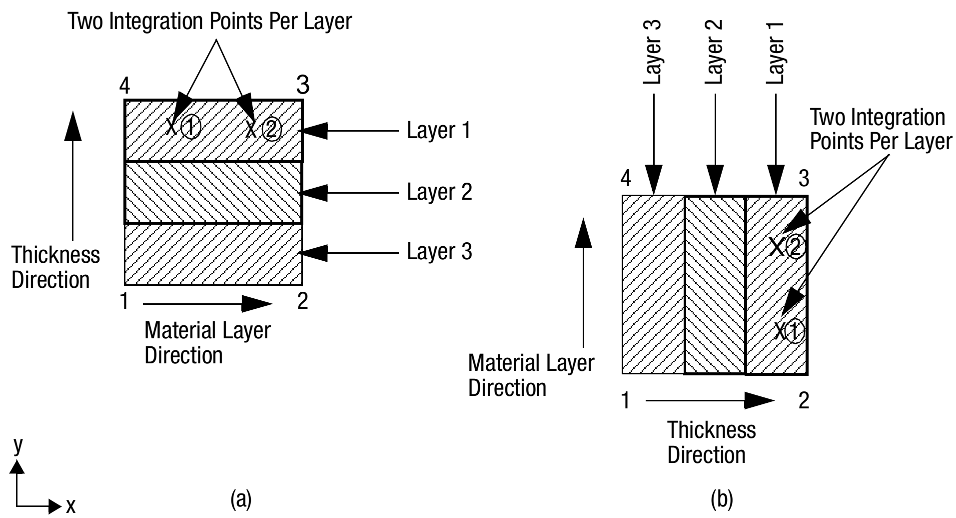


Figure 3-228 Typical Four-node, Quadrilateral, Plane Strain Composite Element

All constitutive equations available can be used for the element. To use the Mooney and Ogden nearly incompressible material model, the Updated Lagrange formulation must be used.

Integration

The number of continuum layers within an element is input via the COMPOSITE option. To ensure the stability of the element, a minimum number of two layers is required within the element. Each layer is assumed to be placed parallel to a pair of opposite element edges, so that the “thickness” direction is from one of the element edges to its opposite one. For instance (see Figure 3-228), if the layer is parallel to the 1, 2 and 3, 4 edges of the element, the “thickness” direction is from element edge 1, 2 to element edge 3, 4. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains two integration points (see Figure 3-228). On each layer, you must input, via the COMPOSITE option, the thickness (or percentage of the thickness) and the material set id. See Marc Volume C: Program Input for the COMPOSITE option. A maximum of 1020 layers can be used within each element.

Quick Reference

Type 151

Four-node, isoparametric, quadrilateral, plane strain composite element.

Connectivity

Four nodes per element. Node numbering of the element is the same as for element type 11.

Geometry

If a nonzero value is entered in the second data field (EGEOM2), the volumetric strain is constant throughout the element. This is particularly useful when materials in one or more layers are nearly incompressible.

The isoparametric direction of layers is defined in the third field (EGEOM3), enter 1 or 2:

1 = Material layers are parallel to the 1, 2 and 3, 4 edges; the thickness direction is from the 1, 2 edge to the 3, 4 edge of the element (Figure 3-228 (a)).

2 = Material layers are parallel to the 1, 4 and 2, 3 edges; the thickness direction is from the 1, 4 edge to the 2, 3 edge of the element (Figure 3-228 (b)).

Coordinates

Two global coordinates in the x- and y-directions.

Degrees of Freedom

1 - u
2 - v

Distributed Loads

Load types for distributed loads are as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force by unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element; magnitude supplied through the FORCEM user subroutine.
4	Nonuniform body force per unit volume in first coordinate direction; magnitude supplied through the FORCEM user subroutine.
5	Nonuniform body force per unit volume in second coordinate direction; magnitude supplied through the FORCEM user subroutine.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element; magnitude supplied through the FORCEM user subroutine.
8	Uniform pressure on 3-4 face of the element.
9	Nonuniform pressure on 3-4 face of the element; magnitude supplied through the FORCEM user subroutine.
10	Uniform pressure on 4-1 face of the element.
11	Nonuniform pressure on 4-1 face of the element; magnitude supplied through the FORCEM user subroutine.
20	Uniform shear force on side 1-2 (positive from 1 to 2).

Load Type	Description
21	Nonuniform shear force on side 1-2; magnitude supplied through the FORCEM user subroutine.
22	Uniform shear force on side 2-3 (positive from 2 to 3).
23	Nonuniform shear force on side 2-3; magnitude supplied through the FORCEM user subroutine.
24	Uniform shear force on side 3-4 (positive from 3 to 4).
25	Nonuniform shear force on side 3-4; magnitude supplied through the FORCEM user subroutine.
26	Uniform shear force on side 4-1 (positive from 4 to 1).
27	Nonuniform shear force on side 4-1; magnitude supplied through the FORCEM user subroutine.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter the magnitude of gravity acceleration in the z-direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

All pressures are positive when directed into the element. In addition, point loads can be applied at the nodes.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strain and Stresses

Stresses and Strains are output at each integration point. For the case of large deformations, the stresses are the second Piola-Kirchhoff stresses and the strains are the Green strains.

1 = global xx strain
2 = global yy strain
3 = global zz strain
4 = global xy strain
5 = global yz strain
6 = global xz strain

Note: The stress and strain tensors represent a full 3-D tensor, although the kinematics of the element only allows a plane strain deformation where strain components 5 and 6 must be identical to zero. These components may become nonzero if transformations to a preferred orientation system are being used as defined in the COMPOSITE model definition option with a nonzero ply orientation angle. The associated stress components 5 and 6 in this system may become nonzero and with nonisotropic material behavior even the stress components 5 and 6 in the global system may become nonzero. The user should make sure that the layer build-up is such, that globally the plane strain deformation is maintained by good approximation, so that the averaged stress state of the element remains a plane strain stress state by good approximation, although such a stress state may have been violated on a layer by layer basis (e.g., by having one layer with some positive ply orientation angle and the next layer with the same negative ply orientation angle).

The PRINT ELEMENT or PRINT CHOICE options can be used to print out stress and strain results for specific integration points in the output file. The PRINT ELEMENT option is only valid for the first 28 integration points while the PRINT CHOICE option can be used for any integration point number (from 1 to a maximum of 2040). On the post file, stresses and strains can be obtained for the four integration points of each layer. When no layer is specified, post file results are presented for layer 1.

Interlaminar normal and shear stresses, as well as their directions, are output.

By using post code 501 or 511 (representing interlaminar normal and shear stresses, respectively), the interlaminar normal or shear stress can be written into the post file in the form of a stress tensor defined in the global coordinate directions. Mentat can be used to plot the principal directions of the stress tensor to show the magnitude and the direction of the stress, and their changes based on deformation.

Transformation

Any local set (u, v) can be used at any node.

Local Element Coordinate System

A local element coordinate system is used to facilitate the definition of anisotropic composite layered materials. The first element direction is the material layer direction. The second element direction is normal to the material layers pointing to the thickness direction. The third element direction is the cross products of the first and second element. The material preferred directions are then defined by a rotation of the element coordinate system about the second element direction with an orientation angle defined in the COMPOSITE model definition option in the Marc Volume C: Program Input manual.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 177. See Element 177 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux generated by dissipated plastic energy is specified with type 101.

Element 152

Quadrilateral, Axisymmetric, Four-node Composite Element

This is a isoparametric, axisymmetric, four-node composite element. Different material properties can be used for different layers within the element (see Figure 3-229).

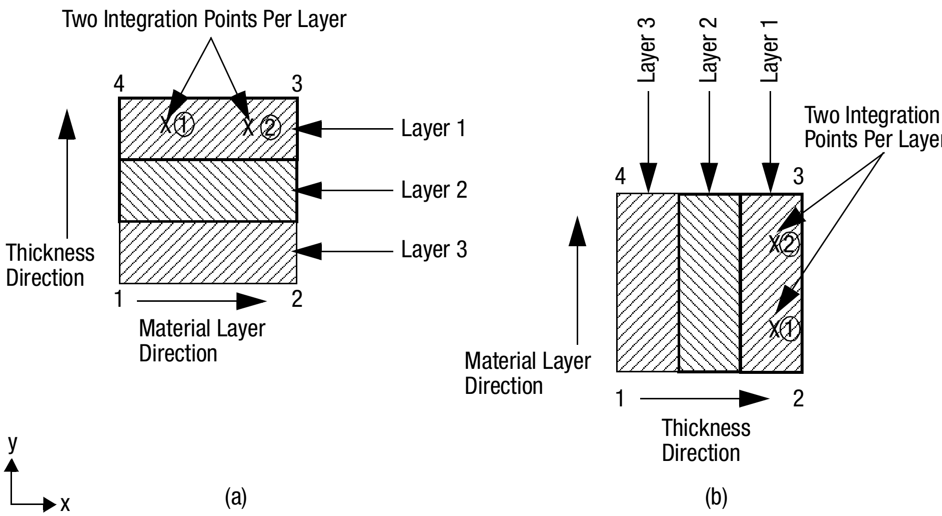


Figure 3-229 Typical Four-node, Quadrilateral, Axisymmetric Composite Element

All constitutive equations available can be used for the element. To use the Mooney and Ogden nearly incompressible material model, the Updated Lagrange formulation must be used.

Integration

The number of continuum layers within an element is input via the COMPOSITE option. A maximum number of five layers can be used. To ensure the stability of the element, a minimum number of two layers is required within the element. Each layer is assumed to be placed parallel to a pair of opposite element edges, so that the "thickness" direction is from one of the element edges to its opposite one. For instance (see Figure 3-229), if the layer is parallel to the 1, 2 and 3, 4 edges of the element, the "thickness" direction is from element edge 1, 2 to element edge 3, 4. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains two integration points (see Figure 3-229). On each layer, you must input, via the COMPOSITE option, the thickness (or percentage of the thickness) and the material set id. See Marc Volume C: Program Input for the COMPOSITE option. A maximum of 1020 layers can be used within each element.

Quick Reference

Type 152

Four-node, isoparametric, quadrilateral, axisymmetric composite element.

Connectivity

Four nodes per element. Node numbering of the element is the same as for element type 10.

Geometry

If a nonzero value is entered in the second data field (`EGEOM2`), the volumetric strain is constant throughout the element. This is particularly useful when materials in one or more layers are nearly incompressible.

The isoparametric direction of layers is defined in the third field (`EGEOM3`), enter 1 or 2:

1 = Material layers are parallel to the 1, 2 and 3, 4 edges; the thickness direction is from the 1, 2 edge to the 3, 4 edge of the element (Figure 3-229 (a)).

2 = Material layers are parallel to the 1, 4 and 2, 3 edges; the thickness direction is from the 1, 4 edge to the 2, 3 edge of the element (Figure 3-229 (b)).

Coordinates

Two global coordinates in the z- and r-directions.

Degrees of Freedom

1 - u global z-direction displacement (axial)
2 - v global r-direction displacement (radial)

Distributed Loads

Load types for distributed loads are as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force by unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element; magnitude supplied through the FORCEM user subroutine.
4	Nonuniform body force per unit volume in first coordinate direction; magnitude supplied through the FORCEM user subroutine.
5	Nonuniform body force per unit volume in second coordinate direction; magnitude supplied through the FORCEM user subroutine.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element; magnitude supplied through the FORCEM user subroutine.
8	Uniform pressure on 3-4 face of the element.
9	Nonuniform pressure on 3-4 face of the element; magnitude supplied through the FORCEM user subroutine.
10	Uniform pressure on 4-1 face of the element.
11	Nonuniform pressure on 4-1 face of the element; magnitude supplied through the FORCEM user subroutine.
20	Uniform shear force on side 1-2 (positive from 1 to 2).

Load Type	Description
21	Nonuniform shear force on side 1-2; magnitude supplied through the FORCEM user subroutine.
22	Uniform shear force on side 2-3 (positive from 2 to 3).
23	Nonuniform shear force on side 2-3; magnitude supplied through the FORCEM user subroutine.
24	Uniform shear force on side 3-4 (positive from 3 to 4).
25	Nonuniform shear force on side 3-4; magnitude supplied through the FORCEM user subroutine.
26	Uniform shear force on side 4-1 (positive from 4 to 1).
27	Nonuniform shear force on side 4-1; magnitude supplied through the FORCEM user subroutine.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

All pressures are positive when directed into the element. In addition, point loads can be applied at the nodes. The magnitude of point loads must correspond to the load integrated around the circumference.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strain and Stresses

Stresses and Strains are output at each integration point. For the case of large deformations, the stresses are the second Piola-Kirchhoff stresses and the strains are the Green strains.

1 = global zz strain
2 = global rr strain
3 = global $\theta\theta$ strain
4 = global zr strain
5 = global rθ strain
6 = global θz strain

Note: The stress and strain tensors represent a full 3-D tensor, although the kinematics of the element only allows an axisymmetric deformation where strain components 5 and 6 must be identical to zero. These components may become nonzero if transformations to a preferred orientation system are being used as defined in the COMPOSITE model definition option with a nonzero ply orientation angle. The associated stress components 5 and 6 in this system may become nonzero and with nonisotropic material behavior even the stress components 5 and 6 in the global system may become nonzero. The user should make sure that the layer build-up is such, that globally the axisymmetric deformation is maintained by good approximation, so that the averaged stress state of the element remains a plane strain stress state by good approximation, although such a stress state may have been violated on a layer by layer basis (e.g., by having one layer with some positive ply orientation angle and the next layer with the same negative ply orientation angle).

The PRINT ELEMENT or PRINT CHOICE options can be used to print out stress and strain results for specific integration points in the output file. The PRINT ELEMENT option is only valid for the first 28 integration points while the PRINT CHOICE option can be used for any integration point number (from 1 to a maximum of 2040). On the post file, stresses and strains can be obtained for the four integration points of each layer. When no layer is specified, post file results are presented for layer 1.

Interlaminar normal and shear stresses, as well as their directions, are output.

By using post code 501 or 511 (representing interlaminar normal and shear stresses, respectively), the interlaminar normal or shear stress can be written into the post file in the form of a stress tensor defined in the global coordinate directions. Mentat can be used to plot the principal directions of the stress tensor to show the magnitude and the direction of the stress, and their changes based on deformation.

Transformation

Any local set (z, r) can be used at any node.

Local Element Coordinate System

A local element coordinate system is used to facilitate the definition of anisotropic composite layered materials. The first element direction is the material layer direction. The second element direction is normal to the material layers pointing to the thickness direction. The third element direction is the cross products of the first and second element. The material preferred directions are then defined by a rotation of the element coordinate system about the second element direction with an orientation angle defined in the COMPOSITE model definition option in the Marc Volume C: Program Input manual.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 178. See Element 178 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux generated by dissipated plastic energy is specified with type 101.

Element 153

Quadrilateral, Plane Strain, Eight-node Composite Element

This is a isoparametric, plane strain, eight-node composite element. Different material properties can be used for different layers within the element (see Figure 3-230).

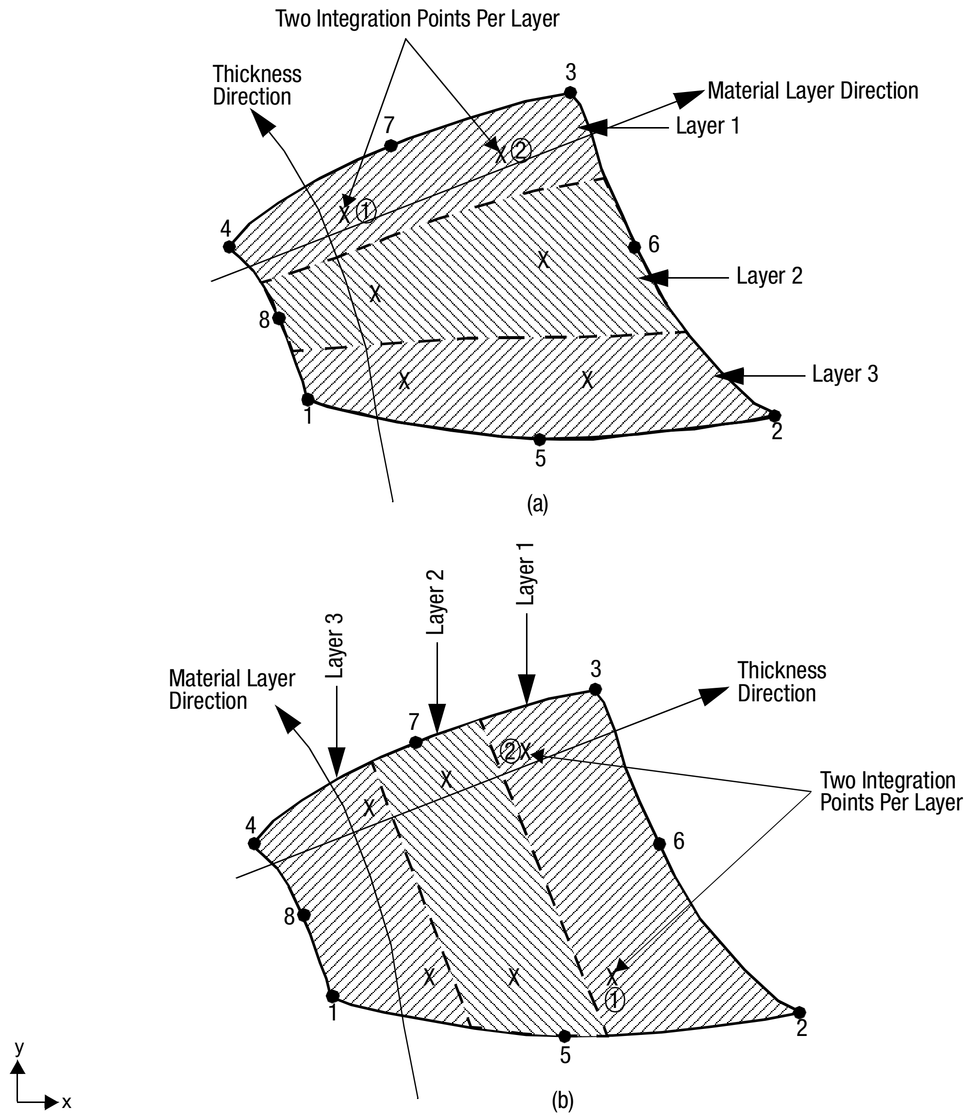


Figure 3-230 Typical Eight-node, Quadrilateral, Plane Strain Composite Element

All constitutive equations available can be used for the element. To use the Mooney and Ogden nearly incompressible material model, the Updated Lagrange formulation must be used.

Integration

The number of continuum layers within an element is input via the COMPOSITE option. To ensure the stability of the element, a minimum number of two layers is required within the element. Each layer is assumed to be placed parallel to a pair of opposite element edges, so that the “thickness” direction is from one of the element edges to its opposite one. For instance (see Figure 3-230), if the layer is parallel to the 1, 2 and 3, 4 edges of the element, the “thickness” direction is from element edge 1, 2 to element edge 3, 4. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains two integration points (see Figure 3-230). On each layer, you must input, via the COMPOSITE option, the thickness (or percentage of the thickness) and the material set id. See Marc Volume C: Program Input for the COMPOSITE option. A maximum of 1020 layers can be used within each element.

Quick Reference

Type 153

Eight-node, isoparametric, quadrilateral, plane strain composite element.

Connectivity

Eight nodes per element. Node numbering of the element is the same as for element type 27.

Geometry

The isoparametric direction of layers is defined in the third field (`EGEOM3`), enter 1 or 2:

1 = Material layers are parallel to the 1, 2 and 3, 4 edges; the thickness direction is from the 1, 2 edge to the 3, 4 edge of the element (Figure 3-230 (a)).

2 = Material layers are parallel to the 1, 4 and 2, 3 edges; the thickness direction is from the 1, 4 edge to the 2, 3 edge of the element (Figure 3-230 (b)).

Coordinates

Two global coordinates in the x- and y-directions.

Degrees of Freedom

1 - u
2 - v

Distributed Loads

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type (IBODY)	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.

Load Type (IBODY)	Description
4	Uniform body force in y-direction.
5	Nonuniform body force in the y-direction.
6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
8	Uniform pressure on 2-6-3 face.
9	Nonuniform pressure on 2-6-3 face.
10	Uniform pressure on 3-7-4 face.
11	Nonuniform pressure on 3-7-4 face.
12	Uniform pressure on 4-8-1 face.
13	Nonuniform pressure on 4-8-1 face.
20	Uniform shear force on 1-2-5 face in the 1⇒2⇒5 direction.
21	Nonuniform shear force on side 1-5-2.
22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
23	Nonuniform shear force on side 2-6-3.
24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
25	Nonuniform shear force on side 3-7-4.
26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strain and Stresses

Stresses and Strains are output at each integration point. For the case of large deformations, the stresses are the second Piola-Kirchhoff stresses and the strains are the Green strains.

1 = global xx strain
2 = global yy strain
3 = global zz strain
4 = global xy strain
5 = global yz strain
6 = global xz strain

Note: The stress and strain tensors represent a full 3-D tensor, although the kinematics of the element only allows a plane strain deformation where strain components 5 and 6 must be identical to zero. These components may become nonzero if transformations to a preferred orientation system are being used as defined in the COMPOSITE model definition option with a nonzero ply orientation angle. The associated stress components 5 and 6 in this system may become nonzero and with nonisotropic material behavior even the stress components 5 and 6 in the global system may become nonzero. The user should make sure that the layer build-up is such, that globally the plane strain deformation is maintained by good approximation, so that the averaged stress state of the element remains a plane strain stress state by good approximation, although such a stress state may have been violated on a layer by layer basis (e.g., by having one layer with some positive ply orientation angle and the next layer with the same negative ply orientation angle).

The PRINT ELEMENT or PRINT CHOICE options can be used to print out stress and strain results for specific integration points in the output file. The PRINT ELEMENT option is only valid for the first 28 integration points while the PRINT CHOICE option can be used for any integration point number (from 1 to a maximum of 2040). On the post file, stresses and strains can be obtained for the four integration points of each layer. When no layer is specified, post file results are presented for layer 1.

Interlaminar normal and shear stresses, as well as their directions, are output.

By using post code 501 or 511 (representing interlaminar normal and shear stresses, respectively), the interlaminar normal or shear stress can be written into the post file in the form of a stress tensor defined in the global coordinate directions. Mentat can be used to plot the principal directions of the stress tensor to show the magnitude and the direction of the stress, and their changes based on deformation.

Transformation

Any local set (u, v) can be used at any node.

Local Element Coordinate System

A local element coordinate system is used to facilitate the definition of anisotropic composite layered materials. The first element direction is the material layer direction. The second element direction is normal to the material layers pointing to the thickness direction. The third element direction is the cross products of the first and second element. The material preferred directions are then defined by a rotation of the element coordinate system about the second element direction with an orientation angle defined in the COMPOSITE model definition option in the Marc Volume C: Program Input manual.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 179. See Element 179 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux generated by dissipated plastic energy is specified with type 101.

Element 154

Quadrilateral, Axisymmetric, Eight-node Composite Element

This is a isoparametric, axisymmetric, eight-node composite element. Different material properties can be used for different layers within the element (see Figure 3-231).

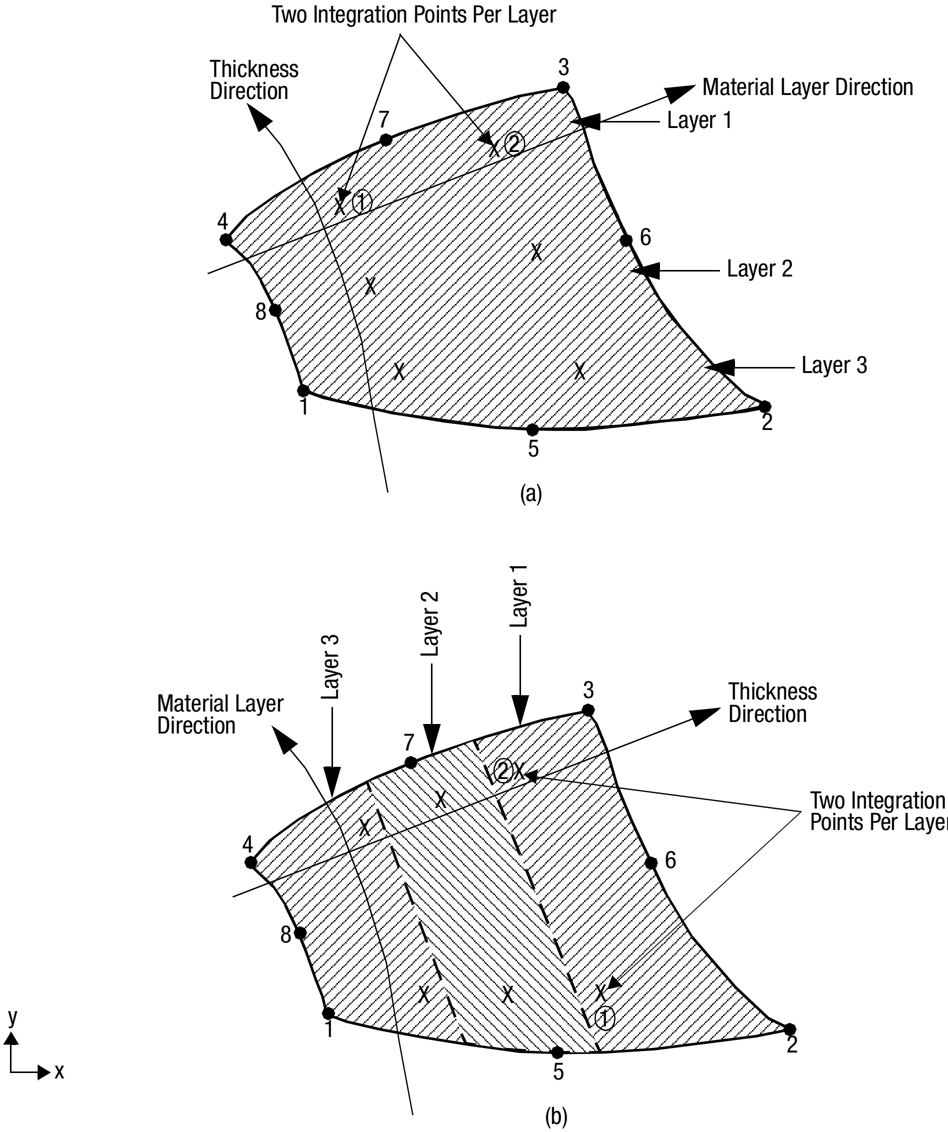


Figure 3-231 Typical Eight-node, Quadrilateral, Axisymmetric Composite Element

All constitutive equations available can be used for the element. To use the Mooney and Ogden nearly incompressible material model, the Updated Lagrange formulation must be used.

Integration

The number of continuum layers within an element is input via the COMPOSITE option. To ensure the stability of the element, a minimum number of two layers is required within the element. Each layer is assumed to be placed parallel to a pair of opposite element edges, so that the "thickness" direction is from one of the element edges to its opposite one. For instance (see Figure 3-231), if the layer is parallel to the 1, 2 and 3, 4 edges of the element, the "thickness" direction is from element edge 1, 2 to element edge 3, 4. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains two integration points (see Figure 3-231). On each layer, you must input, via the COMPOSITE option, the thickness (or percentage of the thickness) and the material set id. See Marc Volume C: Program Input for the COMPOSITE option. A maximum of 1020 layers can be used within each element.

Quick Reference

Type 154

Eight-node, isoparametric, quadrilateral, axisymmetric composite element.

Connectivity

Eight nodes per element. Node numbering of the element is the same as for element type 28.

Geometry

The isoparametric direction of layers is defined in the third field (`EGEOM3`), enter 1 or 2:

1 = Material layers are parallel to the 1, 2 and 3, 4 edges; the thickness direction is from the 1, 2 edge to the 3, 4 edge of the element (Figure 3-231 (a)).

2 = Material layers are parallel to the 1, 4 and 2, 3 edges; the thickness direction is from the 1, 4 edge to the 2, 3 edge of the element (Figure 3-231 (b)).

Coordinates

Two global coordinates in the z- and r-directions.

Degrees of Freedom

1 - u global z-direction displacement (axial)
2 - v global r-direction displacement (radial)

Distributed Loads

Surface Forces. Pressure and shear surface forces are available for this element as follows:

Load Type	Description
0	Uniform pressure on 1-5-2 face.
1	Nonuniform pressure on 1-5-2 face.
2	Uniform body force in x-direction.
3	Nonuniform body force in the x-direction.
4	Uniform body force in y-direction.

Load Type	Description
5	Nonuniform body force in the y-direction.
6	Uniform shear force in 1⇒5⇒2 direction on 1-5-2 face.
7	Nonuniform shear in 1⇒5⇒2 direction on 1-5-2 face.
8	Uniform pressure on 2-6-3 face.
9	Nonuniform pressure on 2-6-3 face.
10	Uniform pressure on 3-7-4 face.
11	Nonuniform pressure on 3-7-4 face.
12	Uniform pressure on 4-8-1 face.
13	Nonuniform pressure on 4-8-1 face.
20	Uniform shear force on 1-5-2 face in the 1⇒5⇒2 direction.
21	Nonuniform shear force on side 1-5-2.
22	Uniform shear force on side 2-6-3 in the 2⇒6⇒3 direction.
23	Nonuniform shear force on side 2-6-3.
24	Uniform shear force on side 3-7-4 in the 3⇒7⇒4 direction.
25	Nonuniform shear force on side 3-7-4.
26	Uniform shear force on side 4-8-1 in the 4⇒8⇒1 direction.
27	Nonuniform shear force on side 4-8-1.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For all nonuniform loads, the load magnitude is supplied via the FORCEM user subroutine.

Concentrated nodal loads must be the value of the load integrated around the circumference.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strain and Stresses

Stresses and Strains are output at each integration point. For the case of large deformations, the stresses are the second Piola-Kirchhoff stresses and the strains are the Green strains.

1 = global zz strain
2 = global rr strain
3 = global $\theta\theta$ strain
4 = global zr strain
5 = global rθ strain
6 = global θz strain

Note: The stress and strain tensors represent a full 3-D tensor, although the kinematics of the element only allows an axisymmetric deformation where strain components 5 and 6 must be identical to zero. These components may become nonzero if transformations to a preferred orientation system are being used as defined in the COMPOSITE model definition option with a nonzero ply orientation angle. The associated stress components 5 and 6 in this system may become nonzero and with nonisotropic material behavior even the stress components 5 and 6 in the global system may become nonzero. The user should make sure that the layer build-up is such, that globally the axisymmetric deformation is maintained by good approximation, so that the averaged stress state of the element remains a plane strain stress state by good approximation, although such a stress state may have been violated on a layer by layer basis (e.g., by having one layer with some positive ply orientation angle and the next layer with the same negative ply orientation angle).

The PRINT ELEMENT or PRINT CHOICE options can be used to print out stress and strain results for specific integration points in the output file. The PRINT ELEMENT option is only valid for the first 28 integration points while the PRINT CHOICE option can be used for any integration point number (from 1 to a maximum of 2040). On the post file, stresses and strains can be obtained for the four integration points of each layer. When no layer is specified, post file results are presented for layer 1.

Interlaminar normal and shear stresses, as well as their directions, are output.

By using post code 501 or 511 (representing interlaminar normal and shear stresses, respectively), the interlaminar normal or shear stress can be written into the post file in the form of a stress tensor defined in the global coordinate directions. Mentat can be used to plot the principal directions of the stress tensor to show the magnitude and the direction of the stress, and their changes based on deformation.

Transformation

Any local set (z, r) can be used at any node.

Local Element Coordinate System

A local element coordinate system is used to facilitate the definition of anisotropic composite layered materials. The first element direction is the material layer direction. The second element direction is normal to the material layers pointing to the thickness direction. The third element direction is the cross products of the first and second element. The material preferred directions are then defined by a rotation of the element coordinate system about the second element direction with an orientation angle defined in the COMPOSITE model definition option in the Marc Volume C: Program Input manual.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 180. See Element 180 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux generated by dissipated plastic energy is specified with type 101.

Element 155

Plane Strain, Low-order, Triangular Element, Herrmann Formulations

This is a isoparametric, plane strain, 3 + 1-node low-order, triangular element with an additional pressure degree of freedom at each of the three corner nodes (see Figure 3-232). It is written for incompressible or nearly incompressible plane strain applications. The shape function for the center node is a bubble function. Therefore, the displacements and the coordinates for the element are linearly distributed along the element boundaries. The stiffness of this element is formed using three Gaussian integration points. The degrees of freedom of the center node are condensed out on the element level before the assembly of the global matrix.

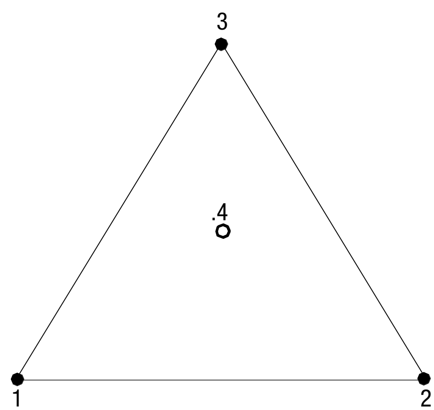


Figure 3-232 Form of Element 155

This element can be used for incompressible elasticity via total Lagrangian formulations or for rubber elasticity and elasto-plasticity via updated Lagrangian (F^eF^p) formulations. To activate large strain analysis via updated Lagrangian formulations, use either the LARGE STRAIN,2 parameter (see Marc Volume A: Theory and User Information and Marc Volume C: Program Input for more information).

Integration

Three integration points are used to correctly interpolate the cubic shape function.

Quick Reference

Type 155

3 + 1-node, isoparametric, plane strain, triangular element using Herrmann formulation. Written for incompressible or nearly incompressible applications.

Connectivity

Four nodes per element (see Figure 3-232). Node numbering for the first three nodes is the same as for element type 6; that is, counterclockwise at three corners. The fourth node is located at the element center.

Geometry

Thickness of the element stored in the first data field (EGEOM1). Default thickness is unity. Other fields are not used.

Coordinates

Two global coordinates in the x- and y-directions. Marc automatically calculates the coordinates of the fourth (center) node of the element.

Degrees of Freedom

Displacement output in global components is:

1	= u	
2	= v	
3	= σ_{kk}/E	= mean pressure variable (isotropic)
	= formula below (orthotropic)	

$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1 + \frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2 + \frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3 \right] \frac{9}{E_1 + E_2 + E_3}$$

	= -p	= negative pressure (for Mooney, Ogden, or Soil)
	= p / K	(for Herrmann elements using additive decomposition; K here is the effective bulk modulus)

Distributed Loads

Load types for distributed loads are as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force per unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element; magnitude supplied through the FORCEM user subroutine.
4	Nonuniform body force per unit volume in first coordinate direction; magnitude supplied through the FORCEM user subroutine.
5	Nonuniform body force per unit volume in second coordinate direction; magnitude supplied through the FORCEM user subroutine.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element; magnitude supplied through the FORCEM user subroutine.

Load Type	Description
8	Uniform pressure on 3-1 face of the element.
9	Nonuniform pressure on 3-1 face of the element; magnitude supplied through the FORCEM user subroutine.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strain and Stresses

Stresses and Strains are output at each integration point. For the case of large deformations, the stresses are the second Piola-Kirchhoff stresses and the strains are the Green strains.

1	= global xx strain	
2	= global yy strain	
3	= global zz strain	
4	= global xy strain	
5	= σ_{kk}/E	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)

$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1 + \frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2 + \frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3\right]\frac{9}{E_1+E_2+E_3}$$

If a 1 is entered in the 14th field of the 2nd data block of the POST option, the post file contains only one integration point for each element. The element stresses and strains in the point are the averaged results over three integration points of the element. This is to reduce the size of the post file.

Transformation

Any local set (u, v) can be used at any node.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available.

Element 156

Axisymmetric, Low-order, Triangular Element, Herrmann Formulations

This is an isoparametric, axisymmetric, 3 + 1-node, low-order, triangular element with an additional pressure degree of freedom at each of the three corner nodes (see Figure 3-233). It is written for incompressible or nearly incompressible axisymmetric applications. The shape function for the center node is a bubble function. Therefore, the displacements and the coordinates for the element are linearly distributed along the element boundaries. The stiffness of this element is formed using three Gaussian integration points. The degrees of freedom of the center node are condensed out on the element level before the assembly of the global matrix.

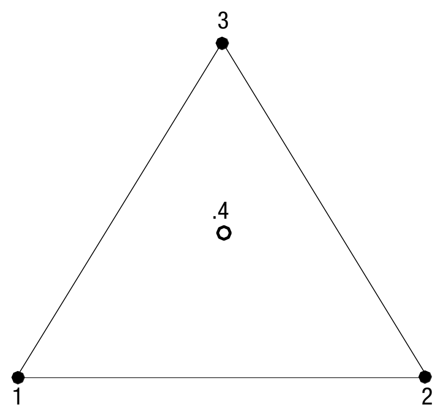


Figure 3-233 Form of Element 156

This element can be used for incompressible elasticity via total Lagrangian formulations or for rubber elasticity and elasto-plasticity via updated Lagrangian (F^eF^p) formulations.To activate large strain analysis via updated Lagrangian formulations, use either the LARGE STRAIN,2 parameter (see Marc Volume A: Theory and User Information and Marc Volume C: Program Input for more information).

Integration

Three integration points are used to correctly interpolate the cubic shape function.

Quick Reference

Type 156

3 + 1-node, isoparametric, axisymmetric, triangular element using Herrmann formulation. Written for incompressible or nearly incompressible applications.

Connectivity

Four nodes per element (see Figure 3-233). Node numbering for the first three nodes is the same as for element type 2; that is, counterclockwise at three corners. The fourth node is located at the element center.

Coordinates

Two global coordinates in the z- and r-directions. Marc automatically calculates the coordinates of the fourth (center) node of the element.

Degrees of Freedom

1	= u	
2	= v	
3	= σ_{kk}/E	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	= -p	= negative pressure (for Mooney, Ogden, or Soil)
	= p / K	(for Herrmann elements using additive decomposition; K here is the effective bulk modulus)

$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}} \sigma_1 + \frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}} \sigma_2 + \frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}} \sigma_3 \right] \frac{9}{E_1 + E_2 + E_3}$$

Distributed Loads

Load types for distributed loads are as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force per unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element; magnitude supplied through the FORCEM user subroutine.
4	Nonuniform body force per unit volume in first coordinate direction; magnitude supplied through the FORCEM user subroutine.
5	Nonuniform body force per unit volume in second coordinate direction; magnitude supplied through the FORCEM user subroutine.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element; magnitude supplied through the FORCEM user subroutine.
8	Uniform pressure on 3-1 face of the element.
9	Nonuniform pressure on 3-1 face of the element; magnitude supplied through the FORCEM user subroutine.

Load Type	Description
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

All pressures are positive when directed into the element. In addition, point loads can be applied at the nodes.

Concentrated loads applied at the nodes must be the value of the load integrated around the circumference.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strain and Stresses

Stresses and Strains are output at each integration point. For the case of large deformations, the stresses are the second Piola-Kirchhoff stresses and the strains are the Green strains.

1	= global zz strain	
2	= global rr strain	
3	= global $\theta\theta$ strain	
4	= global zr strain	
5	= σ_{kk}/E	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)

$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1+\frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2+\frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3\right]\frac{9}{E_1+E_2+E_3}$$

If a 1 is entered in the 14th field of the 2nd data block of the POST option, the post file contains only one integration point for each element. The element stresses and strains in the point are the averaged results over three integration points of the element. This is to reduce the size of the post file.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available.

Element 157

Three-dimensional, Low-order, Tetrahedron, Herrmann Formulations

This element is a three-dimensional, isoparametric, 4 + 1-node, low-order, tetrahedron with an additional pressure degree of freedom at each of the four corner nodes (see Figure 3-234). It is written for incompressible or nearly incompressible three-dimensional applications. The shape function for the center node is a bubble function. Therefore, the displacements and the coordinates for the element are linearly distributed along the element boundaries. The stiffness of this element is formed using four Gaussian integration points. The degrees of freedom of the center node are condensed out on the element level before the assembly of the global matrix.

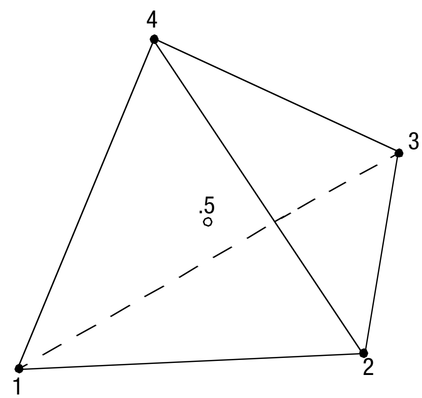


Figure 3-234 Form of Element 157

This element can be used for incompressible elasticity via total Lagrangian formulations or for rubber elasticity and elasto-plasticity via updated Lagrangian (F^eF^p) formulations.To activate large strain analysis via updated Lagrangian formulations use either the LARGE STRAIN,2 parameter (see Marc Volume A: Theory and User Information and Marc Volume C: Program Input for more information).

Integration

Four integration points are used to correctly interpolate the cubic shape function. For the mass matrix in volumetric loads, 16 integration points are used.

Quick Reference

Type 157

4 + 1-node, isoparametric, three-dimensional, tetrahedron using Herrmann formulation. Written for incompressible or nearly incompressible applications.

Connectivity

Five nodes per element (see Figure 3-234). Node numbering for the first four nodes is the same as for element type 134; that is, nodes 1, 2, 3, being the corners of the first face in counterclockwise order when viewed from inside the element and node 4 on the opposing vertex. The fifth node is located at the element center.

Coordinates

Three global coordinates in the x-, y- and z-directions. Marc automatically calculates the coordinates of the fifth (center) node of the element.

Degrees of Freedom

1	= u	
2	= v	
3	= w	
4	= σ_{kk}/E	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	= -p	= negative pressure (for Mooney, Ogden, or Soil)
	= p / K	(for Herrmann elements using additive decomposition; K here is the effective bulk modulus)

$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1 + \frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\Delta_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2 + \frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3 \right] \frac{9}{E_1 + E_2 + E_3}$$

Distributed Loads

Distributed loads chosen by value of `IBODY` are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3 face.
1	Nonuniform pressure on 1-2-3 face.
2	Uniform pressure on 1-2-4 face.
3	Nonuniform pressure on 1-2-4 face.
4	Uniform pressure on 2-3-4 face.
5	Nonuniform pressure on 2-3-4 face.
6	Uniform pressure on 1-3-4 face.
7	Nonuniform pressure on 1-3-4 face.
8	Uniform body force per unit volume in x direction.
9	Nonuniform body force per unit volume in x direction.
10	Uniform body force per unit volume in y direction.
11	Nonuniform body force per unit volume in y direction.
12	Uniform body force per unit volume in z direction.
13	Nonuniform body force per unit volume in z direction.

Load Type	Description
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strain and Stresses

Stresses and Strains are output at each integration point. For the case of large deformations, the stresses are the second Piola-Kirchhoff stresses and the strains are the Green strains.

1	= global xx strain	
2	= global yy strain	
3	= global zz strain	
4	= global xy strain	
5	= global yz strain	
6	= global zx strain	
7	= σ_{kk}/E	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	$H = \left[\dfrac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1 + \dfrac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\Delta_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2 + \dfrac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3}+\frac{\nu_{12}\nu_{23}}{E_1E_2}+\frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3 \right] \dfrac{9}{E_1+E_2+E_3}$	
	= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)

If a 1 is entered in the 14th field of the 2nd data block of the POST option, the post file contains only one integration point for each element. The element stresses and strains in the point are the averaged results over three integration points of the element. This is to reduce the size of the post file.

Transformation

Any local set (u, v, w) can be used at any node.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available.

Element 158

Three-node Membrane Element

Element 158 is a three-node, isoparametric, flat triangle written for membrane applications. As a membrane has no bending stiffness, the element is very unstable. The strains are constant throughout the element.

In general, you need more of these lower-order elements than the higher-order elements such as 200. Hence, use a fine mesh.

The stiffness of this element is formed using a single-point integration scheme.

All constitutive models can be used with this element.

This element is usually used with the LARGE DISP parameter, in which case the (tensile) initial stress stiffness increase the rigidity of the element.

Geometric Basis

The first two element base vectors lie in the plane of the three corner nodes. The first one (V_1) points from node 1 to node 2; the second one (V_2) is perpendicular to V_1 and lies on the same side as node 3. The third one (V_3) is determined such that V_1, V_2, and V_3 form a right-hand system (Figure 3-235).

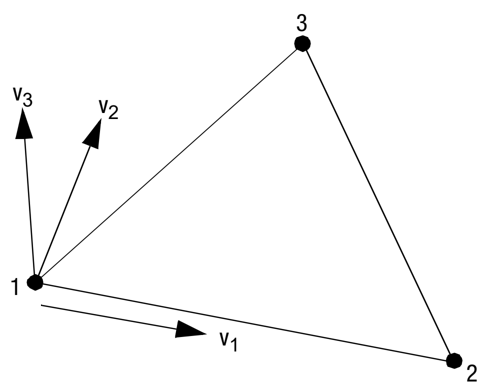


Figure 3-235 Form of Element 158

There is one integration point in the plane of the element at the centroid.

Quick Reference

Type 158

Three-node membrane element (straight edge) in three-dimensional space.

Connectivity

Three nodes per element (see Figure 3-235).

Geometry

The thickness is input in the first data field (EGEOM1). The other two data fields are not used.

Coordinates

Three global Cartesian coordinates x, y, z.

Degrees of Freedom

Three global degrees of freedom u, v, and w.

Tractions

Four distributed load types are available, depending on the load type definition:

Load Type	Description
1	Gravity load, proportional to surface area, in negative global z direction.
2	Pressure (load per unit area) positive when in direction of normal V_3.
3	Nonuniform gravity load, proportional to surface area, in negative global z direction (use the FORCEM user subroutine).
4	Nonuniform pressure, proportional to surface area, positive when in direction of normal V_3 (use the FORCEM user subroutine).
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains and Stresses

Output of stress (σ_{11}, σ_{22}, τ_{12}) and strain (ε_{11}, ε_{22}, γ_{12}) is in the local (V_1, V_2) directions defined above.

Transformation

Nodal degrees of freedom can be transformed to local degrees of freedom.

Output Points

Centroid.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 196. See Element 196 for a description of the conventions used for entering the flux and film data for this element.

Design Variables

The thickness can be considered a design variable for this element.

Element 159

Four-node, Thick Shell Element

Not available at this time.

Element 160

Arbitrary Plane Stress Piezoelectric Quadrilateral

Element type 160 is a four-node, isoparametric, arbitrary quadrilateral written for plane stress piezoelectric applications. The mechanical part of this element is based on element type 3. The electrical part of this element is added at the third degree of freedom. This element can be used in static, modal, transient, harmonic, and buckling analysis. The shear (or bending) characteristics can be improved by using alternative interpolation functions. This assumed strain procedure is flagged through the GEOMETRY option. A description of the piezoelectric capabilities is included in Marc Volume A: Theory and User Information .

The stiffness of this element is formed using four-point Gaussian integration.

This element can only be used with elastic constitutive relations. The electrical properties and the coupling between mechanical and electric behavior can be applied with the PIEZOELECTRIC option. The PIEZO parameter must be included.

Note:	To improve the bending characteristics of the element, the interpolation functions are modified for the assumed strain formulation.

Quick Reference

Type 160

Plane stress piezoelectric quadrilateral.

Connectivity

Four nodes per element. Node numbering must be counterclockwise (see Figure 3-236).

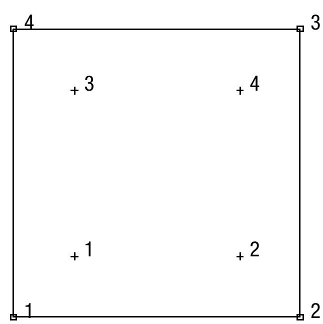


Figure 3-236 Plane Stress Quadrilateral

Geometry

The thickness is entered in the first data field (`EGEOM1`). Default thickness is one.

The second field is not used.

If a one is placed in the third field, the assumed strain formulation is activated.

Coordinates

Two coordinates in the global x- and y-direction.

Degrees of Freedom

1 = u (displacement in the global x-direction)

2 = v (displacement in the global y-direction)

3 = V (Electric Potential)

Distributed Loads

Load types for distributed loads are as follows:

Load Type	Description
* 0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force by unit volume in second coordinate direction.
* 3	Nonuniform pressure on 1-2 face of the element; magnitude supplied through the FORCEM user subroutine.
4	Nonuniform body force per unit volume in first coordinate direction; magnitude supplied through the FORCEM user subroutine.
5	Nonuniform body force per unit volume in second coordinate direction; magnitude supplied through the FORCEM user subroutine.
* 6	Uniform pressure on 2-3 face of the element.
* 7	Nonuniform pressure on 2-3 face of the element; magnitude supplied through the FORCEM user subroutine.
* 8	Uniform pressure on 3-4 face of the element.
* 9	Nonuniform pressure on 3-4 face of the element; magnitude supplied through the FORCEM user subroutine.
* 10	Uniform pressure on 4-1 face of the element.
* 11	Nonuniform pressure on 4-1 face of the element; magnitude supplied through the FORCEM user subroutine.
* 20	Uniform shear force on side 1-2 (positive from 1 to 2).
* 21	Nonuniform shear force on side 1-2; magnitude supplied through the FORCEM user subroutine
* 22	Uniform shear force on side 2-3 (positive from 2 to 3).
* 23	Nonuniform shear force on side 2-3; magnitude supplied through the FORCEM user subroutine.
* 24	Uniform shear force on side 3-4 (positive from 3 to 4).
* 25	Nonuniform shear force on side 3-4; magnitude supplied through the FORCEM user subroutine.
* 26	Uniform shear force on side 4-1 (positive from 4 to 1).

Load Type	Description
* 27	Nonuniform shear force on side 4-1; magnitude supplied through the FORCEM user subroutine.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cyclex/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

All pressures are positive when directed into the element. Load types shown with an asterisk (*) require the magnitude of the load to be entered as force per unit area. To prescribe these loads in force per unit length, add 70 to the load type. This is often useful in design optimization where the thickness changes, but it is desired that the applied force remain the same. In addition, point loads can be applied at the nodes.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Distributed Charges

Charge types for distributed charges are as follows:

Charge Type	Description
50	Uniform charge per unit area 1-2 side of the element.
51	Uniform charge per unit volume on whole element.
52	Uniform charge per unit volume on whole element.
53	Nonuniform charge per unit area on 1-2 side of the element; magnitude is given in the FLUX user subroutine.
54	Nonuniform charge per unit volume on whole element; magnitude is given in the FLUX user subroutine.
55	Nonuniform charge per unit volume on whole element; magnitude is given in the FLUX user subroutine.
56	Uniform charge per unit area on 2-3 side of the element.
57	Nonuniform charge per unit area on 2-3 side of the element; magnitude is given in the FLUX user subroutine.
58	Uniform charge per unit area on 3-4 side of the element.

Charge Type	Description
59	Nonuniform charge per unit area on 3-4 side of the element; magnitude is given in the FLUX user subroutine.
60	Uniform charge per unit area on 4-1 side of the element.
61	Nonuniform charge per unit area on 4-1 side of the element; magnitude is given in the FLUX user subroutine.

All distributed charges are positive when adding charge to the element. In addition, point charges can be applied at the nodes.

Output of Strains

Output of stains at the centroid of the element in global coordinates is:

$1 = \varepsilon_{xx}$
$2 = \varepsilon_{yy}$
$3 = \gamma_{xy}$

Note:	Although $\varepsilon_{zz} = \frac{-\nu}{E}(\sigma_{xx} + \sigma_{yy})$, it is not printed and is posted as 0 for isotropic materials. For Mooney or Ogden (TL formulation) Marc post code 49 provides the thickness strain for plane stress elements. See Marc Volume A: Theory and User Information , Chapter 12 Output Results, Element Information for von Mises intensity calculation for strain.

Output of Stresses

Same as for Output of Strains .

Output of Electric Properties

Two components of:

Electric field vector	E
Electric displacement vector	D

Transformation

The two global degrees of freedom can be transformed to local coordinates.

Tying

Use UFORMSN user subroutine.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Coupled Analysis

In a coupled thermal-piezoelectric analysis, the associated heat transfer element is type 39. See Element 39 for a description of the conventions used for entering the flux and film data for this element.

Assumed Strain

The assumed strain formulation is available to improve the in-plane bending behavior. Although this increases the stiffness assembly costs per element, it improves the accuracy.

Element 161

Arbitrary Plane Strain Piezoelectric Quadrilateral

Element type 161 is a four-node, isoparametric, arbitrary quadrilateral written for plane strain piezo-electric applications. The mechanical part of this element is based on element type 11. The electrical part of this element is added at the third degree of freedom. This element can be used in static, modal, transient, harmonic, and buckling analysis. The shear (or bending) characteristics can be improved by using alternative interpolation functions. This assumed strain procedure is flagged through the GEOMETRY option. A description of the piezoelectric capabilities is included in Marc Volume A: Theory and User Information.

The stiffness of this element is formed using four-point Gaussian integration.

This element can only be used with elastic constitutive relations. The electrical properties and the coupling between mechanical and electric behavior can be applied with the PIEZOELECTRIC option. The PIEZO parameter must be included.

Note:	To improve the bending characteristics of the element, the interpolation functions are modified for the assumed strain formulation.

Quick Reference

Type 161

Plane strain piezoelectric quadrilateral.

Connectivity

Four nodes per element. Node numbering must be counterclockwise (see Figure 3-237).

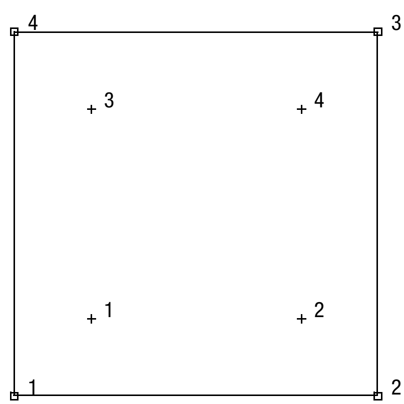


Figure 3-237 Integration Points for Element 161

Geometry

The thickness is entered in the first data field (EGEOM1). Default thickness is one.

If a one is placed in the third field, the assumed strain formulation is activated.

Coordinates

Two coordinates in the global x- and y-direction.

Degrees of Freedom

1 = u (displacement in the global x-direction)

2 = v (displacement in the global y-direction)

3 = V (Electric Potential)

Distributed Loads

Load types for distributed loads are as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 side of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force by unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 side of the element; magnitude supplied through the FORCEM user subroutine.
4	Nonuniform body force per unit volume in first coordinate direction; magnitude supplied through the FORCEM user subroutine.
5	Nonuniform body force per unit volume in second coordinate direction; magnitude supplied through the FORCEM user subroutine.
6	Uniform pressure on 2-3 side of the element.
7	Nonuniform pressure on 2-3 side of the element; magnitude supplied through the FORCEM user subroutine.
8	Uniform pressure on 3-4 side of the element.
9	Nonuniform pressure on 3-4 side of the element; magnitude supplied through the FORCEM user subroutine.
10	Uniform pressure on 4-1 side of the element.
11	Nonuniform pressure on 4-1 side of the element; magnitude supplied through the FORCEM user subroutine.
20	Uniform shear force on side 1-2 (positive from 1 to 2).
21	Nonuniform shear force on side 1-2; magnitude supplied through the FORCEM. user subroutine
22	Uniform shear force on side 2-3 (positive from 2 to 3).
23	Nonuniform shear force on side 2-3; magnitude supplied through the FORCEM user subroutine.
24	Uniform shear force on side 3-4 (positive from 3 to 4).
25	Nonuniform shear force on side 3-4; magnitude supplied through the FORCEM user subroutine.
26	Uniform shear force on side 4-1 (positive from 4 to 1).

Load Type	Description
27	Nonuniform shear force on side 4-1; magnitude supplied through the FORCEM user subroutine.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cyclex/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

All pressures are positive when directed into the element. In addition, point loads can be applied at the nodes.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Distributed Charges

Charge types for distributed charges are as follows:

Charge Type	Description
50	Uniform charge per unit area 1-2 side of the element.
51	Uniform charge per unit volume on whole element.
52	Uniform charge per unit volume on whole element.
53	Nonuniform charge per unit area on 1-2 side of the element; magnitude is given in the FLUX user subroutine.
54	Nonuniform charge per unit volume on whole element; magnitude is given in the FLUX user subroutine.
55	Nonuniform charge per unit volume on whole element; magnitude is given in the FLUX user subroutine.
56	Uniform charge per unit area on 2-3 side of the element.
57	Nonuniform charge per unit area on 2-3 side of the element; magnitude is given in the FLUX user subroutine.
58	Uniform charge per unit area on 3-4 side of the element.
59	Nonuniform charge per unit area on 3-4 side of the element; magnitude is given in the FLUX user subroutine.

Charge Type	Description
60	Uniform charge per unit area on 4-1 side of the element.
61	Nonuniform charge per unit area on 4-1 side of the element; magnitude is given in the FLUX user subroutine.

All distributed charges are positive when adding charge to the element. In addition, point charges can be applied at the nodes.

Output of Strains

Output of stains at the centroid of the element in global coordinates is:

$1 = \varepsilon_{xx}$
$2 = \varepsilon_{yy}$
$3 = \varepsilon_{\theta\theta}$
$4 = \gamma_{xy}$

Output of Stresses

Same as for Output of Strains .

Output of Electric Properties

Two components of:

Electric field vector	E
Electric displacement vector	D

Transformation

The two global degrees of freedom can be transformed to local coordinates.

Tying

Use UFORMSN user subroutine.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Coupled Analysis

In a coupled thermal-piezoelectric analysis, the associated heat transfer element is type 39. See Element 39 for a description of the conventions used for entering the flux and film data for this element.

Assumed Strain

The assumed strain formulation is available to improve the in-plane bending behavior. Although this increases the stiffness assembly costs per element, it improves the accuracy.

Element 162

Arbitrary Quadrilateral Piezoelectric Axisymmetric Ring

Element type 162 is a four-node, isoparametric, arbitrary quadrilateral written for axisymmetric piezoelectric applications. The mechanical part of this element is based on element type 10. The electrical part of this element is added at the third degree of freedom. This element can be used in static, modal, transient, harmonic, and buckling analysis. A description of the piezoelectric capabilities is included in Marc Volume A: Theory and User Information.

The stiffness of this element is formed using four-point Gaussian integration.

This element can only be used with elastic constitutive relations. The electrical properties and the coupling between mechanical and electric behavior can be applied with the PIEZOELECTRIC option. The PIEZO parameter must be included.

Quick Reference

Type 162

Axisymmetric, piezoelectric arbitrary ring with a quadrilateral cross section.

Connectivity

Four nodes per element. Node numbering must be counterclockwise (see Figure 3-238).

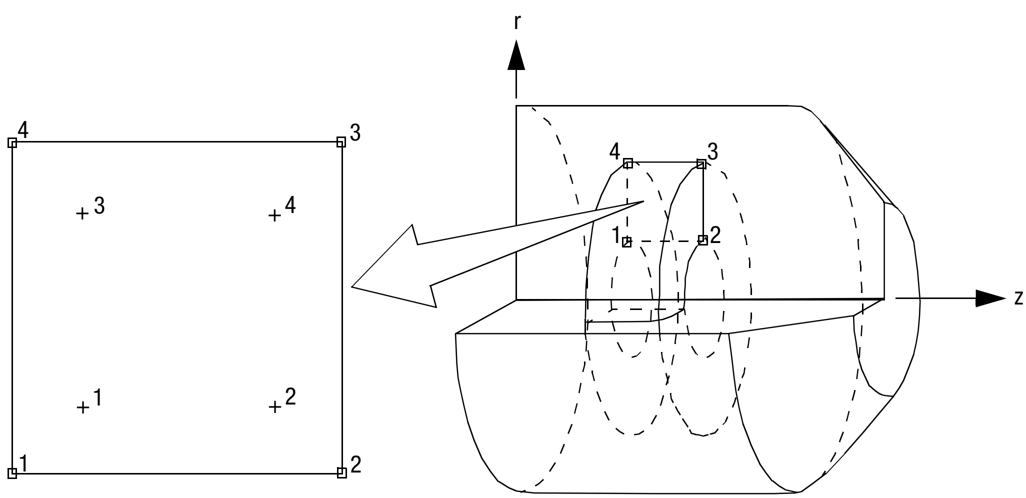


Figure 3-238 Gaussian Integration Points for Element Type 162

Geometry

Not applicable.

Coordinates

Two global coordinates in the global z- and r-direction.

Degrees of Freedom

1 = u (displacement in the global z-direction)

2 = v (displacement in the global r-direction)

3 = V (Electric Potential)

Distributed Loads

Load types for distributed loads are as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 side of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force by unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 side of the element; magnitude supplied through the FORCEM user subroutine.
4	Nonuniform body force per unit volume in first coordinate direction; magnitude supplied through the FORCEM user subroutine.
5	Nonuniform body force per unit volume in second coordinate direction; magnitude supplied through the FORCEM user subroutine.
6	Uniform pressure on 2-3 side of the element.
7	Nonuniform pressure on 2-3 side of the element; magnitude supplied through the FORCEM user subroutine.
8	Uniform pressure on 3-4 side of the element.
9	Nonuniform pressure on 3-4 side of the element; magnitude supplied through the FORCEM user subroutine.
10	Uniform pressure on 4-1 side of the element.
11	Nonuniform pressure on 4-1 side of the element; magnitude supplied through the FORCEM user subroutine.
20	Uniform shear force on side 1-2 (positive from 1 to 2).
21	Nonuniform shear force on side 1-2; magnitude supplied through the FORCEM. user subroutine
22	Uniform shear force on side 2-3 (positive from 2 to 3).
23	Nonuniform shear force on side 2-3; magnitude supplied through the FORCEM user subroutine.
24	Uniform shear force on side 3-4 (positive from 3 to 4).
25	Nonuniform shear force on side 3-4; magnitude supplied through the FORCEM user subroutine.
26	Uniform shear force on side 4-1 (positive from 4 to 1).
27	Nonuniform shear force on side 4-1; magnitude supplied through the FORCEM user subroutine.

Load Type	Description
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cyclex/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

All pressures are positive when directed into the element. In addition, point loads can be applied at the nodes. The magnitude of point loads must correspond to the load integrated around the circumference.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Distributed Charges

Charge types for distributed charges are as follows:

Charge Type	Description
50	Uniform charge per unit area 1-2 side of the element.
51	Uniform charge per unit volume on whole element.
52	Uniform charge per unit volume on whole element.
53	Nonuniform charge per unit area on 1-2 side of the element; magnitude is given in the FLUX user subroutine.
54	Nonuniform charge per unit volume on whole element; magnitude is given in the FLUX user subroutine.
55	Nonuniform charge per unit volume on whole element; magnitude is given in the FLUX user subroutine.
56	Uniform charge per unit area on 2-3 side of the element.
57	Nonuniform charge per unit area on 2-3 side of the element; magnitude is given in the FLUX user subroutine.
58	Uniform charge per unit area on 3-4 side of the element.
59	Nonuniform charge per unit area on 3-4 side of the element; magnitude is given in the FLUX user subroutine.

Charge Type	Description
60	Uniform charge per unit area on 4-1 side of the element.
61	Nonuniform charge per unit area on 4-1 side of the element; magnitude is given in the FLUX user subroutine.

All distributed charges are positive when adding charge to the element. In addition, point charges can be applied at the nodes. The magnitude of the point charge must correspond to the charge integrated around the circumference.

Output of Strains

Output of stains at the centroid of the element in global coordinates is:

1 = ε_{zz}
2 = ε_{rr}
3 = ε_{qq}
4 = γ_{rz}

Output of Stresses

Same as for Output of Strains .

Output of Electric Properties

Two components of:

Electric field vector	E
Electric displacement vector	D

Transformation

The two global degrees of freedom can be transformed to local coordinates.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Coupled Analysis

In a coupled thermal-piezoelectric analysis, the associated heat transfer element is type 40. See Element 40 for a description of the conventions used for entering the flux and film data for this element.

Element 163 Three-dimensional Piezoelectric Arbitrary Distorted Brick

Element type 163 is an eight-node, isoparametric, arbitrary hexahedral written for piezo-electric applications. The mechanical part of this element is based on element type 7. The electrical part of this element is added at the fourth degree of freedom. This element can be used in static, modal, transient, harmonic, and buckling analysis. The shear (or bending) characteristics can be improved by using alternative interpolation functions. This assumed strain procedure is flagged through the GEOMETRY option. A description of the piezoelectric capabilities is included in Marc Volume A: Theory and User Information.

The stiffness of this element is formed using eight-point Gaussian integration.

This element can only be used with elastic constitutive relations. The electrical properties and the coupling between mechanical and electric behavior can be applied with the PIEZOELECTRIC option. The PIEZO parameter must be included.

Note:	To improve the bending characteristics of the element, the interpolation functions are modified for the assumed strain formulation.

Quick Reference

Type 163

Three-dimensional, eight-node, first-order, isoparametric piezo-electric element (arbitrarily distorted piezoelectric brick).

Connectivity

Eight nodes per element. Node numbering must follow the scheme below (see Figure 3-239).

Nodes 1, 2, 3, and 4 are corners of one face, given in counterclockwise order when viewed from inside the element. Node 5 has the same edge as node 1. Node 6 has the same edge as node 2. Node 7 has the same edge as node 3. Node 8 has the same edge as node 4.

Geometry

If a one is placed in the third field, the assumed strain formulation is activated.

Coordinates

Three coordinates in the global x-, y-, and z-direction.

Degrees of Freedom

1 = u (displacement in the global x-direction)
2 = v (displacement in the global y-direction)
3 = w (displacement in the global z-direction)
4 = V (Electric Potential)

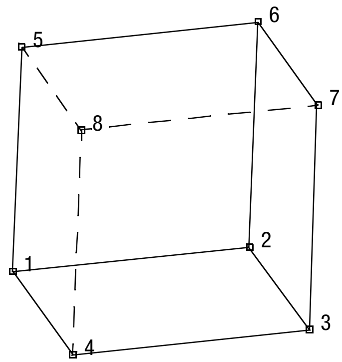


Figure 3-239 Arbitrarily Distorted Cube

Distributed Loads

Load types for distributed loads are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3-4 face.
1	Nonuniform pressure on 1-2-3-4 face; magnitude supplied through the FORCEM user subroutine.
2	Uniform body force per unit volume in -z-direction.
3	Nonuniform body force per unit volume (e.g., centrifugal force); magnitude and direction supplied through the FORCEM user subroutine.
4	Uniform pressure on 6-5-8-7 face.
5	Nonuniform pressure on 6-5-8-7 face (FORCEM user subroutine).
6	Uniform pressure on 2-1-5-6 face.
7	Nonuniform pressure on 2-1-5-6 face (FORCEM user subroutine).
8	Uniform pressure on 3-2-6-7 face.
9	Nonuniform pressure on 3-2-6-7 face (FORCEM user subroutine).
10	Uniform pressure on 4-3-7-8 face.
11	Nonuniform pressure on 4-3-7-8 face (FORCEM user subroutine).
12	Uniform pressure on 1-4-8-5 face.
13	Nonuniform pressure on 1-4-8-5 face (FORCEM user subroutine).
20	Uniform pressure on 1-2-3-4 face.
21	Nonuniform load on 1-2-3-4 face; magnitude and direction supplied in the FORCEM user subroutine.
22	Uniform body force per unit volume in -z-direction.
23	Nonuniform body force per unit volume (e.g., centrifugal force); magnitude and direction supplied through the FORCEM user subroutine.

Load Type	Description
24	Uniform pressure on 6-5-8-7 face.
25	Nonuniform load on 6-5-8-7 face; magnitude and direction supplied in the FORCEM user subroutine.
26	Uniform pressure on 2-1-5-6 face.
27	Nonuniform load on 2-1-5-6 face; magnitude and direction supplied in the FORCEM user subroutine.
28	Uniform pressure on 3-2-6-7 face.
29	Nonuniform load on 3-2-6-7 face; magnitude and direction supplied in the FORCEM user subroutine.
30	Uniform pressure on 4-3-7-8 face.
31	Nonuniform load on 4-3-7-8 face; magnitude and direction supplied in the FORCEM user subroutine.
32	Uniform pressure on 1-4-8-5 face.
33	Nonuniform load on 1-4-8-5 face; magnitude and direction supplied in the FORCEM user subroutine.
40	Uniform shear 1-2-3-4 face in the 1-2 direction.
41	Nonuniform shear 1-2-3-4 face in the 1-2 direction.
42	Uniform shear 1-2-3-4 face in the 2-3 direction.
43	Nonuniform shear 1-2-3-4 face in the 2-3 direction.
48	Uniform shear 6-5-8-7 face in the 5-6 direction.
49	Nonuniform shear 6-5-8-7 face in the 5-6 direction.
50	Uniform shear 6-5-8-7 face in the 6-7 direction.
51	Nonuniform shear 6-5-8-7 face in the 6-7 direction.
52	Uniform shear 2-1-5-6 face in the 1-2 direction.
53	Nonuniform shear 2-1-5-6 face in the 1-2 direction.
54	Uniform shear 2-1-5-6 face in the 1-5 direction.
55	Nonuniform shear 2-1-5-6 face in the 1-5 direction.
56	Uniform shear 3-2-6-7 face in the 2-3 direction.
57	Nonuniform shear 3-2-6-7 face in the 2-3 direction.
58	Uniform shear 3-2-6-7 face in the 2-6 direction.
59	Nonuniform shear 2-3-6-7 face in the 2-6 direction.
60	Uniform shear 4-3-7-8 face in the 3-4 direction.
61	Nonuniform shear 4-3-7-8 face in the 3-4 direction.
62	Uniform shear 4-3-7-8 face in the 3-7 direction.
63	Nonuniform shear 4-3-7-8 face in the 3-7 direction.
64	Uniform shear 1-4-8-5 face in the 4-1 direction.
65	Nonuniform shear 1-4-8-5 face in the 4-1 direction.

Load Type	Description
66	Uniform shear 1-4-8-5 in the 1-5 direction.
67	Nonuniform shear 1-4-8-5 face in the 1-5 direction.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cyclex/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

All pressures are positive when directed into the element. In addition, point loads can be applied at the nodes.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Distributed Charges

Charge types for distributed charges are as follows:

Charge Type	Description
70	Uniform charge on 1-2-3-4 face.
71	Nonuniform charge on 1-2-3-4 face.
74	Uniform charge on 5-6-7-8 face.
75	Nonuniform charge on 5-6-7-8 face.
76	Uniform charge on 1-2-6-5 face.
77	Nonuniform charge on 1-2-6-5 face.
78	Uniform charge on 2-3-7-6 face.
79	Nonuniform charge on 2-3-7-6 face
80	Uniform charge on 3-4-8-7 face.
81	Nonuniform charge on 3-4-8-7 face.

Charge Type	Description
82	Uniform charge on 1-4-8-5 face.
83	Nonuniform charge on 1-4-8-5 face.

For all nonuniform distributed charges, the magnitude is supplied through the FLUX user subroutine. All distributed charges are positive when adding charge to the element. In addition, point charges can be applied at the nodes.

Output of Strains

Output of stains at the centroid of the element in global coordinates is:

$1 = \varepsilon_{xx}$
$2 = \varepsilon_{yy}$
$3 = \varepsilon_{zz}$
$4 = \gamma_{xy}$
$5 = \gamma_{yz}$
$6 = \gamma_{xz}$

Output of Stresses

Same as for Output of Strains .

Output of Electric Properties

Three components of:

Electric field vector	E
Electric displacement vector	D

Transformation

Standard transformation of three global degrees of freedom to local degrees of freedom.

Tying

No special tying available.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Coupled Analysis

In a coupled thermal-piezoelectric analysis, the associated heat transfer element is type 43. See Element 43 for a description of the conventions used for entering the flux and film data for this element.

Assumed Strain

The assumed strain formulation is available to improve the in-plane bending behavior. Although this increases the stiffness assembly costs per element, it improves the accuracy.

Note:	The element can be collapsed to a tetrahedron.

Element 164

Three-dimensional Four-node Piezo-Electric Tetrahedron

Element type 164 is a four-node, linear isoparametric three-dimensional tetrahedron written for piezo-electric applications. The mechanical part of this element is based on element type 134. The electrical part of this element is added at the fourth degree of freedom. This element can be used in static, modal, transient, harmonic, and buckling analysis. The element is integrated numerically using one point at the centroid of the element. A description of the piezo-electric capabilities is included in Marc Volume A: Theory and User Information.

This element can only be used with elastic constitutive relations. The electrical properties and the coupling between mechanical and electric behavior can be applied with the PIEZOELECTRIC option. The PIEZO parameter must be included.

Quick Reference

Type 164

Three-dimensional, four-node, first-order, isoparametric piezo-electric element (arbitrarily distorted piezo-electric tetrahedron).

Connectivity

Four nodes per element. Node numbering must follow the scheme below (see Figure 3-240).

> Nodes 1, 2, 3 are the corners of the first face, given in counterclockwise order when viewed from inside the element. Node 4 is on the opposing vertex. Note that in most normal cases, the elements are generated automatically via a preprocessor (such as a Mentat or a CAD program) so that you need not be concerned with the node numbering scheme.

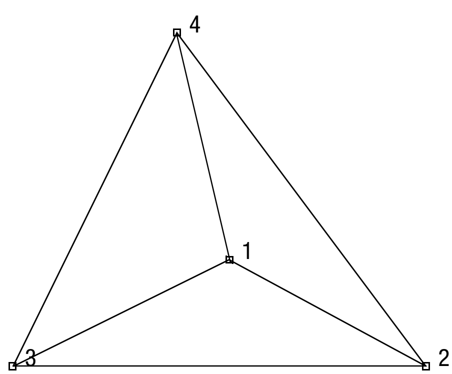


Figure 3-240 Form of Element 164

Geometry

Not required.

Coordinates

Three coordinates in the global x-, y-, and z-direction.

Degrees of Freedom

1 = u (displacement in the global x-direction)
2 = v (displacement in the global y-direction)
3 = w (displacement in the global z -direction)
4 = V (Electric Potential)

Distributed Loads

Load types for distributed loads are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3 face.
1	Nonuniform pressure on 1-2-3 face.
2	Uniform pressure on 1-2-4 face.
3	Nonuniform pressure on 1-2-4 face.
4	Uniform pressure on 2-3-4 face.
5	Nonuniform pressure on 2-3-4 face.
6	Uniform pressure on 1-3-4 face.
7	Nonuniform pressure on 1-3-4 face.
8	Uniform body force per unit volume in x direction.
9	Nonuniform body force per unit volume in x direction.
10	Uniform body force per unit volume in y direction.
11	Nonuniform body force per unit volume in y direction.
12	Uniform body force per unit volume in z direction.
13	Nonuniform body force per unit volume in z direction.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

All pressures are positive when directed into the element. For all nonuniform loads, the magnitude is supplied through the FORCEM user subroutine. In addition, point loads can be applied at the nodes.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Distributed Charges

Charge types for distributed charges are as follows:

Charge Type	Description
70	Uniform charge on 1-2-3 face.
71	Nonuniform charge on 1-2-3 face.
72	Uniform charge on 1-2-4 face.
73	Nonuniform charge on 1-2-4 face.
74	Uniform charge on 2-3-4 face.
75	Nonuniform charge on 2-3-4 face.
76	Uniform charge on 1-3-4 face.
77	Nonuniform charge on 1-3-4 face

For all nonuniform distributed charges, the magnitude is supplied through the FLUX user subroutine. All distributed charges are positive when adding charge to the element. In addition, point charges can be applied at the nodes.

Output of Strains

Output of stains at the centroid of the element in global coordinates is:

$1 = \varepsilon_{xx}$
$2 = \varepsilon_{yy}$
$3 = \varepsilon_{zz}$
$4 = \gamma_{xy}$
$5 = \gamma_{yz}$
$6 = \gamma_{xz}$

Output of Stresses

Same as for Output of Strains .

Output of Electric Properties

Three components of:

Electric field vector	E
Electric displacement vector	D

Transformation

Standard transformation of three global degrees of freedom to local degrees of freedom.

Tying

Use UFORMSN user subroutine.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Coupled Analysis

In a coupled thermal-piezoelectric analysis, the associated heat transfer element is type 135. See Element 135 for a description of the conventions used for entering the flux and film data for this element.

Element 165

Two-node Plane Strain Rebar Membrane Element

This element is isoparametric, plane strain, 2-node hollow line in which you can place single strain members such as reinforcing rods or cords (that is, rebars). The element is then used in conjunction with the 4-node plane strain continuum element (for example, element 11 or 80) to represent cord reinforced composite materials. The mesh for this rebar element can be different from its corresponding continuum element mesh. The INSERT option is used to enforce the compatibility between the two meshes. This technique allows the rebar and the filler to be represented accurately with respected to their stress distribution, so that separate constitutive theories can be used in each (for example, cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via the REBAR option or the REBAR user subroutine.

Integration

It is assumed that several "layers" of rebars are presented. The number of such "layers" is input by you via the REBAR option or, if the REBAR user subroutine is used, in the second element geometry field. A maximum number of five layers can be used within a rebar element. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains two integration points.

At each such integration point on each layer, you must input (via either the REBAR option or the REBAR user subroutine) the position, equivalent thickness (or, alternatively, spacing and area of cross-section), and orientation of the rebars. See Marc Volume C: Program Input for the REBAR option or Marc Volume D: User Subroutines and Special Routines for the REBAR user subroutine.

Quick Reference

Type 165

Two-node, isoparametric, plane strain rebar element to be used with 4-node plane strain continuum element.

Connectivity

Two nodes per element (see Figure 3-241).

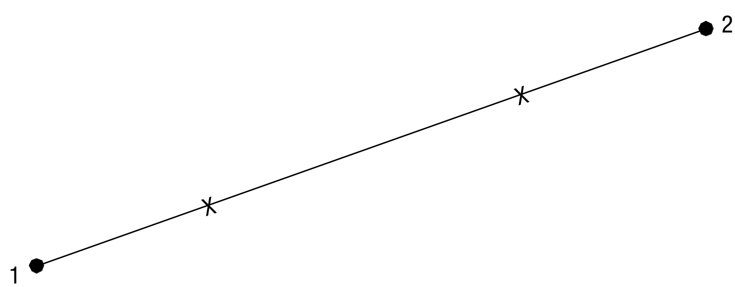


Figure 3-241 Two-node Plane Strain Rebar: Two Gauss Points per Layer

Geometry

Element thickness (in z-direction) in first field. Default thickness is unity. Note, this should not be confused with the "thickness" concept associated with rebar layers.

The number of rebar layers are defined using the REBAR model definition option.

Coordinates

Two global coordinates x- and y-directions.

Degrees of Freedom

Displacement output in global components is as follows:

1 - u
2 - v

Tractions

No distributed loads are available. Loads are applied only to corresponding 4-node plane strain elements (e.g., element types 11 or 80).

Output of Stress and Strain

One stress and one strain are output at each integration point – the axial rebar value.

By using post code 471 and 481 (representing Second Piola Kirchhoff stress in undeformed configuration and Cauchy stress in deformed configuration, respectively), the rebar stress can be written into the post file in the form of a stress tensor defined in the global coordinate directions. Mentat can be used to plot the principal directions of the stress tensor to show the magnitude of rebar stress, rebar orientation, and their changes based on deformation.

Transformation

Any local set (u, v) can be used at any node.

Special Consideration

Either the REBAR option or the REBAR user subroutine is needed to input the position, size, and orientation of the rebars.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Element 166

Two-node Axisymmetric Rebar Membrane Element

This element is isoparametric, axisymmetric, 2-node hollow line in which you can place single strain members such as reinforcing rods or cords (that is, rebars). The element is then used in conjunction with the 4-node axisymmetric continuum element (for example, element 10 or 82) to represent cord reinforced composite materials. The mesh for this rebar element can be different from its corresponding continuum element mesh. The INSERT option is used to enforce the compatibility between the two meshes. This technique allows the rebar and the filler to be represented accurately with respected to their stress distribution, so that separate constitutive theories can be used in each (for example, cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via the REBAR option or the REBAR user subroutine.

Integration

It is assumed that several "layers" of rebars are presented. The number of such "layers" is input by you via the REBAR option or, if the REBAR user subroutine is used, in the second element geometry field. A maximum number of five layers can be used within a rebar element. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains two integration points.

At each such integration point on each layer, you must input (via either the REBAR option or the REBAR user subroutine) the position, equivalent thickness (or, alternatively, spacing and area of cross-section), and orientation of the rebars. See Marc Volume C: Program Input for the REBAR option or Marc Volume D: User Subroutines and Special Routines for the REBAR user subroutine.

Quick Reference

Type 166

Two-node, isoparametric, axisymmetric rebar element to be used with 4-node axisymmetric continuum element.

Connectivity

Two nodes per element (see Figure 3-242).

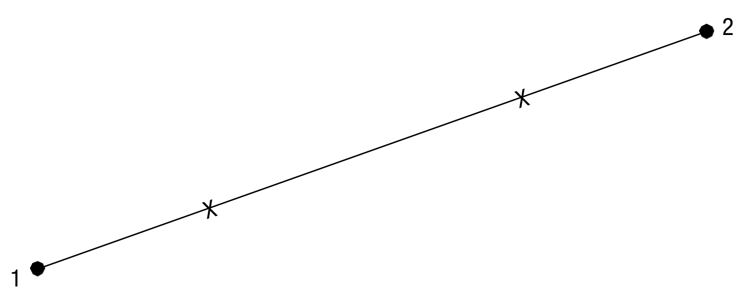


Figure 3-242 Two-node Axisymmetric Rebar: Two Gauss Points per Layer

Geometry

The number of rebar layers are defined using the REBAR model definition option.

Coordinates

Two global coordinates z- and r-directions.

Degrees of Freedom

Displacement output in global components is as follows:

1 - axial displacement in z-direction
2 - radial displacement in r-direction

Tractions

No distributed loads are available. Loads are applied only to corresponding 4-node axisymmetric elements (e.g., element types 10 or 82).

Output of Stress and Strain

One stress and one strain are output at each integration point – the axial rebar value.

By using post code 471 and 481 (representing Second Piola Kirchhoff stress in undeformed configuration and Cauchy stress in deformed configuration, respectively), the rebar stress can be written into the post file in the form of a stress tensor defined in the global coordinate directions. Mentat can be used to plot the principal directions of the stress tensor to show the magnitude of rebar stress, rebar orientation, and their changes based on deformation.

Transformation

Any local set (u, v) can be used at any node.

Special Consideration

Either the REBAR option or the REBAR user subroutine is needed to input the position, size, and orientation of the rebars.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Element 167 Two-node Axisymmetric Rebar Membrane Element with Twist

This element is isoparametric, axisymmetric, 2-node hollow line with twist degrees of freedom in which you can place single strain members such as reinforcing rods or cords (that is, rebars). The element is then used in conjunction with the 4-node axisymmetric continuum element with twist (for example, element 20 or 83) to represent cord reinforced composite materials. The mesh for this rebar element can be different from its corresponding continuum element mesh. The INSERT option is used to enforce the compatibility between the two meshes. This technique allows the rebar and the filler to be represented accurately with respected to their stress distribution, so that separate constitutive theories can be used in each (for example, cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via the REBAR option or the REBAR user subroutine.

Integration

It is assumed that several “layers” of rebars are presented. The number of such “layers” is input by you via the REBAR option or, if the REBAR user subroutine is used, in the second element geometry field. A maximum number of five layers can be used within a rebar element. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains two integration points.

At each such integration point on each layer, you must input (via either the REBAR option or the REBAR user subroutine) the position, equivalent thickness (or, alternatively, spacing and area of cross-section), and orientation of the rebars. See Marc Volume C: Program Input for the REBAR option or Marc Volume D: User Subroutines and Special Routines for the REBAR user subroutine.

Quick Reference

Type 167

Two-node, isoparametric, axisymmetric rebar element with twist degrees of freedom to be used with 4-node axisymmetric continuum element with twist.

Connectivity

Two nodes per element (see Figure 3-243).

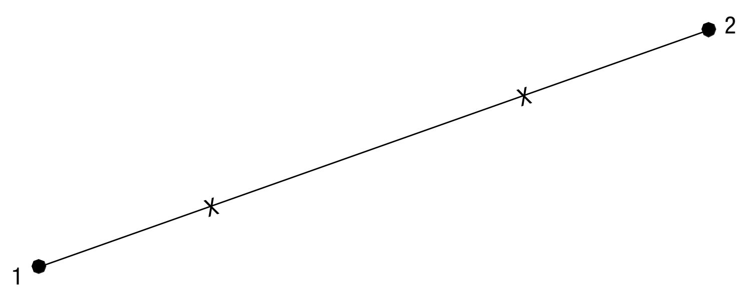


Figure 3-243 Two-node Axisymmetric Rebar with Twist: Two Gauss Points per Layer

Geometry

The number of rebar layers are defined using the REBAR model definition option.

Coordinates

Two global coordinates z- and r-directions.

Degrees of Freedom

Displacement output in global components is as follows:

1 - axial displacement in z-direction
2 - radial displacement in r-direction
3 - angular rotation about symmetric axis (in radians)

Tractions

No distributed loads are available. Loads are applied only to corresponding 4-node axisymmetric with twist elements (e.g., element types 20 or 83).

Output of Stress and Strain

One stress and one strain are output at each integration point – the axial rebar value.

By using post code 471 and 481 (representing Second Piola Kirchhoff stress in undeformed configuration and Cauchy stress in deformed configuration, respectively), the rebar stress can be written into the post file in the form of a stress tensor defined in the global coordinate directions. Mentat can be used to plot the principal directions of the stress tensor to show the magnitude of rebar stress, rebar orientation, and their changes based on deformation.

Transformation

Any local set (u, v) can be used at any node.

Special Consideration

Either the REBAR option or the REBAR user subroutine is needed to input the position, size, and orientation of the rebars.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Element 168

Three-node Plane Strain Rebar Membrane Element

This element is isoparametric, plane strain, 3-node hollow line in which you can place single strain members such as reinforcing rods or cords (that is, rebars). The element is then used in conjunction with the 8-node plane strain continuum element (for example, element 27 or 32) to represent cord reinforced composite materials. The mesh for this rebar element can be different from its corresponding continuum element mesh. The INSERT option is used to enforce the compatibility between the two meshes. This technique allows the rebar and the filler to be represented accurately with respected to their stress distribution, so that separate constitutive theories can be used in each (for example, cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via the REBAR option or the REBAR user subroutine.

Integration

It is assumed that several "layers" of rebars are presented. The number of such "layers" is input by you via the REBAR option or, if the REBAR user subroutine is used, in the second element geometry field. A maximum number of five layers can be used within a rebar element. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains two integration points.

At each such integration point on each layer, you must input (via either the REBAR option or the REBAR user subroutine) the position, equivalent thickness (or, alternatively, spacing and area of cross-section), and orientation of the rebars. See Marc Volume C: Program Input for the REBAR option or Marc Volume D: User Subroutines and Special Routines for the REBAR user subroutine.

Quick Reference

Type 168

Three-node, isoparametric, plane strain rebar element to be used with 8-node plane strain continuum element.

Connectivity

Three nodes per element (see Figure 3-244).

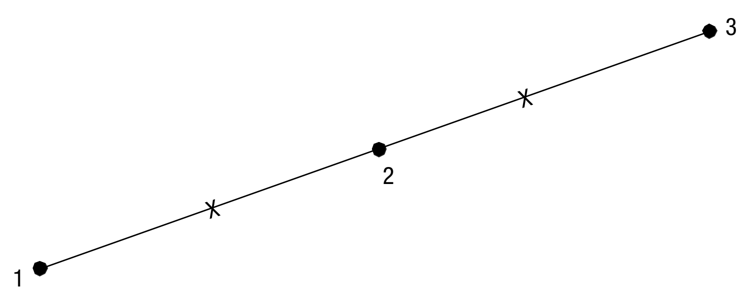


Figure 3-244 Three-node Plane Strain Rebar: Two Gauss Points per Layer

Geometry

Element thickness (in z-direction) in first field. Default thickness is unity. Note, this should not be confused with the "thickness" concept associated with rebar layers.

The number of rebar layers are defined using the REBAR model definition option.

Coordinates

Two global coordinates x- and y-directions.

Degrees of Freedom

Displacement output in global components is as follows:

1 - u
2 - v

Tractions

No distributed loads are available. Loads are applied only to corresponding 4-node plane strain elements (e.g., element types 27 or 32).

Output of Stress and Strain

One stress and one strain are output at each integration point – the axial rebar value.

By using post code 471 and 481 (representing Second Piola Kirchhoff stress in undeformed configuration and Cauchy stress in deformed configuration, respectively), the rebar stress can be written into the post file in the form of a stress tensor defined in the global coordinate directions. Mentat can be used to plot the principal directions of the stress tensor to show the magnitude of rebar stress, rebar orientation, and their changes based on deformation.

Transformation

Any local set (u, v) can be used at any node.

Special Consideration

Either the REBAR option or the REBAR user subroutine is needed to input the position, size, and orientation of the rebars.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Element 169 — Three-node Axisymmetric Rebar Membrane Element

This element is isoparametric, axisymmetric, 3-node hollow line in which you can place single strain members such as reinforcing rods or cords (that is, rebars). The element is then used in conjunction with the 8-node axisymmetric continuum element (for example, element 28 or 33) to represent cord reinforced composite materials. The mesh for this rebar element can be different from its corresponding continuum element mesh. The INSERT option is used to enforce the compatibility between the two meshes. This technique allows the rebar and the filler to be represented accurately with respected to their stress distribution, so that separate constitutive theories can be used in each (for example, cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via the REBAR option or the REBAR user subroutine.

Integration

It is assumed that several “layers” of rebars are presented. The number of such “layers” is input by you via the REBAR option or, if the REBAR user subroutine is used, in the second element geometry field. A maximum number of five layers can be used within a rebar element. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains two integration points.

At each such integration point on each layer, you must input (via either the REBAR option or the REBAR user subroutine) the position, equivalent thickness (or, alternatively, spacing and area of cross-section), and orientation of the rebars. See Marc Volume C: Program Input for the REBAR option or Marc Volume D: User Subroutines and Special Routines for the REBAR user subroutine.

Quick Reference

Type 169

Three-node, isoparametric, axisymmetric rebar element to be used with 8-node axisymmetric continuum element.

Connectivity

Three nodes per element (see Figure 3-245).

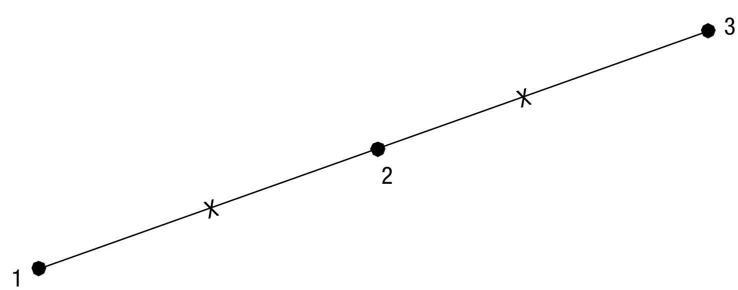


Figure 3-245 Three-node Axisymmetric Rebar: Two Gauss Points per Layer

Geometry

The number of rebar layers are defined using the REBAR model definition option.

Coordinates

Two global coordinates z- and r-directions.

Degrees of Freedom

Displacement output in global components is as follows:

1 - axial displacement in z-direction
2 - radial displacement in r-direction

Tractions

No distributed loads are available. Loads are applied only to corresponding 4-node axisymmetric elements (e.g., element types 28 or 33).

Output of Stress and Strain

One stress and one strain are output at each integration point – the axial rebar value.

By using post code 471 and 481 (representing Second Piola Kirchhoff stress in undeformed configuration and Cauchy stress in deformed configuration, respectively), the rebar stress can be written into the post file in the form of a stress tensor defined in the global coordinate directions. Mentat can be used to plot the principal directions of the stress tensor to show the magnitude of rebar stress, rebar orientation, and their changes based on deformation.

Transformation

Any local set (u, v) can be used at any node.

Special Consideration

Either the REBAR option or the REBAR user subroutine is needed to input the position, size, and orientation of the rebars.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Element 170 Three-node Axisymmetric Rebar Membrane Element with Twist

This element is isoparametric, axisymmetric, 3-node hollow line with twist degrees of freedom in which you can place single strain members such as reinforcing rods or cords (that is, rebars). The element is then used in conjunction with the 8-node axisymmetric continuum element with twist (for example, element 66 or 67) to represent cord reinforced composite materials. The mesh for this rebar element can be different from its corresponding continuum element mesh. The INSERT option is used to enforce the compatibility between the two meshes. This technique allows the rebar and the filler to be represented accurately with respected to their stress distribution, so that separate constitutive theories can be used in each (for example, cracking concrete and yield rebar). The position, size, and orientation of the rebars are input either via the REBAR option or the REBAR user subroutine.

Integration

It is assumed that several “layers” of rebars are presented. The number of such “layers” is input by you via the REBAR option or, if the REBAR user subroutine is used, in the second element geometry field. A maximum number of five layers can be used within a rebar element. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains two integration points.

At each such integration point on each layer, you must input (via either the REBAR option or the REBAR user subroutine) the position, equivalent thickness (or, alternatively, spacing and area of cross-section), and orientation of the rebars. See Marc Volume C: Program Input for the REBAR option or Marc Volume D: User Subroutines and Special Routines for the REBAR user subroutine.

Quick Reference

Type 170

Three-node, isoparametric, axisymmetric rebar element with twist degrees of freedom to be used with 8-node axisymmetric continuum element with twist.

Connectivity

Three nodes per element (see Figure 3-246).

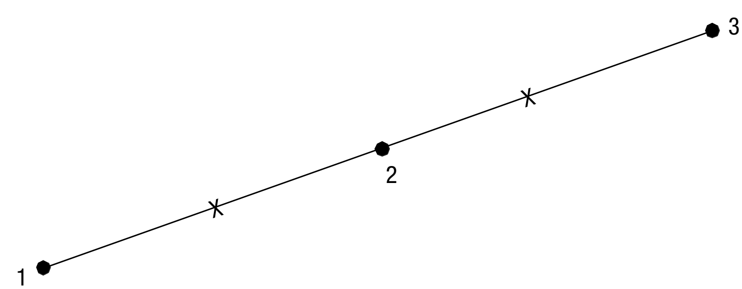


Figure 3-246 Three-node Axisymmetric Rebar with Twist: Two Gauss Points per Layer

Geometry

The number of rebar layers are defined using the REBAR model definition option.

Coordinates

Two global coordinates z- and r-directions.

Degrees of Freedom

Displacement output in global components is as follows:

1 - axial displacement in z-direction
2 - radial displacement in r-direction
3 - angular rotation about symmetric axis (in radians)

Tractions

No distributed loads are available. Loads are applied only to corresponding 4-node axisymmetric with twist elements (e.g., element types 66 or 67).

Output of Stress and Strain

One stress and one strain are output at each integration point – the axial rebar value.

By using post code 471 and 481 (representing Second Piola Kirchhoff stress in undeformed configuration and Cauchy stress in deformed configuration, respectively), the rebar stress can be written into the post file in the form of a stress tensor defined in the global coordinate directions. Mentat can be used to plot the principal directions of the stress tensor to show the magnitude of rebar stress, rebar orientation, and their changes based on deformation.

Transformation

Any local set (u, v) can be used at any node.

Special Consideration

Either the REBAR option or the REBAR user subroutine is needed to input the position, size, and orientation of the rebars.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Element 171

Two-node 2-D Cavity Surface Element

Element 171 can be used to define cavity surfaces for cavity volume calculation purposes. It can be used in both plane strain and plane stress cases. This element does not contribute to the stiffness matrix of the system. No material properties are needed. It does not undergo any deformation except if it is attached or glued to another element or surface.

Quick Reference

Type 171

Two-node 2-D cavity surface element

Connectivity

Two nodes per element (see Figure 3-247).

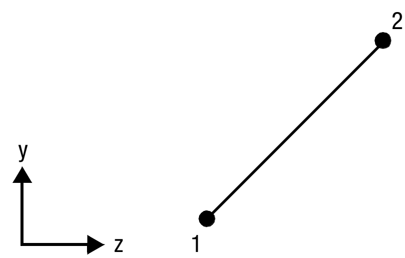


Figure 3-247 Two-node 2-D Cavity Surface Element

Geometry

The thickness is stored in the first data field (EGEOM1). Default thickness is one.

Coordinates

Two global coordinates x and y.

Degrees of Freedom

1 = u (displacement in the global x-direction)
2 = v (displacement in the global y-direction)

Tractions

This element cannot be loaded.

The value of IBODY is always zero for the element. This IBODY is used to calculate IBODY_CAVITY, which associates the element with a specific cavity id and defines the type of cavity loading. (See Cavity Pressure Loading in Marc Volume A: Theory and User Information , Chapter 9) for details.

Output of Stress and Strains

No element output available for this element.

Transformation

Two global degrees of freedom can be transformed to local coordinates.

Element 172

Two-node Axisymmetric Cavity Surface Element

Element 172 can be used to define cavity surfaces for cavity volume calculation purposes for axisymmetric problems. This element does not contribute to the stiffness matrix of the system. No material properties are needed. It does not undergo any deformation except if it is attached or glued to another element or surface.

Quick Reference

Type 172

Two-node axisymmetric cavity surface element.

Connectivity

Two nodes per element (see Figure 3-248).

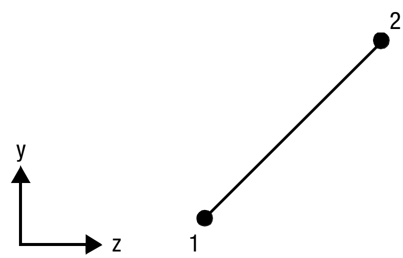


Figure 3-248 Two-node Axisymmetric Cavity Surface Element

Geometry

Not required for this element.

Coordinates

Two global coordinates z and r.

Degrees of Freedom

1 = u (axial displacement in the global z-direction)
2 = v (radial displacement in the global r-direction)

Tractions

This element cannot be loaded.

The value of `IBODY` is always zero for the element. This `IBODY` is used to calculate `IBODY_CAVITY`, which associates the element with a specific cavity id and defines the type of cavity loading. (See Cavity Pressure Loading in Marc Volume A: Theory and User Information , Chapter 9) for details.

Output of Stress and Strains

No element output available for this element.

Transformation

Two global degrees of freedom can be transformed to local coordinates.

Element 173

Three-node 3-D Cavity Surface Element

Element 173 can be used to define cavity surfaces for cavity volume calculation purposes for 3-D problems. This element does not contribute to the stiffness matrix of the system. No material properties are needed. It does not undergo any deformation except if it is attached or glued to another element or surface.

Quick Reference

Type 173

Three-node 3-D cavity surface element.

Connectivity

Three nodes per element (see Figure 3-249).

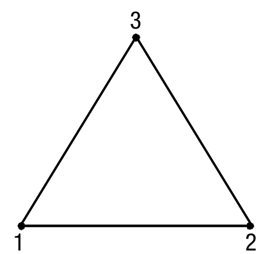


Figure 3-249 Three-node 3-D Cavity Surface Element

Geometry

Not required for this element.

Coordinates

Three global coordinates x, y, and z.

Degrees of Freedom

1 = u (displacement in the global x-direction)
2 = v (displacement in the global y-direction)
3 = w (displacement in the global z-direction)

Tractions

This element cannot be loaded.

The value of `IBODY` is always two for the element. This `IBODY` is used to calculate `IBODY_CAVITY`, which associates the element with a specific cavity id and defines the type of cavity loading. (See Cavity Pressure Loading in Marc Volume A: Theory and User Information , Chapter 9) for details.

Output of Stress and Strains

No element output available for this element.

Transformation

Three global degrees of freedom can be transformed to local coordinates.

Element 174

Four-node 3-D Cavity Surface Element

Element 174 can be used to define cavity surfaces for cavity volume calculation purposes for 3-D problems. This element does not contribute to the stiffness matrix of the system. No material properties are needed. It does not undergo any deformation except if it is attached or glued to another element or surface.

Quick Reference

Type 174

Four-node 3-D cavity surface element.

Connectivity

Four nodes per element (see Figure 3-250).

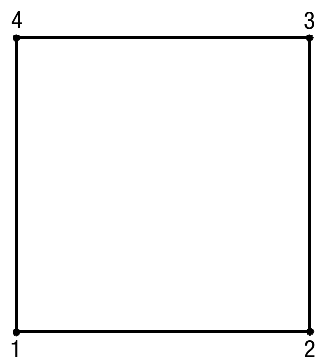


Figure 3-250 Four-node 3-D Cavity Surface Element

Geometry

Not required for this element.

Coordinates

Three global coordinates x, y, and z.

Degrees of Freedom

1 = u (displacement in the global x-direction)
2 = v (displacement in the global y-direction)
3 = w (displacement in the global z-direction)

Tractions

This element cannot be loaded.

The value of IBODY is always two for the element. This IBODY is used to calculate IBODY_CAVITY, which associates the element with a specific cavity id and defines the type of cavity loading. (See Cavity Pressure Loading in Marc Volume A: Theory and User Information , Chapter 9) for details.

Output of Stress and Strains

No element output available for this element.

Transformation

Three global degrees of freedom can be transformed to local coordinates.

Element 175

Three-dimensional, Eight-node Composite Brick Element (Heat Transfer Element)

This element is an isoparametric 8-node composite brick written for three-dimensional heat transfer applications. Different material properties can be used for different layers within the element (see Figure 3-251). This element can also be used for electrostatic applications.

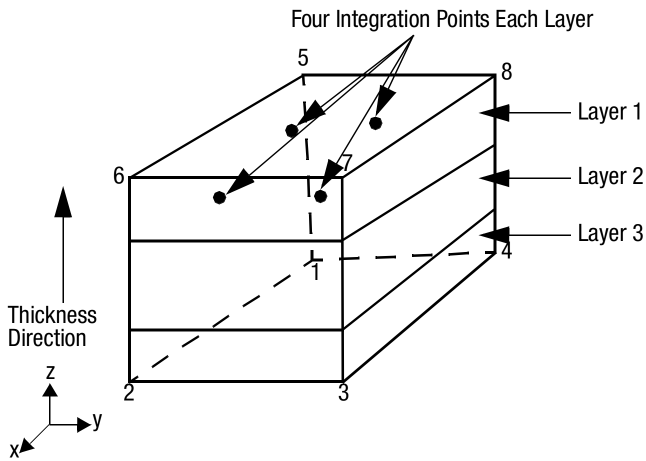


Figure 3-251 Typical 8-node 3-D Composite Heat Transfer Element

All heat transfer capabilities are currently supported for the element, except latent heat, thermal contact via the CONRAD GAP model definition option and fluid channel via the CHANNEL model definition option. Also, the CENTROID parameter cannot be used for this element.

Integration

The number of continuum layers within an element is input via the COMPOSITE option. To ensure the stability of the element, a minimum number of two layers is required within the element. Each layer is assumed to be placed parallel to a pair of opposite element faces, so that the "thickness" direction is from one of the element faces to its opposite one. For instance, if the layer is parallel to the 1, 2, 3, 4 and 5, 6, 7, 8 faces of the element, the "thickness" direction is from element face 1, 2, 3, 4 to element face 5, 6, 7, 8 (see Figure 3-251). The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains four integration points (see Figure 3-251). On each layer, you must input, via the COMPOSITE option, the thickness (or percentage of the thickness) and the material set id. See Marc Volume C: Program Input for the COMPOSITE option. A maximum of 510 layers can be used within each element. The mass matrix of this element is formed using eight-point Gaussian integration.

Quick Reference

Type 175

Eight-node, isoparametric composite brick element for heat transfer.

Connectivity

Eight nodes per element. Node numbering of the element is the same as for element type 43.

Geometry

The isoparametric direction of layers is defined in the third field (EGEOM3), enter 1, 2, or 3:

1	=	Material layers are parallel to the 1, 2, 3, 4 and 5, 6, 7, 8 faces; the thickness direction is from 1, 2, 3, 4 face to the 5, 6, 7, 8 face of the element.
2	=	Material layers are parallel to the 1, 4, 8, 5 and 2, 3, 7, 6 faces; the thickness direction is from 1, 4, 8, 5 face to the 2, 3, 7, 6 face of the element.
3	=	Material layers are parallel to the 2, 1, 5, 6 and 3, 4, 8, 7 faces; the thickness direction is from 2, 1, 5, 6 face to the 3, 4, 8, 7 face of the element.

If a nonzero value is entered in the fourth data field (EGEOM4), the temperatures at the integration points obtained from interpolation of nodal temperatures are constant throughout the element.

Coordinates

Three global coordinates in x-, y-, and z-directions.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)

Fluxes

Fluxes are distributed according to the appropriate selection of a value of IBODY. Surface fluxes are positive when heat energy is added to the element and are evaluated using a 4-point integration scheme, where the integration points have the same location as the nodal points.

Load Type (IBODY)	Description
0	Uniform flux on 1-2-3-4 face.
1	Nonuniform surface flux (supplied via the FLUX user subroutine) on 1-2-3-4 face.
2	Uniform volumetric flux.
3	Nonuniform volumetric flux (with the FLUX user subroutine).
4	Uniform flux on 5-6-7-8 face.
5	Nonuniform surface flux on 5-6-7-8 face (with the FLUX user subroutine).
6	Uniform flux on 1-2-6-5 face.
7	Nonuniform flux on 1-2-6-5 face (with the FLUX user subroutine).
8	Uniform flux on 2-3-7-6 face.

Load Type (IBODY)	Description
9	Nonuniform flux on 2-3-7-6 face (with the FLUX user subroutine).
10	Uniform flux on 3-4-8-7 face.
11	Nonuniform flux on 3-4-8-7 face (with the FLUX user subroutine).
12	Uniform flux on 1-4-8-5 face.
13	Nonuniform flux on 1-4-8-5 face (with the FLUX user subroutine).

For `IBODY= 3`, P in the FLUX user subroutine is the magnitude of volumetric flux at volumetric integration point NN of element N. For `IBODY` odd but not equal to 3, P is the magnitude of surface flux for surface integration point NN of element N. Surface flux is positive when heat energy is added to the elements.

Films

Same specification as Fluxes .

Tying

Use the UFORMSN user subroutine.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Current

Same specification as Fluxes .

Charge

Same specifications as Fluxes .

Output Points

Gaussian integration points. The PRINT ELEMENT or PRINT CHOICE options can be used to print out results for specific integration points in the output file. The PRINT ELEMENT option is only valid for the first 28 integration points while the PRINT CHOICE option can be used for any integration point number (from 1 to a maximum of 2040). On the post file, temperatures, temperature gradients and heat fluxes can be obtained for the four integration points of each layer. When no layer is specified, post file results are presented for layer 1.

Note:	As in all three-dimensional analysis, a large nodal bandwidth results in long computing times. Use the optimizers as much as possible.

Element 176

Three-dimensional, Twenty-node Composite Brick Element (Heat Transfer Element)

This element is an isoparametric 20-node composite brick written for three-dimensional heat transfer applications. Different material properties can be used for different layers within the element (see Figure 3-252). This element can also be used for electrostatic applications

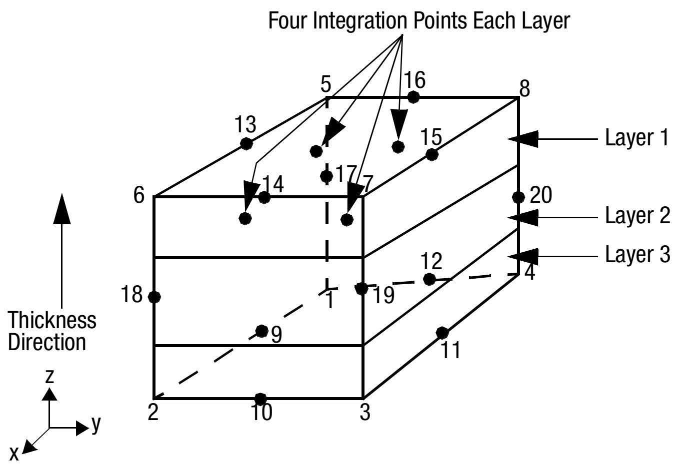


Figure 3-252 Typical 20-node 3-D Composite Heat Transfer Element

All heat transfer capabilities are currently supported for the element, except latent heat, thermal contact via the CONRAD GAP model definition option and fluid channel via the CHANNEL model definition option. Also, the CENTROID parameter cannot be used for this element.

Integration

The number of continuum layers within an element is input via the COMPOSITE option. To ensure the stability of the element, a minimum number of two layers is required within the element. Each layer is assumed to be placed parallel to a pair of opposite element faces, so that the “thickness” direction is from one of the element faces to its opposite one. For instance, if the layer is parallel to the 1, 2, 3, 4 and 5, 6, 7, 8 faces of the element, the “thickness” direction is from element face 1, 2, 3, 4 to element face 5, 6, 7, 8 (see Figure 3-252). The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains four integration points (see Figure 3-252). On each layer, you must input, via the COMPOSITE option, the thickness (or percentage of the thickness) and the material set id. See Marc Volume C: Program Input for the COMPOSITE option. A maximum of 510 layers can be used within each element. The mass matrix of this element is formed using twenty-seven-point Gaussian integration.

Quick Reference

Type 176

Twenty-node, isoparametric composite brick element for heat transfer.

Connectivity

Twenty nodes per element. Node numbering of the element is the same as for element type 44.

Geometry

The isoparametric direction of layers is defined in the third field (EGEOM3), enter 1, 2, or 3:

1	=	Material layers are parallel to the 1, 2, 3, 4 and 5, 6, 7, 8 faces; the thickness direction is from the 1, 2, 3, 4 face to the 5, 6, 7, 8 face of the element.
2	=	Material layers are parallel to the 1, 4, 8, 5 and 2, 3, 7, 6 faces; the thickness direction is from the 1, 4, 8, 5 face to the 2, 3, 7, 6 face of the element.
3	=	Material layers are parallel to the 2, 1, 5, 6 and 3, 4, 8, 7 faces; the thickness direction is from the 2, 1, 5, 6 face to the 3, 4, 8, 7 face of the element.

Coordinates

Three global coordinates in x-, y-, and z-directions.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)

Fluxes

Fluxes are distributed according to the appropriate selection of a value of IBODY. Surface fluxes are assumed positive when directed into the element.

Load Type	Description
0	Uniform flux on 1-2-3-4 face.
1	Nonuniform flux on 1-2-3-4 face (with the FLUX user subroutine).
2	Uniform body flux.
3	Nonuniform body flux (with the FLUX user subroutine).
4	Uniform flux on 6-5-8-7 face.
5	Nonuniform flux on 6-5-8-7 face (with the FLUX user subroutine).
6	Uniform flux on 2-1-5-6 face.
7	Nonuniform flux on 2-1-5-6 face (with the FLUX user subroutine).
8	Uniform flux on 3-2-6-7 face.
9	Nonuniform flux on 3-2-6-7 face (with the FLUX user subroutine).
10	Uniform flux on 4-3-7-8 face.
11	Nonuniform flux on 4-3-7-8 face (with the FLUX user subroutine).

Load Type	Description
12	Uniform flux on 1-4-8-5 face.
13	Nonuniform flux on 1-4-8-5 face (with the FLUX user subroutine).

For IBODY=3, the value of P in the FLUX user subroutine is the magnitude of volumetric flux at volumetric integration point NN of element N. For IBODY odd but not equal to 3, P is the magnitude of surface flux at surface integration point NN of element N. Surface flux is positive when heat energy is added to the element.

Films

Same specification as Fluxes .

Tying

Use the UFORMSN user subroutine.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Current

Same specifications as Fluxes .

Charge

Same specifications as Fluxes .

Output Points

Gaussian integration points. The PRINT ELEMENT or PRINT CHOICE options can be used to print out results for specific integration points in the output file. The PRINT ELEMENT option is only valid for the first 28 integration points while the PRINT CHOICE option can be used for any integration point number (from 1 to a maximum of 2040). On the post file, temperatures, temperature gradients and heat fluxes can be obtained for the four integration points of each layer. When no layer is specified, post file results are presented for layer 1.

Element 177

Quadrilateral Planar Four-node Composite Element (Heat Transfer Element)

This is an isoparametric four-noded composite element which can be used for planar heat transfer applications. Different material properties can be used for different layers within the element (see Figure 3-253). This element can also be used for electrostatic and magnetostatic applications.

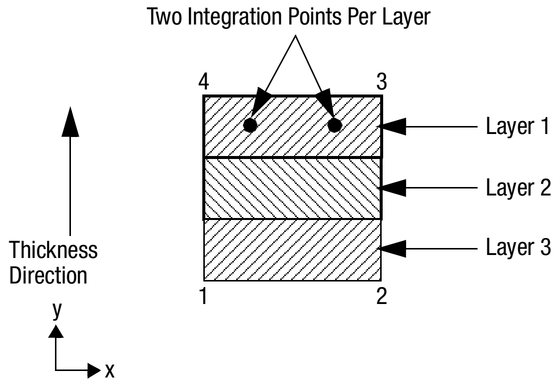


Figure 3-253 Typical Four-node Planar Composite Heat Transfer Element

All heat transfer capabilities are currently supported for the element, except latent heat, thermal contact via the CONRAD GAP model definition option and fluid channel via the CHANNEL model definition option. Also, the CENTROID parameter cannot be used for this element.

Integration

The number of continuum layers within an element is input via the COMPOSITE option. To ensure the stability of the element, a minimum number of two layers is required within the element. Each layer is assumed to be placed parallel to a pair of opposite element edges, so that the "thickness" direction is from one of the element edges to its opposite one. For instance (see Figure 3-253), if the layer is parallel to the 1, 2 and 3, 4 edges of the element, the "thickness" direction is from element edge 1, 2 to element edge 3, 4. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains two integration points (see Figure 3-253). On each layer, you must input, via the COMPOSITE option, the thickness (or percentage of the thickness) and the material set id. See Marc Volume C: Program Input for the COMPOSITE option. A maximum of 1020 layers can be used within each element.

Quick Reference

Type 177

Four-node, isoparametric, planar quadrilateral, heat transfer composite element.

Connectivity

Four nodes per element. Node numbering of the element is the same as for element type 39.

Geometry

The isoparametric direction of layers is defined in the third field (EGEOM3), enter 1 or 2:

1	=	Material layers are parallel to the 1, 2 and 3, 4 edges; the thickness direction is from the 1, 2 edge to the 3, 4 edge of the element.
2	=	Material layers are parallel to the 1, 4 and 2, 3 edges; the thickness direction is from the 1, 4 edge to the 2, 3 edge of the element.

If a nonzero value is entered in the fourth data field (EGEOM4), the temperatures at the integration points obtained from interpolation of nodal temperatures are constant throughout the element.

Coordinates

Two global coordinates in the x- and y-directions.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)
1 = potential (magnetostatic)

Fluxes

Flux types for distributed fluxes are as follows:

Flux Type	Description
0	Uniform flux per unit area 1-2 face of the element.
1	Uniform flux per unit volume on whole element.
2	Uniform flux per unit volume on whole element.
3	Nonuniform flux per unit area on 1-2 face of the element; magnitude given in the FLUX user subroutine.
4	Nonuniform flux per unit volume on whole element; magnitude given in the FLUX user subroutine.
5	Nonuniform flux per unit volume on whole element; magnitude given in the FLUX user subroutine.
6	Uniform flux per unit area on 2-3 face of the element.
7	Nonuniform flux per unit area on 2-3 face of the element; magnitude given in the FLUX user subroutine.
8	Uniform flux per unit area on 3-4 face of the element.
9	Nonuniform flux per unit area on 3-4 face of the element; magnitude given in the FLUX user subroutine.
10	Uniform flux per unit area on 4-1 face of the element.
11	Nonuniform flux per unit area on 4-1 face of the element; magnitude given in the FLUX user subroutine.

For all nonuniform fluxes, the magnitude is given in the FLUX user subroutine.

All fluxes are positive when adding heat to the element. In addition, point fluxes can be applied at the nodes. Edge fluxes are evaluated using a two-point integration scheme where the integration points have the same location as the nodal points.

Films

Same specification as Fluxes .

Tying

Use the UFORMSN user subroutine.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Current

Same specifications as Fluxes .

Charge

Same specifications as Fluxes .

Output Points

Gaussian integration points. The PRINT ELEMENT or PRINT CHOICE options can be used to print out results for specific integration points in the output file. The PRINT ELEMENT option is only valid for the first 28 integration points while the PRINT CHOICE option can be used for any integration point number (from 1 to a maximum of 2040). On the post file, temperatures, temperature gradients and heat fluxes can be obtained for the four integration points of each layer. When no layer is specified, post file results are presented for layer 1.

Element 178

Quadrilateral, Axisymmetric, Four-node Composite Element (Heat Transfer Element)

This is a isoparametric, axisymmetric, four-node composite element written for axisymmetric heat transfer applications. Different material properties can be used for different layers within the element (see Figure 3-254). This element can also be used for electrostatic and magnetostatic applications.

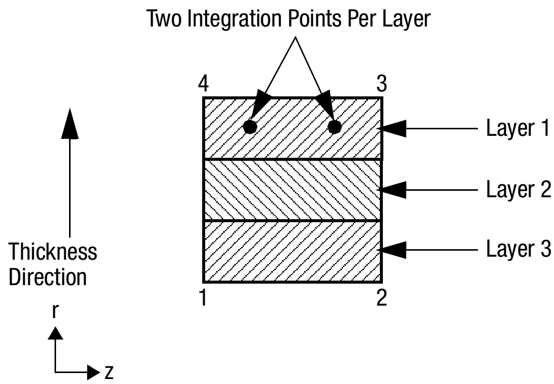


Figure 3-254 Typical Four-node Axisymmetric Composite Heat Transfer Element

All heat transfer capabilities are currently supported for the element, except latent heat, thermal contact via the CONRAD GAP model definition option and fluid channel via the CHANNEL model definition option. Also, the CENTROID parameter cannot be used for this element.

Integration

The number of continuum layers within an element is input via the COMPOSITE option. A maximum number of five layers can be used. To ensure the stability of the element, a minimum number of two layers is required within the element. Each layer is assumed to be placed parallel to a pair of opposite element edges, so that the “thickness” direction is from one of the element edges to its opposite one. For instance (see Figure 3-254), if the layer is parallel to the 1, 2 and 3, 4 edges of the element, the “thickness” direction is from element edge 1, 2 to element edge 3, 4. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains two integration points (see Figure 3-254). On each layer, you must input, via the COMPOSITE option, the thickness (or percentage of the thickness) and the material set id. See Marc Volume C: Program Input for the COMPOSITE option. A maximum of 1020 layers can be used within each element.

Quick Reference

Type 178

Four-node, isoparametric, quadrilateral, axisymmetric composite heat transfer element.

Connectivity

Four nodes per element. Node numbering of the element is the same as for element type 40.

Geometry

The isoparametric direction of layers is defined in the third field (EGEOM3), enter 1 or 2:

1	=	Material layers are parallel to the 1, 2 and 3, 4 edges; the thickness direction is from the 1, 2 edge to the 3, 4 edge of the element.
2	=	Material layers are parallel to the 1, 4 and 2, 3 edges; the thickness direction is from the 1, 4 edge to the 2, 3 edge of the element.

If a nonzero value is entered in the fourth data field (EGEOM4), the temperatures at the integration points obtained from interpolation of nodal temperatures are constant throughout the element.

Coordinates

Two global coordinates in the z- and r-directions.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)
1 = potential (magnetostatic)

Fluxes

Flux types for distributed fluxes are as follows:

Flux Type	Description
0	Uniform flux per unit area 1-2 face of the element.
1	Uniform flux per unit volume on whole element.
2	Uniform flux per unit volume on whole element.
3	Nonuniform flux per unit area on 1-2 face of the element; magnitude given in the FLUX user subroutine.
4	Nonuniform flux per unit volume on whole element; magnitude given in the FLUX user subroutine.
5	Nonuniform flux per unit volume on whole element; magnitude given in the FLUX user subroutine.
6	Uniform flux per unit area on 2-3 face of the element.
7	Nonuniform flux per unit area on 2-3 face of the element; magnitude given in the FLUX user subroutine.
8	Uniform flux per unit area on 3-4 face of the element.
9	Nonuniform flux per unit area on 3-4 face of the element; magnitude given in the FLUX user subroutine.
10	Uniform flux per unit area on 4-1 face of the element.
11	Nonuniform flux per unit area on 4-1 face of the element; magnitude given in the FLUX user subroutine.

For all nonuniform fluxes, the magnitude is given in the FLUX user subroutine.

All fluxes are positive when adding heat to the element. In addition, point fluxes can be applied at the nodes. Edge fluxes are evaluated using a two-point integration scheme where the integration points have the same location as the nodal points.

Films

Same specification as Fluxes .

Tying

Use the UFORMSN user subroutine.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Current

Same specifications as Fluxes .

Charge

Same specifications as Fluxes .

Output Points

Gaussian integration points. The PRINT ELEMENT or PRINT CHOICE options can be used to print out results for specific integration points in the output file. The PRINT ELEMENT option is only valid for the first 28 integration points while the PRINT CHOICE option can be used for any integration point number (from 1 to a maximum of 2040). On the post file, temperatures, temperature gradients and heat fluxes can be obtained for the four integration points of each layer. When no layer is specified, post file results are presented for layer 1.

Element 179

Quadrilateral, Planar Eight-node Composite Element (Heat Transfer Element)

This is a isoparametric eight-node composite element written for planar heat transfer applications. Different material properties can be used for different layers within the element (see Figure 3-255). This element can also be used for electrostatic and magnetostatic applications.

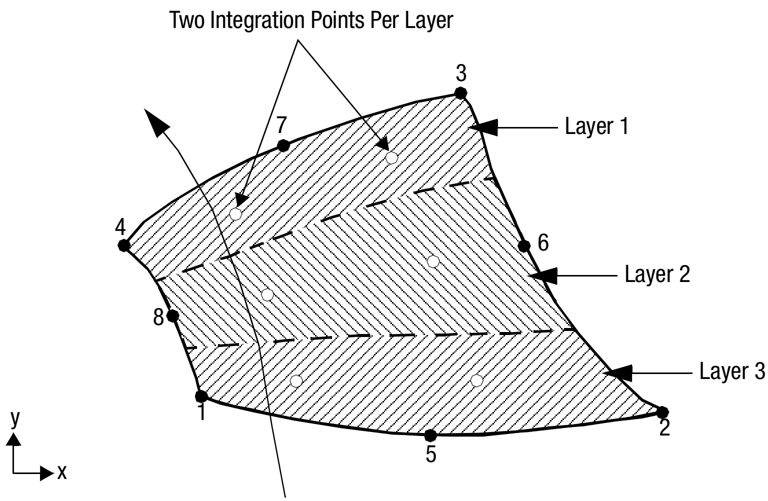


Figure 3-255 Typical Eight-node Planar Heat Transfer Composite Element

All heat transfer capabilities are currently supported for the element, except latent heat, thermal contact via the CONRAD GAP model definition option and fluid channel via the CHANNEL model definition option. Also, the CENTROID parameter cannot be used for this element.

Integration

The number of continuum layers within an element is input via the COMPOSITE option. To ensure the stability of the element, a minimum number of two layers is required within the element. Each layer is assumed to be placed parallel to a pair of opposite element edges, so that the "thickness" direction is from one of the element edges to its opposite one. For instance (see Figure 3-255), if the layer is parallel to the 1, 2 and 3, 4 edges of the element, the "thickness" direction is from element edge 1, 2 to element edge 3, 4. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains two integration points (see Figure 3-255). On each layer, you must input, via the COMPOSITE option, the thickness (or percentage of the thickness) and the material set id. See Marc Volume C: Program Input for the COMPOSITE option. A maximum of 1020 layers can be used within each element.

Quick Reference

Type 179

Eight-node, isoparametric, quadrilateral, planar heat transfer composite element.

Connectivity

Eight nodes per element. Node numbering of the element is the same as for element type 41.

Geometry

The isoparametric direction of layers is defined in the third field (`EGEOM3`), enter 1 or 2:

1	=	Material layers are parallel to the 1, 2 and 3, 4 edges; the thickness direction is from the 1, 2 edge to the 3, 4 edge of the element.
2	=	Material layers are parallel to the 1, 4 and 2, 3 edges; the thickness direction is from the 1, 4 edge to the 2, 3 edge of the element.

Coordinates

Two global coordinates in the x- and y-directions.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)
1 = potential (magnetostatic)

Fluxes

Surface Fluxes

Surface fluxes are specified as below. All are per unit surface area. All nonuniform fluxes are specified through the FLUX user subroutine.

Traction Type	Description
0	Uniform flux on 1-5-2 face.
1	Nonuniform flux on 1-5-2 face.
8	Uniform flux on 2-6-3 face.
9	Nonuniform flux on 2-6-3 face.
10	Uniform flux on 3-7-4 face.
11	Nonuniform flux on 3-7-4 face.
12	Uniform flux on 4-8-1 face.
13	Nonuniform flux on 4-8-1 face.

Volumetric Fluxes

Flux type 2 is uniform flux (per unit volume); type 3 is nonuniform flux per unit volume, with magnitude given through the FLUX user subroutine.

Films

Same specification as Fluxes .

Tying

Use the UFORMSN user subroutine.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Current

Same specifications as Fluxes .

Charge

Same specifications as Fluxes .

Output Points

Gaussian integration points. The PRINT ELEMENT or PRINT CHOICE options can be used to print out results for specific integration points in the output file. The PRINT ELEMENT option is only valid for the first 28 integration points while the PRINT CHOICE option can be used for any integration point number (from 1 to a maximum of 2040). On the post file, temperatures, temperature gradients and heat fluxes can be obtained for the four integration points of each layer. When no layer is specified, post file results are presented for layer 1.

Element 180

Quadrilateral, Axisymmetric, Eight-node Composite Element (Heat Transfer Element)

This is a isoparametric, axisymmetric, eight-node composite element written for axisymmetric heat transfer applications. Different material properties can be used for different layers within the element (see Figure 3-256). This element can also be used for electrostatic and magnetostatic applications.

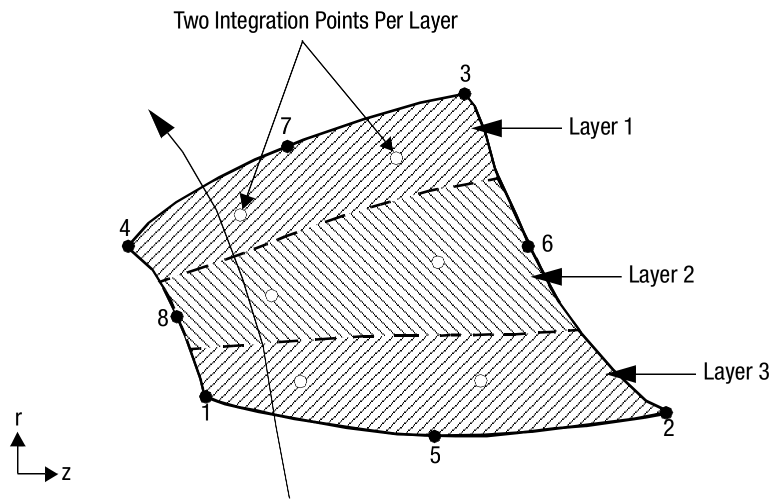


Figure 3-256 Typical Eight-node Axisymmetric Heat Transfer Composite Element

All heat transfer capabilities are currently supported for the element, except latent heat, thermal contact via the CONRAD GAP model definition option and fluid channel via the CHANNEL model definition option. Also, the CENTROID parameter cannot be used for this element.

Integration

The number of continuum layers within an element is input via the COMPOSITE option. To ensure the stability of the element, a minimum number of two layers is required within the element. Each layer is assumed to be placed parallel to a pair of opposite element edges, so that the “thickness” direction is from one of the element edges to its opposite one. For instance (see Figure 3-256), if the layer is parallel to the 1, 2 and 3, 4 edges of the element, the “thickness” direction is from element edge 1, 2 to element edge 3, 4. The element is integrated using a numerical scheme based on Gauss quadrature. Each layer contains two integration points (see Figure 3-256). On each layer, you must input, via the COMPOSITE option, the thickness (or percentage of the thickness) and the material set id. See Marc Volume C: Program Input for the COMPOSITE option. A maximum of 1020 layers can be used within each element.

Quick Reference

Type 180

Eight-node, isoparametric, quadrilateral, axisymmetric heat transfer composite element.

Connectivity

Eight nodes per element. Node numbering of the element is the same as for element type 42.

Geometry

The isoparametric direction of layers is defined in the third field (EGEOM3), enter 1 or 2:

1	=	Material layers are parallel to the 1, 2 and 3, 4 edges; the thickness direction is from the 1, 2 edge to the 3, 4 edge of the element.
2	=	Material layers are parallel to the 1, 4 and 2, 3 edges; the thickness direction is from the 1, 4 edge to the 2, 3 edge of the element.

Coordinates

Two global coordinates in the z- and r-directions.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)
1 = potential (magnetostatic)

Fluxes

Surface Fluxes

Surfaces fluxes are specified as below. All are per unit surface area. All nonuniform fluxes are specified through the FLUX user subroutine.

Flux Type (IBODY)	Description
0	Uniform flux on 1-5-2 face.
1	Nonuniform flux on 1-5-2 face.
8	Uniform flux on 2-6-3 face.
9	Nonuniform flux on 2-6-3 face.
10	Uniform flux on 3-7-4 face.
11	Nonuniform flux on 3-7-4 face.
12	Uniform flux on 4-8-1 face.
13	Nonuniform flux on 4-8-1 face.

Volumetric Fluxes

Flux type 2 is uniform flux (per unit volume); type 3 is nonuniform flux per unit volume, with magnitude given through the FLUX user subroutine.

Films

Same specification as Fluxes .

Tying

Use the UFORMSN user subroutine.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is available.

Current

Same specifications as Fluxes .

Charge

Same specifications as Fluxes .

Output Points

Gaussian integration points. The PRINT ELEMENT or PRINT CHOICE options can be used to print out results for specific integration points in the output file. The PRINT ELEMENT option is only valid for the first 28 integration points while the PRINT CHOICE option can be used for any integration point number (from 1 to a maximum of 2040). On the post file, temperatures, temperature gradients and heat fluxes can be obtained for the four integration points of each layer. When no layer is specified, post file results are presented for layer 1.

Element 181

Three-dimensional Four-node Magnetostatic Tetrahedron

This is a linear isoparametric three-dimensional magnetostatic tetrahedron (see Figure 3-257). As this element uses linear interpolation functions, the magnetic induction is constant throughout the element. The element is integrated numerically using one point at the centroid of the element.

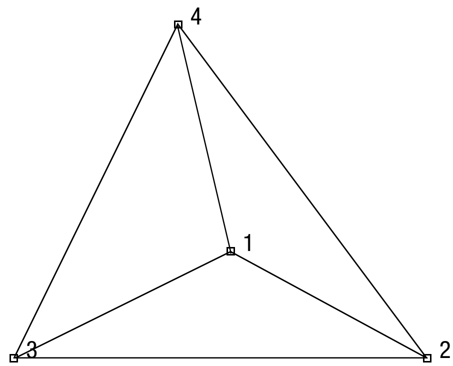


Figure 3-257 Form of Element 181

Connectivity

The convention for the ordering of the connectivity array is as follows:

> Nodes 1, 2, 3 are the corners of the first face, given in counterclockwise order when viewed from inside the element. Node 4 is on the opposing vertex. Note that in most normal cases, the elements are generated automatically via a preprocessor (such as Mentat or a CAD program) so that you need not be concerned with the node numbering scheme.

Quick Reference

Type 181

Four-nodes, isoparametric arbitrary distorted magnetostatic tetrahedron.

Connectivity

Four nodes numbered as described in the connectivity write-up for this element and as shown in Figure 3-257.

Geometry

The constraint $\nabla \cdot \boldsymbol{A} = 0$ is enforced with a penalty formulation. The value of the penalty factor is given in the second field `EGEOM7` (the default is 0.0001).

Coordinates

Three global coordinates in the x-, y-, and z-directions.

Degrees of Freedom

1 = x component of vector potential
2 = y component of vector potential
3 = z component of vector potential

Distributed Currents

Distributed currents chosen by value of IBODY are as follows:

Load Type	Description
0	Uniform current on 1-2-3 face.
1	Nonuniform current on 1-2-3 face.
2	Uniform current on 1-2-4 face.
3	Nonuniform current on 1-2-4 face.
4	Uniform current on 2-3-4 face.
5	Nonuniform current on 2-3-4 face.
6	Uniform current on 1-3-4 face.
7	Nonuniform current on 1-3-4 face.
8	Uniform volumetric current per unit volume in x-direction.
9	Nonuniform volumetric current per unit volume in x-direction; magnitude is supplied through the FORCEM user subroutine.
10	Uniform volumetric current per unit volume in y-direction.
11	Nonuniform volumetric current per unit volume in y-direction; magnitude is supplied through the FORCEM user subroutine.
12	Uniform volumetric current per unit volume in z-direction; magnitude is supplied through the FORCEM user subroutine.
13	Nonuniform volumetric current per unit volume in z direction; magnitude is supplied through the FORCEM user subroutine.
106	Uniform volumetric current in global direction. Enter three magnitudes of the current in respectively global x-, y-, z-direction.
107	Nonuniform volumetric current; magnitude and direction is supplied through the FORCEM user subroutine.

The FORCEM user subroutine is called once per integration point when flagged. The magnitude of load defined by DIST CURRENT is ignored and the FORCEM value is used instead.

Transformation

Three global degrees of freedom can be transformed to local degrees of freedom.

Tying

Use the UFORMSN user subroutine.

Output Points

Centroid.

Element 182

Three-dimensional Ten-node Magnetostatic Tetrahedron

This is a second-order isoparametric three-dimensional magnetostatic tetrahedron. Each edge forms a parabola so that four nodes define the corners of the element and six nodes define the position of the "midpoint" of each edge (Figure 3-258). This allows for an accurate representation of the magnetic induction in magnetostatic analyses.

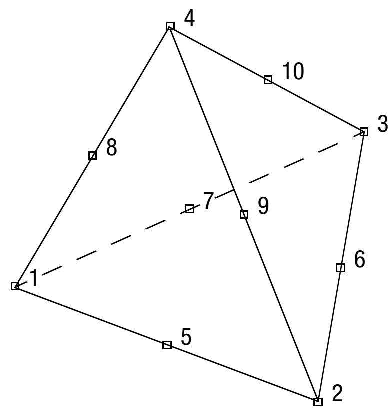


Figure 3-258 Form of Element 182

The coefficient matrix is numerically integrated using four-point integration.

Connectivity

The convention for the ordering of the connectivity array is as follows:

> Nodes 1, 2, 3 are the corners of the first face, given in counterclockwise order when viewed from inside the element. Node 4 is on the opposing vertex. Nodes 5, 6, 7 are on the first face between nodes 1 and 2, 2 and 3, 3 and 1, respectively. Nodes 8, 9, 10 are along the edges between the first face and node 4, between nodes 1 and 4, 2 and 4, 3 and 4, respectively.

Note that in most normal cases, the elements will be generated automatically via a preprocessor (such as Mentat (Mentat)) so that you need not be concerned with the node numbering scheme.

Integration

The element is integrated numerically using four points (Gaussian quadrature). The first plane of such points is closest to the 1, 2, 3 face of the element with the first point closest to the first node of the element (see Figure 3-259).

Quick Reference

Type 182

Ten-nodes, isoparametric arbitrary distorted magnetostatic tetrahedron.

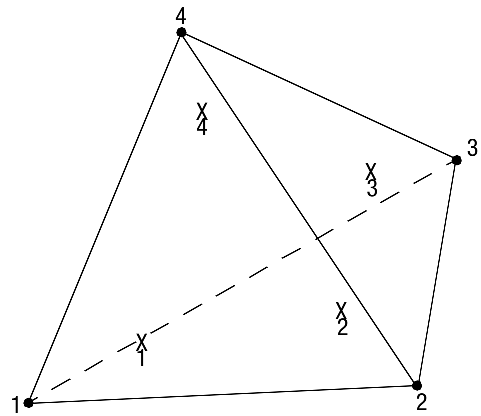


Figure 3-259 Element 182 Integration Plane

Connectivity

Ten nodes numbered as described in the connectivity write-up for this element and as shown in Figure 3-258.

Geometry

The constraint $\nabla \cdot \boldsymbol{A} = 0$ is enforced with a penalty formulation. The value of the penalty factor is given in the second field EGEOM8 (the default is 1.0). The values of the magnetic potential and the magnetic induction are highly dependent upon the value of the penalty factor. For this reason, lower-order elements are recommended.

Coordinates

Three global coordinates in the x-, y-, and z-directions.

Degrees of Freedom

1 = x component of vector potential
2 = y component of vector potential
3 = z component of vector potential

Distributed Currents

Distributed currents chosen by value of IBODY are as follows:

Load Type	Description
0	Uniform current on 1-2-3 face.
1	Nonuniform current on 1-2-3 face.
2	Uniform current on 1-2-4 face.
3	Nonuniform current on 1-2-4 face.
4	Uniform current on 2-3-4 face.
5	Nonuniform current on 2-3-4 face.

Load Type	Description
6	Uniform current on 1-3-4 face.
7	Nonuniform current on 1-3-4 face.
8	Uniform volumetric current per unit volume in x direction.
9	Nonuniform volumetric current per unit volume in x direction; magnitude is supplied through the FORCEM user subroutine.
10	Uniform volumetric current per unit volume in y direction.
11	Nonuniform volumetric current per unit volume in y direction; magnitude is supplied through the FORCEM user subroutine.
12	Uniform volumetric current per unit volume in z direction; magnitude is supplied through the FORCEM user subroutine.
13	Nonuniform volumetric current per unit volume in z direction; magnitude is supplied through the FORCEM user subroutine.
106	Uniform volumetric current in global direction. Enter three magnitudes of the current in respectively global x, y, z direction.
107	Nonuniform volumetric current; magnitude and direction is supplied through the FORCEM user subroutine.

The FORCEM user subroutine is called once per integration point when flagged. The magnitude of load defined by DIST CURRENT is ignored and the FORCEM value is used instead.

Transformation

Three global degrees of freedom can be transformed to local degrees of freedom.

Tying

Use the UFORMSN user subroutine.

Output Points

Centroid or four Gaussian integration points (see Figure 3-259).

You should invoke the appropriate OPTIMIZE option in order to minimize the matrix solution time.

Note:	A large bandwidth results in a lengthy central processing time.

Element 183

Three-dimensional Magnetostatic Current Carrying Wire

This element is a three-dimensional simple, linear, straight line carrying a magnetostatic current. The purpose of the element is to facilitate the application of current to a structure. A connected line of elements forms the wire, then the actual current is applied to these elements in the form of a distributed current. The direction of this current is along the direction of the line elements. These elements can be embedded in a host material using the INSERT option, or they can be placed along element edges, sharing the nodes of the solid elements 109, 181, or 182.

Note:	The purpose of this element is to define an external loading; so, this element does not have material and geometric properties.

Quick Reference

Type 183

Three-dimensional, two-node, magnetostatic current carrying wire.

Connectivity

Two nodes per element (see Figure 3-260).

Figure 3-260 Three-dimensional Magnetostatic Current Carrying Wire

Geometry

This element does not have geometry input.

Coordinates

Three coordinates per node in the global x-, y-, and z-direction.

Degrees of Freedom

Marc converts the current to point currents on the nodes.

Distributed Currents

Distributed currents chosen by value of `IBODY` are as follows:

Flux Type	Description
108	Actual current in the direction of the line element.
109	Nonuniform current in the direction of the line element; magnitude is supplied through the FLUX user subroutine.

Tying

Capability is not available.

Joule Heating

Capability is not available.

Magnetodynamics

Capability is available.

Output Points

The applied current is available as point currents at the two nodes.

Element 184

Three-dimensional Ten-node Tetrahedron

This element is a second-order isoparametric three-dimensional tetrahedron. Each edge forms a parabola so that four nodes define the corners of the element and a further six nodes define the position of the "midpoint" of each edge (Figure 3-261). This allows for an accurate representation of the strain field in elastic analyses.

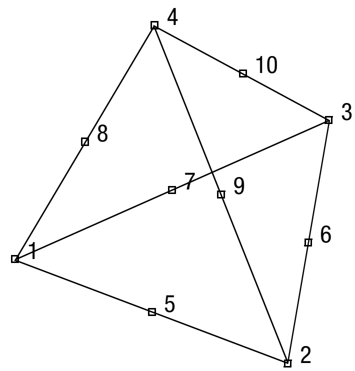


Figure 3-261 Form of Element 184

The stiffness of this element is formed using four-point integration. The mass matrix of this element is formed using sixteen-point Gaussian integration.

This element can be used for all constitutive relations. When using the Mooney or Ogden incompressible material models in the total Lagrange framework, use element type 130. Element type 130 is also preferable for small strain incompressible elasticity.

The element is based on the formulation of element 127 with additional incompatible interpolation functions added. This improves the behavior in bending and alleviates locking due to incompressibility. The enhancing functions satisfy orthogonality principles to maintain convergence. The Hu-Washizu variational principle is used.

Geometry

The geometry of the element is interpolated from the Cartesian coordinates of ten nodes.

Connectivity

The convention for the ordering of the connectivity array is as follows:

> Nodes 1, 2, 3 are the corners of the first face, given in counterclockwise order when viewed from inside the element. Node 4 is on the opposing vertex. Nodes 5, 6, 7 are on the first face between nodes 1 and 2, 2 and 3, 3 and 1, respectively. Nodes 8, 9, 10 are along the edges between the first face and node 4, between nodes 1 and 4, 2 and 4, 3 and 4, respectively.

Note that in most normal cases, the elements will be generated automatically via a preprocessor (such as Mentat) so that you need not be concerned with the node numbering scheme.

Integration

The element is integrated numerically using four points (Gaussian quadrature). The first plane of such points is closest to the 1, 2, 3 face of the element with the first point closest to the first node of the element (see Figure 3-262).

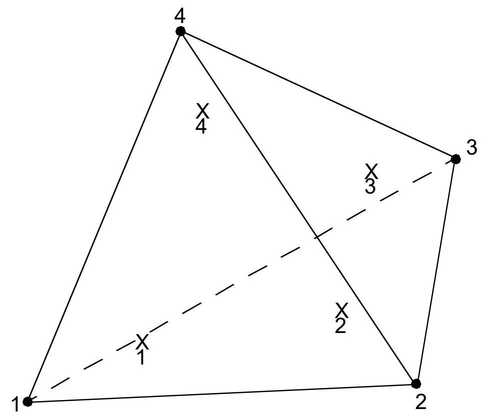


Figure 3-262 Element 184 Integration Plane

Quick Reference

Type 184

Ten nodes, isoparametric arbitrary distorted tetrahedron.

Connectivity

Ten nodes numbered as described in the connectivity write-up for this element and as shown in Figure 3-261.

Geometry

Not required.

Coordinates

Three global coordinates in the x-, y-, and z-directions.

Degrees of Freedom

Three global degrees of freedom, u, v, and w.

Distributed Loads

Distributed loads chosen by value of `IBODY` are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3 face.
1	Nonuniform pressure on 1-2-3 face.

Load Type	Description
2	Uniform pressure on 1-2-4 face.
3	Nonuniform pressure on 1-2-4 face.
4	Uniform pressure on 2-3-4 face.
5	Nonuniform pressure on 2-3-4 face.
6	Uniform pressure on 1-3-4 face.
7	Nonuniform pressure on 1-3-4 face.
8	Uniform body force per unit volume in x-direction.
9	Nonuniform body force per unit volume in x-direction.
10	Uniform body force per unit volume in y-direction.
11	Nonuniform body force per unit volume in y-direction.
12	Uniform body force per unit volume in z-direction.
13	Nonuniform body force per unit volume in z-direction.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

The FORCEM user subroutine is called once per integration point when flagged. The magnitude of load defined by DIST LOADS is ignored and the FORCEM value is used instead.

For nonuniform body force, force values must be provided for the four integration points.

For nonuniform surface pressure, force values need only be supplied for the three integration points on the face of application.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

$1 = \varepsilon_{xx}$

$2 = \varepsilon_{yy}$

$3 = \varepsilon_{zz}$

$4 = \varepsilon_{xy}$

$5 = \varepsilon_{yz}$

$6 = \varepsilon_{zx}$

Output of Stresses

Same as for Output of Strains .

Transformation

Three global degrees of freedom can be transformed to local degrees of freedom.

Tying

Use the UFORMSN user subroutine.

Output Points

Centroid or four Gaussian integration points (see Figure 3-262).

You should invoke the appropriate OPTIMIZE option in order to minimize the matrix solution time.

Note:	A large bandwidth results in a lengthy central processing time.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinate directions.

Note:	Distortion of element during analysis can cause bad solutions. Element type 7 is preferred.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 133. See Element 133 for a description of the conventions used or entering the flux and film data for this element. Volumetric flux due to dissipation of plastic work specified with type 101.

Element 185 Three-dimensional Eight-node Solid Shell, Selective Reduced Integration

Element type 185 is a shell element with eight-node brick topology. The element uses enhanced assumed strain formulation for transverse normal component (ε_{zz}) and assumed strain formulation for transverse shear components. This element is only available in analysis using Finite Strain Plasticity or Hookean elasticity (using additive decomposition of strain rates). The element is not available in analysis using total Lagrange procedure (Hookean elasticity and elastomers) and updated Lagrange procedure (elastomers and finite strain plasticity) using multiplicative decomposition of deformation gradient. For details refer to:

- Alves de Sousa, R.J., Yoon, J.W.,Cardoso, R.P.R., Fontes Valente,R.A., Gracio, J.J, “On the use of reduced enhanced solid-shell (RESS) element for sheet forming simulations”, *Int. J. Plasticity*, Vol. 23, pp. 490-515 (2007).
- Cardoso, R.P.R, Yoon, J.W., Marhadika, M., Choudhry, S., Alves de Sousa, R.J., Fontes Valente, R.A., “Enhanced Assumed Strain (EAS) and Assumed natural Strain (ANS) Methods for One-Point Quadrature Solid-Shell Element”, *Int. J. For Numerical Methods in Eng.* submitted, 2007)

The stiffness of this element is formed using one integration point in the element plane and a user defined number through the element thickness. In this way the element can capture accurate material plasticity under bending load. An additional variationally consistent stiffness term is included to eliminate the hourglass modes that are normally associated with reduced integration.

This element may replace classical shell elements in applications that require double-sided contact.

Geometric Basis

The element is defined geometrically by the (x,y,z) coordinates of the eight corner nodes (see Figure 3-263). Please note that the thickness orientation of the element must align with the direction of face 1-2-3-4 to face 5-6-7-8.

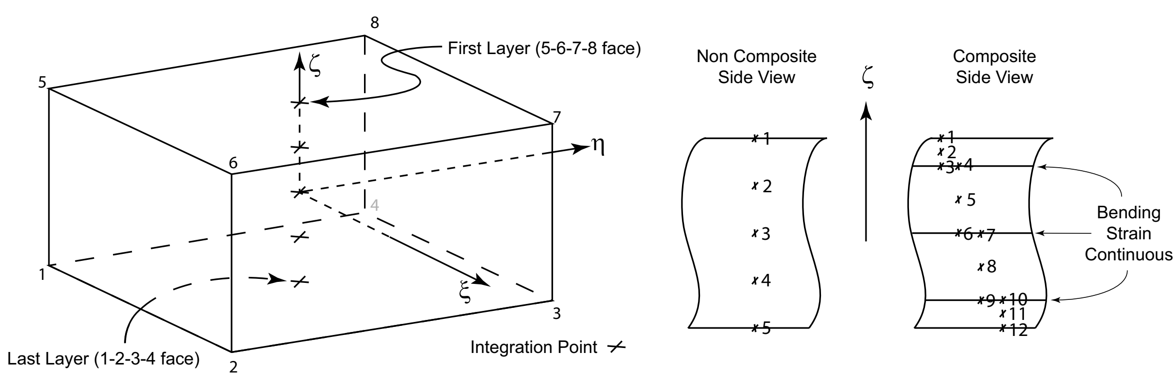


Figure 3-263 Element 185 Connectivity and Integration Points for Composite and Non Composite Materials

The stress and strain output are given in local orthogonal system (V_1, V_2, V_3). It is defined at the element centroid, as follows:

$$V_1 = \frac{\partial x}{\partial \xi} \Big/ \left|\frac{\partial x}{\partial \xi}\right| , \qquad t_2 = \frac{\partial x}{\partial \eta} \Big/ \left|\frac{\partial x}{\partial \eta}\right|$$

$$V_3 = (V_1 \times t_2) / \left|V_1 \times t_2\right|$$

$$V_2 = V_3 \times V_1$$

The element thickness is defined as the projection of the distance between the upper and lower integration points to V_3-axis.

Quick Reference

Type 185

Three-dimensional, eight-node, solid shell element.

Connectivity

Eight nodes per element. Node numbering must follow the scheme below (see Figure 3-263):

Nodes 1, 2, 3, and 4 are corners of one face, given in counterclockwise order when viewed from inside the element. Node 5 has the same edge as node 1. Node 6 has the same edge as node 2. Node 7 has the same edge as node 3. Node 8 has the same edge as node 4.

Geometry

If the automatic brick to shell constraints are to be used, the first field must contain the transition thickness (see Figure 3-264). Note that in a coupled analysis, there are no constraints for the temperature degrees of freedom.

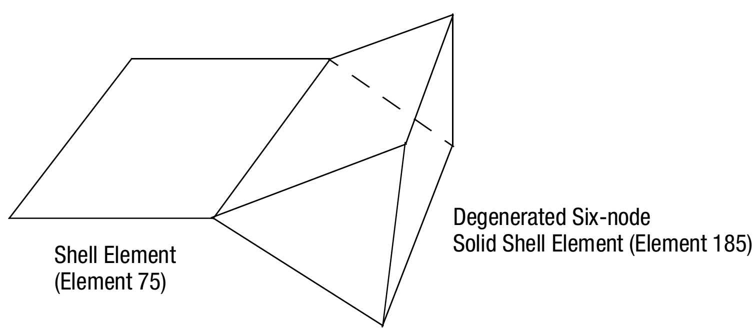


Figure 3-264 Shell-to-Solid Automatic Constraint

When this element refers to non-composite material, the fifth field may be used to input a factor greater than 0 and less than 1 to scale the transverse shear modulus (the common value for isotropic material is 5/6).

Integration

The element is integrated numerically using a user-defined number of points through the element thickness based upon the number of layers given on the SHELL SECT parameter or COMPOSITE model definition option. The numerical integration for non-composite material is done using Simpson's rule. The first and the last points are located at 5-6-7-8 face and 1-2-3-4 face, respectively (see Figure 3-263). For composite material, every layer will have three integration points.

Coordinates

Three global coordinates in the x-, y-, and z-directions.

Degrees of Freedom

Three global degrees of freedom, u, v, and w.

Distributed Loads

Distributed loads chosen by value of `IBODY` are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3-4 face.
1	Nonuniform pressure on 1-2-3-4 face; magnitude supplied through the FORCEM user subroutine.
2	Uniform body force per unit volume in -z-direction.
3	Nonuniform body force per unit volume (e.g., centrifugal force); magnitude and direction supplied through the FORCEM user subroutine.
4	Uniform pressure on 6-5-8-7 face.
5	Nonuniform pressure on 6-5-8-7 face (FORCEM user subroutine).
6	Uniform pressure on 2-1-5-6 face.
7	Nonuniform pressure on 2-1-5-6 face (FORCEM user subroutine).
8	Uniform pressure on 3-2-6-7 face.
9	Nonuniform pressure on 3-2-6-7 face (FORCEM user subroutine).
10	Uniform pressure on 4-3-7-8 face.
11	Nonuniform pressure on 4-3-7-8 face (FORCEM user subroutine).
12	Uniform pressure on 1-4-8-5 face.
13	Nonuniform pressure on 1-4-8-5 face (FORCEM user subroutine).
20	Uniform pressure on 1-2-3-4 face.
21	Nonuniform load on 1-2-3-4 face; magnitude and direction supplied in the FORCEM user subroutine.
22	Uniform body force per unit volume in -z-direction.
23	Nonuniform body force per unit volume (e.g., centrifugal force); magnitude and direction supplied through the FORCEM user subroutine.
24	Uniform pressure on 6-5-8-7 face.
25	Nonuniform load on 6-5-8-7 face; magnitude and direction supplied in the FORCEM user subroutine.

Load Type	Description
26	Uniform pressure on 2-1-5-6 face.
27	Nonuniform load on 2-1-5-6 face; magnitude and direction supplied in the FORCEM user subroutine.
28	Uniform pressure on 3-2-6-7 face.
29	Nonuniform load on 3-2-6-7 face; magnitude and direction supplied in the FORCEM user subroutine.
30	Uniform pressure on 4-3-7-8 face.
31	Nonuniform load on 4-3-7-8 face; magnitude and direction supplied in the FORCEM user subroutine.
32	Uniform pressure on 1-4-8-5 face.
33	Nonuniform load on 1-4-8-5 face; magnitude and direction supplied in the FORCEM user subroutine.
40	Uniform shear 1-2-3-4 face in the 1-2 direction.
41	Nonuniform shear 1-2-3-4 face in the 1-2 direction.
42	Uniform shear 1-2-3-4 face in the 2-3 direction.
43	Nonuniform shear 1-2-3-4 face in the 2-3 direction.
48	Uniform shear 6-5-8-7 face in the 5-6 direction.
49	Nonuniform shear 6-5-8-7 face in the 5-6 direction.
50	Uniform shear 6-5-8-7 face in the 6-7 direction.
51	Nonuniform shear 6-5-8-7 face in the 6-7 direction.
52	Uniform shear 2-1-5-6 face in the 1-2 direction.
53	Nonuniform shear 2-1-5-6 face in the 1-2 direction.
54	Uniform shear 2-1-5-6 face in the 1-5 direction.
55	Nonuniform shear 2-1-5-6 face in the 1-5 direction.
56	Uniform shear 3-2-6-7 face in the 2-3 direction.
57	Nonuniform shear 3-2-6-7 face in the 2-3 direction.
58	Uniform shear 3-2-6-7 face in the 2-6 direction.
59	Nonuniform shear 2-3-6-7 face in the 2-6 direction.
60	Uniform shear 4-3-7-8 face in the 3-4 direction.
61	Nonuniform shear 4-3-7-8 face in the 3-4 direction.
62	Uniform shear 4-3-7-8 face in the 3-7 direction.
63	Nonuniform shear 4-3-7-8 face in the 3-7 direction.
64	Uniform shear 1-4-8-5 face in the 4-1 direction.
65	Nonuniform shear 1-4-8-5 face in the 4-1 direction.
66	Uniform shear 1-4-8-5 in the 1-5 direction.
67	Nonuniform shear 1-4-8-5 face in the 1-5 direction.

Load Type	Description
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Pressure forces are positive into element faces.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

1 = ε_{xx}
2 = ε_{yy}
3 = ε_{zz}
4 = γ_{xy}
5 = γ_{yz}
6 = γ_{zx}

Output of Stresses

Same as for Output of Strains .

Transformation

Three global degrees of freedom can be transformed to local degrees of freedom.

Tying

Use the UFORMSN user subroutine.

Output Points

User defined integration points (see Figure 3-263).

You should invoke the appropriate OPTIMIZE option in order to minimize the matrix solution time.

Note:	A large bandwidth results in a lengthy central processing time.

Updated Lagrange Procedure, Finite Strain Plasticity and Elastomers

Finite strain plasticity is available in the additive decomposition mode only; F^eF^p is not available. Elastomer capability is not available with either total or updated Lagrange.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 175. See Element 149 for a description of the conventions used or entering the flux and film data for this element. Volumetric flux due to dissipation of plastic work specified with type 101.

Assumed Strain

Extra parameters for enhanced assumed strain formulation is available to improve the in-plane bending behavior. This increases the stiffness assembly costs per element, but it improves the accuracy.

Element 186

Four-node Planar Interface Element

Element type 186 is a mechanical four-node planar interface element, which can be used to simulate the onset and progress of delamination.

The connectivity of the element is shown in Figure 3-265. The element is typically used to model the interface between different materials, where nodes 1 and 2 correspond to one side of the interface (called the bottom) and nodes 3 and 4 to the other (called the top). The stress components of the element are one normal and one shear traction, which are expressed with respect to the local coordinate system $(\underset{\sim}{v}_1, \underset{\sim}{v}_2)$, indicated in Figure 3-265. The corresponding deformations are the relative displacements between the top and the bottom edge of the element. The element is allowed to be infinitely thin, in which case the edges 1-2 and 3-4 coincide.

The constitutive behavior of the element is defined via the COHESIVE model definition option.

Because the element is thin, there is no mass associated with the element. It should be noted that there is also no geometric or initial stress stiffening.

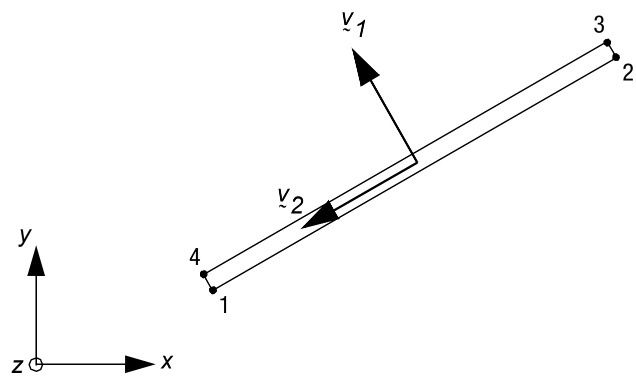


Figure 3-265 Element Type 186: Connectivity and Local Coordinate System

Geometric Basis

The first element base vector $\underset{\sim}{v}_1$ is obtained by rotating the direction vector from the middle of edge 1-4 to the middle of edge 2-3 counterclockwise over 90 degrees. Together with the first element base vector and the global z-axis, the second element base vector $\underset{\sim}{v}_2$ forms a right-hand system (see Figure 3-265).

The element stiffness matrix is integrated numerically using a two-point integration scheme. By default, the position of the integration points corresponds to the Gauss integration scheme. As an alternative, the Newton-Cotes/Lobatto scheme can be selected via the GEOMETRY option, in which case the integration points are located at the middle of the edges 1-4 and 2-3 (see Figure 3-266). The latter scheme may be advantageous in cases where the interface is relatively stiff compared to the surrounding material.

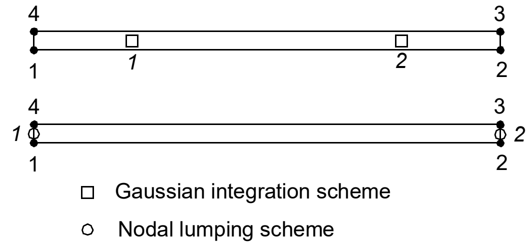


Figure 3-266 Element 186: Mid-line and Location of Integration Points

Quick Reference

Type 186

Linear, four-node planar interface element.

Connectivity

Four nodes per element. Node numbering must be counterclockwise (see Figure 3-265), so that the two sides of the interface are the element edges 1-2 and 3-4.

Geometry

The thickness is entered in the first data field. Default thickness is one. If the fifth data field is set to one, the Newton-Cotes/Lobatto integration scheme is used instead of the default Gaussian integration scheme.

Coordinates

Two coordinates in the global x- and y-direction.

Degrees of Freedom

1 = u displacement (x-direction).

2 = v displacement (y-direction).

Distributed Loading

There are no distributed load types available for this element.

Output of Strain

The two strain components are given at the element integration points. They are determined by the relative displacements between the top and bottom edge and are given in the local element system:

$$1 = \underset{\sim}{u}_{top} - \underset{\sim}{u}_{bottom}$$

$$2 = \underset{\sim}{v}_{top} - \underset{\sim}{v}_{bottom}$$

Output of Stress

The output of stress components is similar to the output of strain components.

Transformation

The global degrees of freedom can be transformed into local directions.

Tying

Use the TYING model definition option or the UFORMSN user subroutine.

Output Points

Output occurs at the element integration points.

Updated Lagrange Procedure and Finite Strain Plasticity

The cohesive material model for this element does not support plasticity. When used in a nonlinear analysis, either in the total or updated Lagrange formulation, the local element coordinate system is updated based on the deformed element configuration.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 220. See Element 220 for more details.

Element 187

Eight-node Planar Interface Element

Element type 187 is a mechanical eight-node planar interface element, which can be used to simulate the onset and progress of delamination.

The connectivity of the element is shown in Figure 3-267. The element is typically used to model the interface between different materials, where nodes 1, 5, and 2 correspond to one side of the interface (called the bottom) and nodes 3, 7, and 4 to the other (called the top). The stress components of the element are one normal and one shear traction, which are expressed with respect to the local coordinate system, $(\underset{\sim}{v}_1, \underset{\sim}{v}_2)$, indicated in Figure 3-267. The corresponding deformations are the relative displacements between the top and the bottom edge of the element. The element is allowed to be infinitely thin, in which case the edges 1-5-2 and 3-7-4 coincide.

The constitutive behavior of the element is defined via the COHESIVE model definition option.

Because the element is thin, there is no mass associated with the element. It should be noted that there is also no geometric or initial stress stiffening.

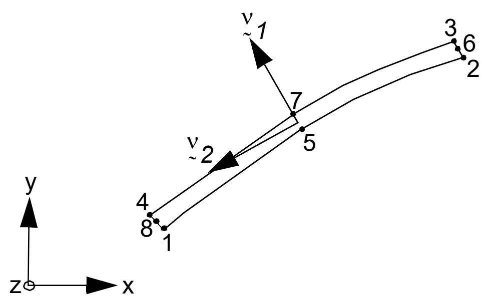


Figure 3-267 Element Type 187: Connectivity and Local Coordinate System

Geometric Basis

In each integration point, the first element base vector $\underset{\sim}{v}_1$ is obtained by rotating the tangent vector to the line through the points halfway the nodes 1 and 4, 5 and 7, and 2 and 3, counterclockwise over 90 degrees. Together with the first element base vector and the global z-axis, the second element base vector system $\underset{\sim}{v}_2$ forms a right-hand system (see Figure 3-267).

The element stiffness matrix is integrated numerically using a three-point integration scheme. By default, the position of the integration points corresponds to the Gauss integration scheme. As an alternative, the Newton-Cotes/Lobatto scheme can be selected via the GEOMETRY option, in which case the integration points are located at the middle of the edges 1-4 and 2-3 and at the element centroid (see Figure 3-268). The latter scheme may be advantageous to accurately describe a stress gradient in cases where the interface is relatively stiff compared to the surrounding material.

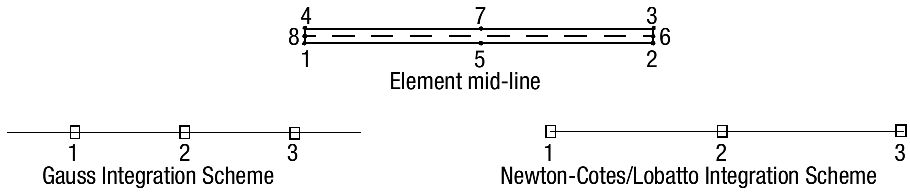


Figure 3-268 Element 187: Mid-line and Location of Integration Points

Quick Reference

Type 187

Quadratic, eight-node planar interface element.

Connectivity

Eight nodes per element. Node numbering must be counterclockwise (see Figure 3-267), so that the two sides of the interface are the element edges 1-5-2 and 3-7-4.

Note:	Nodes 6 and 8 are actually not needed in the element formulation; they appear only to make the element compatible with eight-node quadrilateral planar elements.

Geometry

The thickness is entered in the first data field. Default thickness is one. If the fifth data field is set to one, the Newton-Cotes/Lobatto integration scheme is used instead of the default Gauss integration scheme.

Coordinates

Two coordinates in the global x- and y-direction.

Degrees of Freedom

1 = u displacement (x-direction).
2 = v displacement (y-direction).

Distributed Loading

There are no distributed load types available for this element.

Output of Strain

The two strain components are given at the element integration points. They are determined by the relative displacements between the top and bottom edge and are given in the local element system:

$$1 = u_{top} - u_{bottom}$$

$$2 = v_{top} - v_{bottom}$$

Output of Stress

The output of stress components is similar to the output of strain components.

Transformation

The global degrees of freedom can be transformed into local directions.

Tying

Use the TYING model definition option or the UFORMSN user subroutine.

Output Points

Output occurs at the element integration points.

Updated Lagrange Procedure and Finite Strain Plasticity

The cohesive material model for this element does not support plasticity. When used in a nonlinear analysis, either in the total or updated Lagrange formulation, the local element coordinate system is updated based on the deformed element configuration.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 221. See Element 221 for more details.

Element 188

Eight-node Three-dimensional Interface Element

Element type 188 is a mechanical eight-node three-dimensional interface element, which can be used to simulate the onset and progress of delamination.

The connectivity of the element is shown in Figure 3-269. The element is typically used to model the interface between different materials, where nodes 1, 2, 3 and 4 correspond to one side (called the bottom) of the interface and nodes 5, 6, 7 and 8 to the other (called the top). The stress components of the element are one normal traction and two shear tractions, which are expressed with respect to the local coordinate system $(\underset{\sim}{v}_1, \underset{\sim}{v}_2, \underset{\sim}{v}_3)$, indicated in Figure 3-269. The corresponding deformations are the relative displacements between the top and the bottom face of the element. The element is allowed to be infinitely thin, in which case the faces 1-2-3-4 and 5-6-7-8 coincide.

The constitutive behavior of the element is defined via the COHESIVE model definition option.

Because the element volume is zero, there is no mass associated with the element. It should be noted that there is also no geometric or initial stress stiffening.

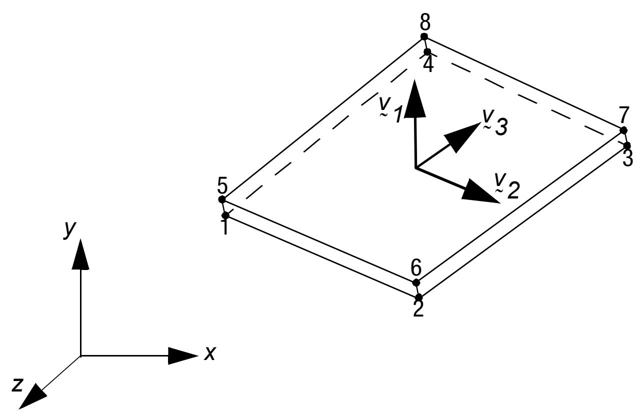


Figure 3-269 Element Type 188: Connectivity and Local Coordinate System

Geometric Basis

In order to define the local base vectors, at each integration point normalized tangent vectors to the curves with constant isoparametric coordinates are introduced (see Figure 3-270):

$$\underset{\sim}{t}_1 = \frac{\partial x}{\partial \xi} \Big/ \left\| \frac{\partial x}{\partial \xi} \right\| \text{ and } \underset{\sim}{t}_2 = \frac{\partial x}{\partial \eta} \Big/ \left\| \frac{\partial x}{\partial \eta} \right\|$$

where $\underset{\sim}{x}$ is the position vector of a point on the element mid-plane.

Now the first element base vector $\underset{\sim}{v}_1$ is the local normal vector and is given by:

$$\underset{\sim}{v}_1 = \frac{\underset{\sim}{t}_1 \times \underset{\sim}{t}_2}{\left\| \underset{\sim}{t}_1 \times \underset{\sim}{t}_2 \right\|} .$$

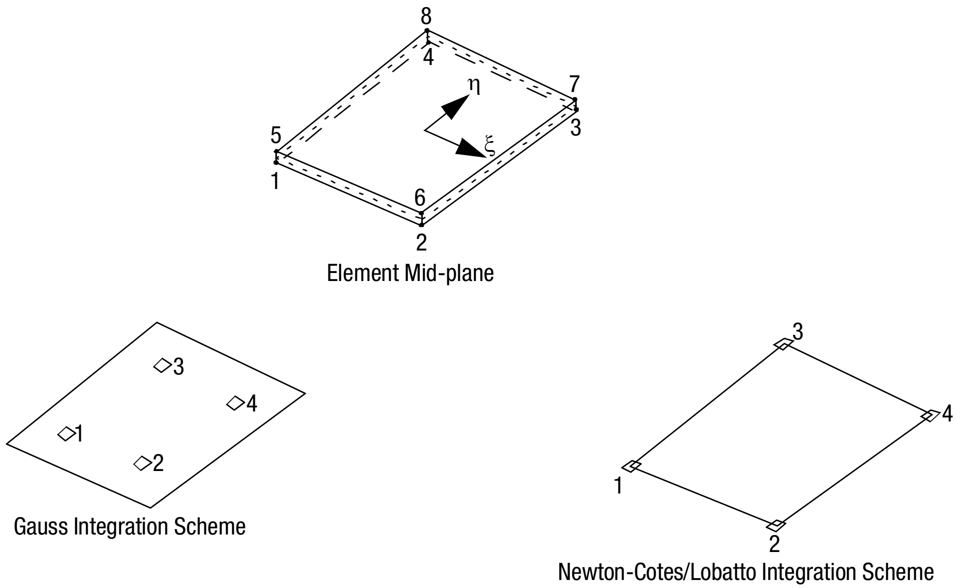


Figure 3-270 Element 188: Mid-plane and Location of Integration Points

The in-plane element base vector $\mathbf{v}_3$ is constructed to be perpendicular to the plane spanned by the local normal vector $\mathbf{v}_1$ and the first mid-plane edge vector $\mathbf{e}_1$:

$$\mathbf{v}_3 = \frac{\mathbf{v}_1 \times \mathbf{e}_1}{\|\mathbf{v}_1 \times \mathbf{e}_1\|},$$

in which $\mathbf{e}_1$ is defined as:

$$\mathbf{e}_1 = \frac{1}{2}(\mathbf{x}^2 + \mathbf{x}^6) - \frac{1}{2}(\mathbf{x}^1 + \mathbf{x}^5),$$

where the superscript refers to a node number of the element.

Finally, the in-plane element base vector $\mathbf{v}_2$ is parallel to the plane of the local normal vector and the first min-plane edge and follows from the cross product of the former two element base vectors:

$$\mathbf{v}_2 = \mathbf{v}_3 \times \mathbf{v}_1 .$$

The element stiffness matrix is integrated numerically using a four-point integration scheme. By default, the position of the integration points corresponds to the Gauss integration scheme. As an alternative, the Newton/Cotes/Lobatto integration scheme can be selected via the GEOMETRY option, in which case the integration points are located at the middle of the edges 1-5, 2-6, 3-7, and 4-8 (see Figure 3-270). The latter scheme may be advantageous in cases where the interface is relatively stiff compared to the surrounding material.

Quick Reference

Type 188

Linear, eight-node three-dimensional interface element.

Connectivity

Eight nodes per element. Node numbering must be according to Figure 3-269, so that the two sides of the interface are the element faces 1-2-3-4 and 5-6-7-8.

Geometry

If the fifth data field is set to one, the Newton-Cotes/Lobatto integration scheme is used instead of the default Gauss integration scheme.

Coordinates

Three coordinates in the global x-, y- and z-direction.

Degrees of Freedom

1 = u displacement (x-direction).
2 = v displacement (y-direction).
3 = w displacement (z-direction).

Distributed Loading

There are no distributed load types available for this element.

Output of Strain

The three strain components are given at the element integration points. They are determined by the relative displacements between the top and bottom face and are given in the local element system:

$$1 = \underset{\sim}{u}_{top} - \underset{\sim}{u}_{bottom}$$

$$2 = \underset{\sim}{v}_{top} - \underset{\sim}{v}_{bottom}$$

$$3 = \underset{\sim}{w}_{top} - \underset{\sim}{w}_{bottom}$$

Output of Stress

1 = σ_n - normal stress
2 = τ_1 - shear stress
3 = τ_2 - shear stress

Transformation

The global degrees of freedom can be transformed into local directions.

Tying

Use the TYING model definition option or the UFORMSN user subroutine.

Output Points

Output occurs at the element integration points.

Updated Lagrange Procedure and Finite Strain Plasticity

The cohesive material model for this element does not support plasticity. When used in a nonlinear analysis, either in the total or updated Lagrange formulation, the local element coordinate system is updated based on the deformed element configuration.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 222. See Element 222 for more details.

Element 189

Twenty-node Three-dimensional Interface Element

Element type 189 is a mechanical twenty-node three-dimensional interface element, which can be used to simulate the onset and progress of delamination. The connectivity of the element is shown in Figure 3-271. The element is typically used to model the interface between different materials, where nodes 1, 9, 2, 10, 3, 11, 4, and 12 correspond to one side (called the bottom) of the interface and nodes 5, 13, 6, 14, 7, 15, 8, and 16 to the other (called the top). The stress components of the element are one normal traction and two shear tractions, which are expressed with respect to the local coordinate system, indicated in Figure 3-271.

The corresponding deformations are the relative displacements between the top and the bottom face of the element. The element is allowed to be infinitely thin, in which case the faces 1-9-2-10-3-11-4-12 and 5-13-6-14-7-15-8-16 coincide.

The constitutive behavior of the element is defined via the COHESIVE model definition option.

Because the element is thin, there is no mass associated with the element. It should be noted that there is also no geometric or initial stress stiffening.

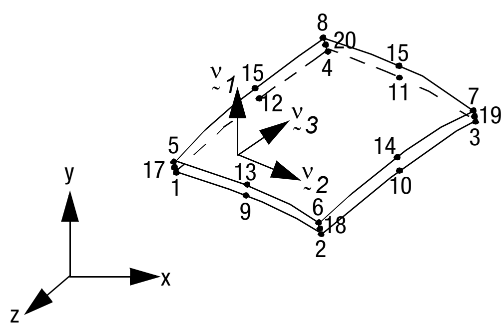


Figure 3-271 Element Type 189: Connectivity and Local Coordinate System

Geometric Basis

In order to define the local base vectors, at each integration point normalized tangent vectors to the curves with constant isoparametric coordinates are introduced (see Figure 3-272):

$$\underset{\sim}{t}_1 = \frac{\partial x}{\partial \xi} / \left\| \frac{\partial x}{\partial \xi} \right\| \text{ and } \underset{\sim}{t}_2 = \frac{\partial x}{\partial \eta} / \left\| \frac{\partial x}{\partial \eta} \right\|$$

where $\underset{\sim}{x}$ is the position vector of a point on the element mid-plane.

Now the first element base vector $\underset{\sim}{v}_1$ is the local normal vector and is given by:

$$\underset{\sim}{v}_1 = \frac{\underset{\sim}{t}_1 \times \underset{\sim}{t}_2}{\left\| \underset{\sim}{t}_1 \times \underset{\sim}{t}_2 \right\|} .$$

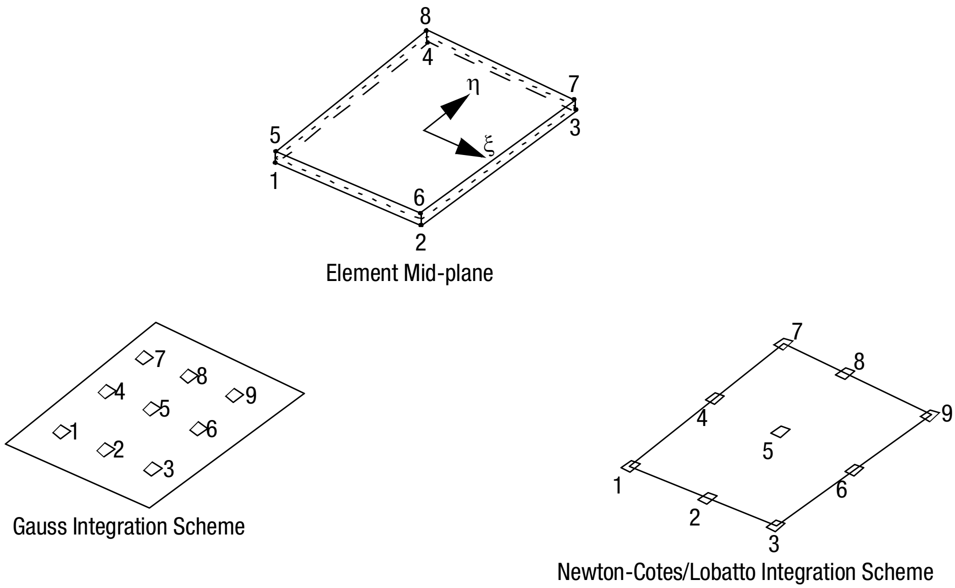


Figure 3-272 Element 189: Mid-plane and Location of Integration Points

The in-plane element base vector v_3 is constructed to be perpendicular to the plane spanned by the local normal vector v_1 and the first mid-plane edge vector t_1:

$$v_3 = \frac{v_1 \times t_1}{\|v_1 \times t_1\|},$$

in which t_1 is defined as:

$$t_1 = \frac{1}{2}(x^2 + x^6) - \frac{1}{2}(x^1 + x^5),$$

where the superscript refers to a node number of the element.

Finally, the in-plane element base vector v_2 is parallel to the plane of the local normal vector and the first min-plane edge and follows from the cross product of the former two element base vectors:

$$v_2 = v_3 \times v_1 .$$

The element stiffness matrix is integrated numerically using a nine-point integration scheme. By default, the position of the integration points corresponds to the Gauss integration scheme. As an alternative, the Newton/Cotes/Lobatto integration scheme can be selected via the GEOMETRY option, in which case the integration points are located at the middle of the edges 1-5, 2-6, 3-7, and 4-8; between nodes 9-13, 10-14, 11-15, 12-16, and at the centroid xx (see Figure 3-272). The latter scheme may be advantageous in cases where the interface is relatively stiff compared to the surrounding material.

Quick Reference

Type 189

Quadratic, twenty-node three-dimensional interface element.

Connectivity

Twenty nodes per element. Node numbering must be according to Figure 3-271, so that the two sides of the interface are the element faces 1-9-2-10-3-11-4-12 and 5-13-6-14-7-15-8-16.

Note: Nodes 17, 18, 19, and 20 are actually not needed in the element formulation; they only appear to make the element compatible with twenty-node hexahedral volume elements.

Geometry

If the fifth data field is set to one, the Newton-Cotes/Lobatto integration scheme is used instead of the default Gauss integration scheme.

Coordinates

Three coordinates in the global x-, y- and z-direction.

Degrees of Freedom

1 = u displacement (x-direction).
2 = v displacement (y-direction).
3 = w displacement (z-direction).

Distributed Loading

There are no distributed load types available for this element.

Output of Strain

The three strain components are given at the element integration points. They are determined by the relative displacements between the top and bottom face and are given in the local element system:

$$1 = \underset{\sim}{u}_{top} - \underset{\sim}{u}_{bottom}$$

$$2 = \underset{\sim}{v}_{top} - \underset{\sim}{v}_{bottom}$$

$$3 = \underset{\sim}{w}_{top} - \underset{\sim}{w}_{bottom}$$

Output of Stress

The output of stress components is similar to the output of strain components.

Transformation

The global degrees of freedom can be transformed into local directions.

Tying

Use the TYING model definition option or the UFORMSN user subroutine.

Output Points

Output occurs at the element integration points.

Updated Lagrange Procedure and Finite Strain Plasticity

The cohesive material model for this element does not support plasticity. When used in a nonlinear analysis, either in the total or updated Lagrange formulation, the local element coordinate system is updated based on the deformed element configuration.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 223. See Element 223 for more details.

Element 190

Four-node Axisymmetric Interface Element

Element type 190 is a mechanical four-node axisymmetric interface element, which can be used to simulate the onset and progress of delamination.

The connectivity of the element is shown in Figure 3-273. The element is typically used to model the interface between different materials, where nodes 1 and 2 correspond to one side of the interface (called the bottom) and nodes 3 and 4 to the other (called the top). The stress components of the element are one normal and one shear traction, which are expressed with respect to the local coordinate system $(\underset{\sim}{v}_1, \underset{\sim}{v}_2)$, indicated in Figure 3-273. The corresponding deformations are the relative displacements between the top and the bottom edge of the element. The element is allowed to be infinitely thin, in which case the edges 1-2 and 3-4 coincide.

The constitutive behavior of the element is defined via the COHESIVE model definition option.

Because the element is thin, there is no mass associated with the element. It should be noted that there is also no geometric or initial stress stiffening.

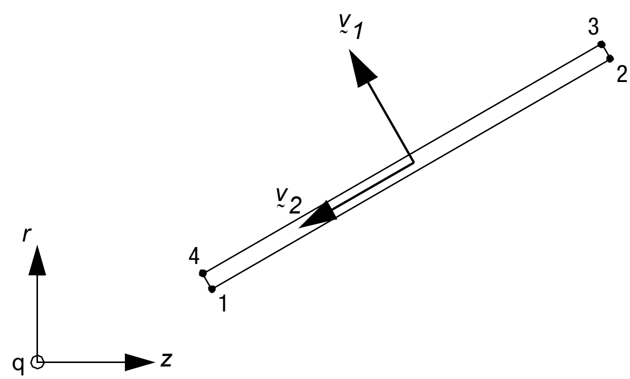


Figure 3-273 Element Type 190: Connectivity and Local Coordinate System

Geometric Basis

The first element base vector $\underset{\sim}{v}_1$ is obtained by rotating the direction vector from the middle of edge 1-4 to the middle of edge 2-3 counterclockwise over 90 degrees. Together with the first element base vector and the global θ-axis, the second element base vector $\underset{\sim}{v}_2$ forms a right-hand system (see Figure 3-273). The element stiffness matrix is integrated numerically using a two-point integration scheme. By default, the position of the integration points corresponds to the Gauss integration scheme. As an alternative, a nodal lumping scheme can be selected via the GEOMETRY option, in which case the integration points are located at the middle of the edges 1-4 and 2-3 (see Figure 3-274). The latter scheme may be advantageous in cases where the interface is relatively stiff compared to the surrounding material.

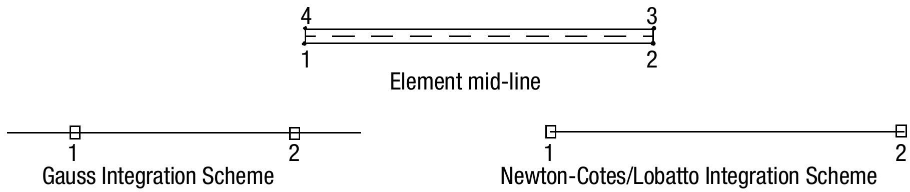


Figure 3-274 Element 190: Mid-line and Location of Integration Points

Quick Reference

Type 190

Linear, four-node axisymmetric interface element.

Connectivity

Four nodes per element. Node numbering must be counterclockwise (see Figure 3-273), so that the two sides of the interface are the element edges 1-2 and 3-4.

Geometry

If the fifth data field is set to one, the Newton-Cotes/Lobatto integration scheme is used instead of the default Gauss integration scheme.

Coordinates

Two coordinates in the global z- and r-direction.

Degrees of Freedom

1 = u displacement (z-direction).
2 = v displacement (r-direction).

Distributed Loading

There are no distributed load types available for this element.

Output of Strain

The two strain components are given at the element integration points. They are determined by the relative displacements between the top and bottom edge and are given in the local element system:

$$1 = \underset{\sim}{u}_{top} - \underset{\sim}{u}_{bottom}$$

$$2 = \underset{\sim}{v}_{top} - \underset{\sim}{v}_{bottom}$$

Output of Stress

The output of stress components is similar to the output of strain components.

Transformation

The global degrees of freedom can be transformed into local directions.

Tying

Use the TYING model definition option or the UFORMSN user subroutine.

Output Points

Output occurs at the element integration points.

Updated Lagrange Procedure and Finite Strain Plasticity

The cohesive material model for this element does not support plasticity. When used in a nonlinear analysis, either in the total or updated Lagrange formulation, the local element coordinate system is updated based on the deformed element configuration.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 224. See Element 224 for more details.

Element 191

Eight-node Axisymmetric Interface Element

Element type 191 is a mechanical eight-node axisymmetric interface element, which can be used to simulate the onset and progress of delamination.

The connectivity of the element is shown in Figure 3-275. The element is typically used to model the interface between different materials, where nodes 1, 5, and 2 correspond to one side of the interface (called the bottom) and nodes 3, 7, and 4 to the other (called the top). The stress components of the element are one normal and one shear traction, which are expressed with respect to the local coordinate system, indicated in Figure 3-275. The corresponding deformations are the relative displacements between the top and the bottom edge of the element. The element is allowed to be infinitely thin, in which case the edges 1-5-2 and 3-7-4 coincide.

The constitutive behavior of the element is defined via the COHESIVE model definition option.

Because the element is thin, there is no mass associated with the element. It should be noted that there is also no geometric or initial stress stiffening.

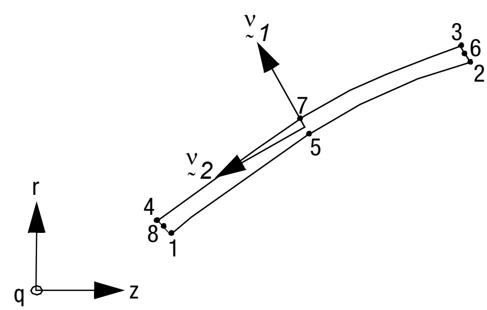


Figure 3-275 Element Type 191: Connectivity and Local Coordinate System

Geometric Basis

In each integration point, the first element base vector $\underset{\sim}{v}_1$ is obtained by rotating the tangent vector to the line through the points halfway the nodes 1 and 4, 5 and 7, and 2 and 3, counterclockwise over 90 degrees. Together with the first element base vector and the global θ-axis, the second element base vector $\underset{\sim}{v}_2$ forms a right-hand system (see Figure 3-275).

The element stiffness matrix is integrated numerically using a three-point integration scheme. By default, the position of the integration points corresponds to the Gauss integration scheme. As an alternative, the Newton- Cotes/Lobatto scheme can be selected via the GEOMETRY option, in which case the integration points are located at the middle of the edges 1-4 and 2-3 and at the element centroid (see Figure 3-276). The latter scheme may be advantageous to accurately describe a stress gradient in cases where the interface is relatively stiff compared to the surrounding material.

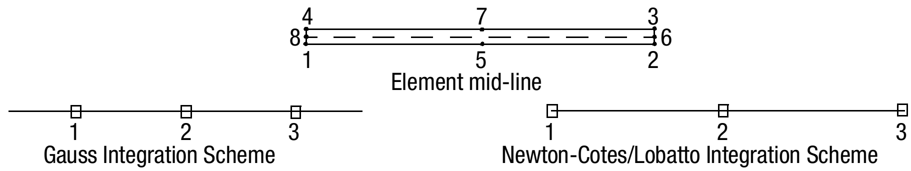


Figure 3-276 Element 191: Mid-line and Location of Integration Points

Quick Reference

Type 191

Quadratic, eight-node axisymmetric interface element.

Connectivity

Eight nodes per element. Node numbering must be counterclockwise (see Figure 3-275), so that the two sides of the interface are the element edges 1-5-2 and 3-7-4.

Note:	Nodes 6 and 8 are actually not needed in the element formulation; they appear only to make the element compatible with eight-node quadrilateral axisymmetric elements.

Geometry

If the fifth data field is set to one, the Newton-Cotes/Lobatto integration scheme is used instead of the default Gauss integration scheme.

Coordinates

Two coordinates in the global z- and r-direction.

Degrees of Freedom

1 = u displacement (z-direction).
2 = v displacement (r-direction).

Distributed Loading

There are no distributed load types available for this element.

Output of Strain

The two strain components are given at the element integration points. They are determined by the relative displacements between the top and bottom edge and are given in the local element system:

$$1 = u_{top} - u_{bottom}$$

$$2 = v_{top} - v_{bottom}$$

Output of Stress

The output of stress components is similar to the output of strain components.

Transformation

The global degrees of freedom can be transformed into local directions.

Tying

Use the TYING model definition option or the UFORMSN user subroutine.

Output Points

Output occurs at the element integration points.

Updated Lagrange Procedure and Finite Strain Plasticity

The cohesive material model for this element does not support plasticity. When used in a nonlinear analysis, either in the total or updated Lagrange formulation, the local element coordinate system is updated based on the deformed element configuration.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 225. See Element 225 for more details.

Element 192

Six-node Three-dimensional Interface Element

Element type 192 is a mechanical six-node three-dimensional interface element, which can be used to simulate the onset and progress of delamination.

The connectivity of the element is shown in Figure 3-277. The element is typically used to model the interface between different materials, where nodes 1, 2, and 3 correspond to one side of the interface (called the bottom) and nodes 4, 5, and 6 to the other (called the top). The stress components of the element are one normal traction and two shear tractions, which are expressed with respect to the local coordinate system $(\underset{\sim}{v}_1, \underset{\sim}{v}_2, \underset{\sim}{v}_3)$, indicated in Figure 3-277. The corresponding deformations are the relative displacements between the top and the bottom face of the element. The element is allowed to be infinitely thin, in which case the faces 1-2-3 and 4-5-6 coincide.

The constitutive behavior of the element is defined via the COHESIVE model definition option.

Because the element is thin, there is no mass associated with the element. It should be noted that there is also no geometric or initial stress stiffening.

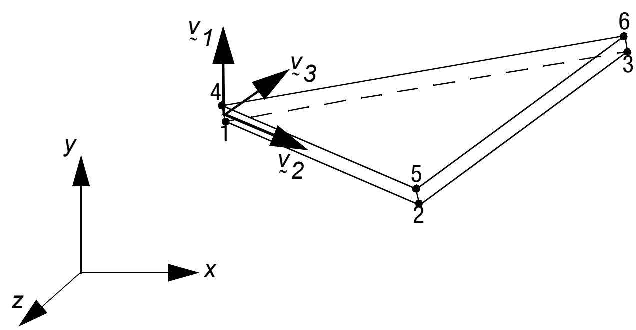


Figure 3-277 Element Type 192: Connectivity and Local Coordinate System

Geometric Basis

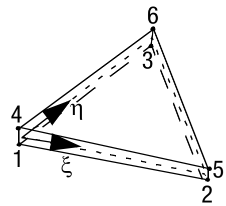
In order to define the local base vectors, at each integration point normalized tangent vectors to the curves with constant isoparametric coordinates are introduced (see Figure 3-278):

$$\underset{\sim}{t}_1 = \frac{\partial \underset{\sim}{x}}{\partial \xi} \Big/ \left\| \frac{\partial \underset{\sim}{x}}{\partial \xi} \right\| \quad \text{and} \quad \underset{\sim}{t}_2 = \frac{\partial \underset{\sim}{x}}{\partial \eta} \Big/ \left\| \frac{\partial \underset{\sim}{x}}{\partial \eta} \right\|$$

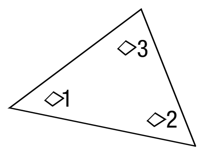
where $\underset{\sim}{x}$ is the position vector of a point on the element mid-plane.

Now the first element base vector $\underset{\sim}{v}_1$ is the local normal vector and is given by:

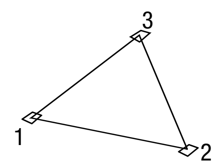
$$\underset{\sim}{v}_1 = \frac{\underset{\sim}{t}_1 \times \underset{\sim}{t}_2}{\left\| \underset{\sim}{t}_1 \times \underset{\sim}{t}_2 \right\|} .$$



Element Mid-plane



Gauss Integration Scheme



Newton-Cotes/Lobatto Integration Scheme

Figure 3-278 Element 192: Mid-plane and Location of Integration Points

The in-plane element base vector v_3 is constructed to be perpendicular to the plane spanned by the local normal vector v_1 and the first mid-plane edge vector e_1:

$$v_3 = \frac{v_1 \times e_1}{\| v_1 \times e_1 \|},$$

in which e_1 is defined as:

$$e_1 = \frac{1}{2}(x^2 + x^6) - \frac{1}{2}(x^1 + x^5),$$

where the superscript refers to a node number of the element.

Finally, the in-plane element base vector v_2 is parallel to the plane of the local normal vector and the first min-plane edge and follows from the cross product of the former two element base vectors:

$$v_2 = v_3 \times v_1 .$$

The element stiffness matrix is integrated numerically using a three-point integration scheme. By default, the position of the integration points corresponds to the Gauss integration scheme. As an alternative, the Newton/Cotes/Lobatto integration scheme can be selected via the GEOMETRY option, in which case the integration points are located at the middle of the edges 1-4, 2-5, and 3-6 (see Figure 3-278). The latter scheme may be advantageous in cases where the interface is relatively stiff compared to the surrounding material.

Quick Reference

Type 192

Linear, six-node three-dimensional interface element.

Connectivity

Six nodes per element. Node numbering must be according to Figure 3-277, so that the two sides of the interface are the element faces 1-2-3 and 4-5-6.

Geometry

If the fifth data field is set to one, the Newton-Cotes/Lobatto integration scheme is used instead of the default Gauss integration scheme.

Coordinates

Three coordinates in the global x-, y- and z-direction.

Degrees of Freedom

1 = u displacement (x-direction).
2 = v displacement (y-direction).
3 = w displacement (z-direction).

Distributed Loading

There are no distributed load types available for this element.

Output of Strain

The three strain components are given at the element integration points. They are determined by the relative displacements between the top and bottom face and are given in the local element system:

$$1 = \underset{\sim}{u}_{top} - \underset{\sim}{u}_{bottom}$$

$$2 = \underset{\sim}{v}_{top} - \underset{\sim}{v}_{bottom}$$

$$3 = \underset{\sim}{w}_{top} - \underset{\sim}{w}_{bottom}$$

Output of Stress

The output of stress components is similar to the output of strain components.

Transformation

The global degrees of freedom can be transformed into local directions.

Tying

Use the TYING model definition option or the UFORMSN user subroutine.

Output Points

Output occurs at the element integration points.

Updated Lagrange Procedure and Finite Strain Plasticity

The cohesive material model for this element does not support plasticity. When used in a nonlinear analysis, either in the total or updated Lagrange formulation, the local element coordinate system is updated based on the deformed element configuration.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 226. See Element 226 for more details.

Element 193

Fifteen-node Three-dimensional Interface Element

Element type 192 is a mechanical fifteen-node three-dimensional interface element, which can be used to simulate the onset and progress of delamination.

The connectivity of the element is shown in Figure 3-279. The element is typically used to model the interface between different materials, where nodes 1, 7, 2, 8, 3, and 9 correspond to one side of the interface (called the bottom) and nodes 4, 10, 5, 11, 6, and 12 to the other (called the top). The stress components of the element are one normal traction and two shear traductions, which are expressed with respect to the local coordinate system $(\underset{\sim}{v}_1, \underset{\sim}{v}_2, \underset{\sim}{v}_3)$, indicated in Figure 3-279.

The corresponding deformations are the relative displacements between the top and the bottom face of the element. The element is allowed to be infinitely thin, in which case the faces 1-7-2-8-3-9 and 4-10-5-11-6-12 coincide.

The constitutive behavior of the element is defined via the COHESIVE model definition option.

Because the element is thin, there is no mass associated with the element. It should be noted that there is also no geometric or initial stress stiffening.

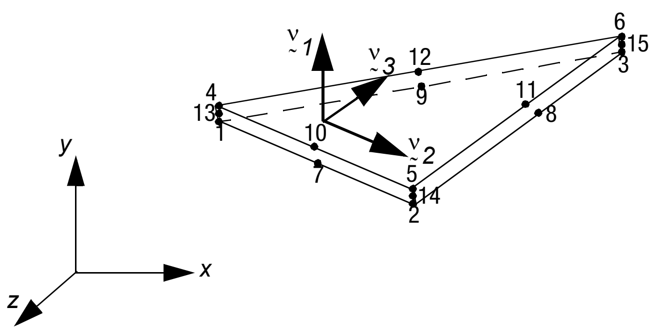


Figure 3-279 Element Type 193: Connectivity and Local Coordinate System

Geometric Basis

In order to define the local base vectors, at each integration point normalized tangent vectors to the curves with constant isoparametric coordinates are introduced (see Figure 3-280):

$$\underset{\sim}{t}_1 = \frac{\partial x}{\partial \xi} \Big/ \left\| \frac{\partial x}{\partial \xi} \right\| \quad \text{and} \quad \underset{\sim}{t}_2 = \frac{\partial x}{\partial \eta} \Big/ \left\| \frac{\partial x}{\partial \eta} \right\|$$

where $\underset{\sim}{x}$ is the position vector of a point on the element midplane.

Now the first element base vector $\underset{\sim}{v}_1$ is the local normal vector and is given by:

$$\underset{\sim}{v}_1 = \frac{\underset{\sim}{t}_1 \times \underset{\sim}{t}_2}{\left\| \underset{\sim}{t}_1 \times \underset{\sim}{t}_2 \right\|} .$$

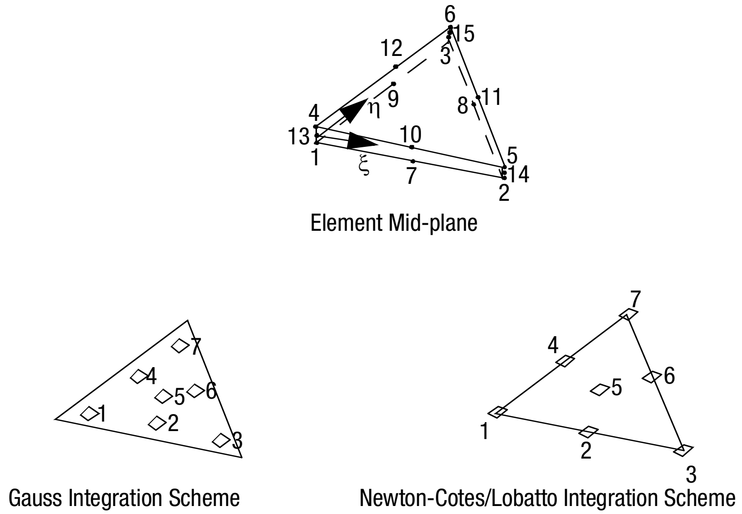


Figure 3-280 Element 193: Mid-plane and Location of Integration Points

The in-plane element base vector $\mathbf{v}_3$ is constructed to be perpendicular to the plane spanned by the local normal vector $\mathbf{v}_1$ and the first mid-plane edge vector $\mathbf{e}_1$:

$$\mathbf{v}_3 = \frac{\mathbf{v}_1 \times \mathbf{e}_1}{\|\mathbf{v}_1 \times \mathbf{e}_1\|},$$

in which $\mathbf{e}_1$ is defined as:

$$\mathbf{e}_1 = \frac{1}{2}(\mathbf{x}^2 + \mathbf{x}^6) - \frac{1}{2}(\mathbf{x}^1 + \mathbf{x}^5),$$

where the superscript refers to a node number of the element.

Finally, the in-plane element base vector $\mathbf{v}_2$ is parallel to the plane of the local normal vector and the first min-plane edge and follows from the cross product of the former two element base vectors:

$$\mathbf{v}_2 = \mathbf{v}_3 \times \mathbf{v}_1 .$$

The element stiffness matrix is integrated numerically using a seven-point integration scheme. By default, the position of the integration points corresponds to the Gauss integration scheme. As an alternative, the Newton/Cotes/Lobatto integration scheme can be selected via the GEOMETRY option, in which case the integration points are located at the middle of the edges 1 and 4, 7 and 10, 2 and 5, 8 and 11, 3 and 6, 12 and 15 and at the element centroid (see Figure 3-280). The latter scheme may be advantageous in cases where the interface is relatively stiff compared to the surrounding material.

Quick Reference

Type 193

Quadratic, fifteen-node three-dimensional interface element.

Connectivity

Fifteen nodes per element. Node numbering must be according to Figure 3-279, so that the two sides of the interface are the element faces 1-7-2-8-3-9 and 4-10-5-11-6-12.

Geometry

If the fifth data field is set to one, the Newton-Cotes/Lobatto integration scheme is used instead of the default Gauss integration scheme.

Coordinates

Three coordinates in the global x-, y- and z-direction.

Degrees of Freedom

1 = u displacement (x-direction).
2 = v displacement (y-direction).
3 = w displacement (z-direction).

Distributed Loading

There are no distributed load types available for this element.

Output of Strain

The three strain components are given at the element integration points. They are determined by the relative displacements between the top and bottom face and are given in the local element system:

$$1 = \underset{\sim}{u}_{top} - \underset{\sim}{u}_{bottom}$$

$$2 = \underset{\sim}{v}_{top} - \underset{\sim}{v}_{bottom}$$

$$3 = \underset{\sim}{w}_{top} - \underset{\sim}{w}_{bottom}$$

Output of Stress

The output of stress components is similar to the output of strain components.

Transformation

The global degrees of freedom can be transformed into local directions.

Tying

Use the TYING model definition option or the UFORMSN user subroutine.

Output Points

Output occurs at the element integration points.

Updated Lagrange Procedure and Finite Strain Plasticity

The cohesive material model for this element does not support plasticity. When used in a nonlinear analysis, either in the total or updated Lagrange formulation, the local element coordinate system is updated based on the deformed element configuration.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 227. See Element 227 for more details.

Element 194

2-D Generalized Spring - CBUSH/CFAST Element

This is a generalized 2-D spring-and-damper structural element that may exhibit nonlinear or frequency dependent behavior. The nominal material and geometric property values for this element are defined through either the PBUSH or PFAST model definition option.

Note:	In the remainder of this section, it refers to Cbush, but this element is also used in conjunction with the CFAST /PFAST options.

Geometric Basis

The local Cbush coordinate system can be defined in one of three ways:

- The Cbush system is attached to the element.
- The Cbush system is the global Cartesian coordinate system.
- The Cbush system is given by a user-defined coordinate system specified by the COORD SYSTEM option.

When the Cbush coordinate system is attached to the element, the local x-axis goes from node 1 to node 2. The local z axis is (0,0,1) and the local y axis forms a right-handed set with the local x and z.

Quick Reference

Type 194

2-D Cbush element.

Connectivity

Two nodes or one node.

When only one node is used for defining the connectivity, the Cbush element is assumed to be grounded. All ground degrees of freedom are taken as 0. In this case, the Cbush coordinate system can only be defined in the global Cartesian system or through the COORD SYSTEM option.

Geometric Properties

The geometric properties of the Cbush element are defined on the PBUSH option. The geometric properties are used to identify the coordinate system of the Cbush element and the nodal offsets for the Cbush element.

The location of the offset point can be specified in one of three ways:

- Offset point is defined to be along the length of the element. The location distance S is user specified and is measured from node 1.
- The components of the offset point are specified in the global Cartesian coordinate system.
- The components of the offset point are specified in a user-defined coordinate system specified by the COORD SYSTEM option.

The nodal offset vectors from each end node are internally calculated and stored. For large displacement analysis, these nodal offsets are updated independently depending on the rotation of each node.

The options for defining the local Cbush coordinate system and the offset point location are shown in Figure 3-281.

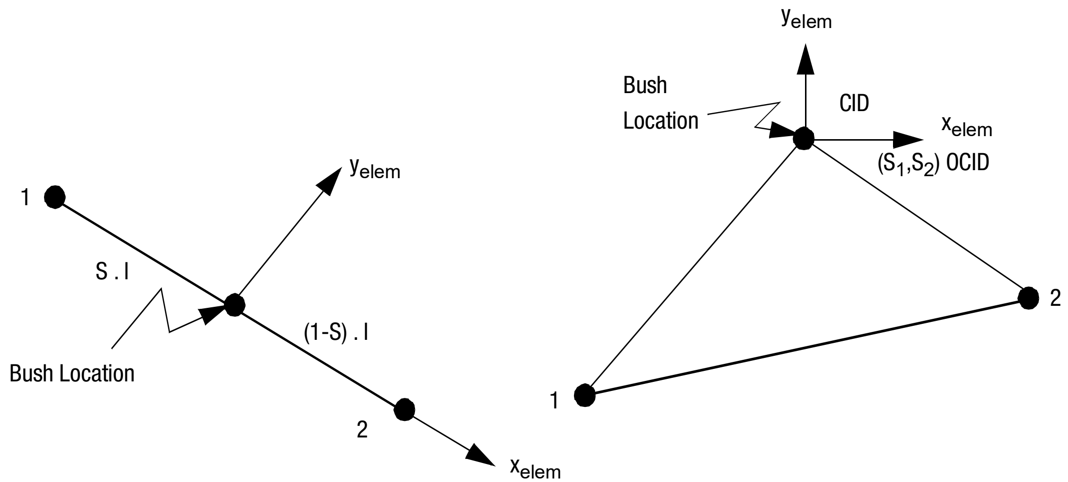


Figure 3-281 2-D Cbush Element

Coordinates

First two coordinates – (x, y) global Cartesian coordinates.

Degrees of Freedom

1 = u_x = global Cartesian x-direction displacement
2 = u_y = global Cartesian y-direction displacement
3 = θ_z = rotation about global z-direction

Tractions

The element does not support any form of distributed loading. Point loads and moments can be applied at the nodes.

Output Of Strains

Generalized strain components are as follows:

Local translational ε_{tx} strain
Local translational ε_{ty} strain
Local rotational ε_{rz} strain

Note that strain components are nonzero only if the strain recovery coefficients on the PBUSH option are nonzero. The strain components are available in the post file through post codes 1-3, 121-123, 301 or 401.

Output of Section Forces and Stresses

Section forces and stresses are output as:

Local T_x force. Local translational σ_{tx} stress
Local T_y force. Local translational σ_{ty} stress
Local M_z moment. Local rotational σ_{rz} stress

Note that stress components are nonzero only if the stress recovery coefficients on the PBUSH option are nonzero.

For the post file, the cbush section forces utilize existing post codes for beams. The three section forces are available in the post file through post codes 264 (Axial Force), 268 (Shear Force V_{yz}, 265 (Moment M_{xx}), respectively. The stress components are available in the post file through post codes 11-13, 41-43, 311 or 341. Real harmonic stress components are available in the post file through post codes 51-53 or 351. Imaginary harmonic stress components are available in the post file through post codes 61-63 or 361.

The direction cosines of the local Z axis of the cbush coordinate system are available in post codes 261 - 263.

Transformation

Displacements x and y can be transformed to local directions.

Output Points

Centroidal section of the element.

Large Displacement Formulation

When the local Cbush coordinate system is attached to the element, a co-rotational formulation is used for large displacement analysis. The direction cosines relating the local coordinate system to the global coordinate system are constantly updated based on the current axis joining the two end-node positions.

When the local Cbush coordinate system is specified through the global coordinate system or a user-specified COORD SYSTEM option, the Cbush coordinate system is not updated for large displacements.

The offset vectors at each end node are updated for large displacement analysis based on the rotations at the end nodes.

No stress stiffness matrix is calculated for large displacement analysis.

Field Analysis

In a heat transfer or coupled thermal-mechanical analysis, the thermal conductance matrix is calculated based on the provided thermal data on the PBUSH option. Similarly, for a Joule heating analysis, the electrical resistance matrix is calculated based on the provided electrical data on the PBUSH option. No distributed fluxes or films are supported.

Element 195

3-D Generalized Spring - CBUSH/CFAST Element

This is a generalized 3-D spring-and-damper structural element that may exhibit nonlinear or frequency dependent behavior. The nominal material and geometric property values for this element are defined through either the PBUSH or PFAST model definition option.

Note:	In the remainder of this section, it refers to Cbush, but this element is also used in conjunction with the CFAST /PFAST options.

Geometric Basis

The local Cbush coordinate system can be defined in one of three ways:

- The Cbush system is attached to the element.
- The Cbush system is the global Cartesian coordinate system.
- The Cbush system is given by a user-defined coordinate system specified by the COORD SYSTEM option.

When the Cbush coordinate system is attached to the element, the local x-axis goes from node 1 to node 2. To calculate the local y and z axis, an orientation vector $\vec{v}$ is prescribed from node 1 of the Cbush element. This orientation vector can be prescribed directly in component form or in terms of an extra node G0. In the former case, the components are in the displacement coordinate system at node 1. In the latter case, the orientation vector is given by the vector joining node 1 of the Cbush element to G0.

Quick Reference

Type 195

3-D Cbush element.

Connectivity

Two nodes or one node.

When only one node is used for defining the connectivity, the Cbush element is assumed to be grounded. All ground degrees of freedom are taken as 0. In this case, the Cbush coordinate system has to be defined in the global system or through the COORD SYSTEM option.

Geometric Properties

The geometric properties of the Cbush element are defined on the PBUSH option. The geometric properties are used to identify the coordinate system of the Cbush element and the nodal offsets for the cbush element.

The location of the offset point can be specified in one of three ways:

- Offset point is defined to be along the length of the element. The location distance S is user specified and is measured from node 1.
- The components of the offset point are specified in the global Cartesian coordinate system.

- The components of the offset point are specified in a user-defined coordinate system specified by the COORD SYSTEM option.

The nodal offset vectors from each end node are internally calculated and stored. For large displacement analysis, these nodal offsets are updated independently depending on the rotation of each node.

The options for defining the local Cbush coordinate system and the offset point location are shown in Figure 3-281.

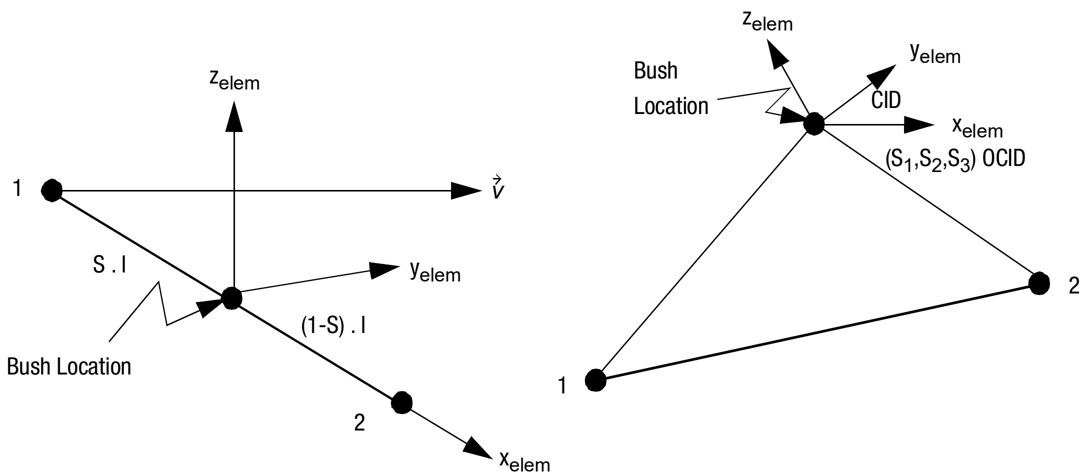


Figure 3-282 3-D Cbush Element

Coordinates

First three coordinates – (x, y, z) global Cartesian coordinates.

Degrees of Freedom

1 = u_x = global Cartesian x-direction displacement
2 = u_y = global Cartesian y-direction displacement
3 = u_z = global Cartesian z-direction displacement
4 = θ_x = rotation about global x-direction
5 = θ_y = rotation about global y-direction
6 = θ_z = rotation about global z-direction

Tractions

The element does not support any form of distributed loading. Point loads and moments can be applied at the nodes.

Output Of Strains

Generalized strain components are as follows:

Local translational ε_{tx} strain
Local translational ε_{ty} strain
Local translational ε_{tz} strain
Local rotational ε_{rx} strain
Local rotational ε_{ry} strain
Local rotational ε_{rz} strain

Note that strain components are nonzero only if the strain recovery coefficients on the PBUSH option are nonzero. The strain components are available in the post file through post codes 1-6, 121-126, 301 or 401.

Output of Section Forces and Stresses

Section forces and stresses are output as:

Local T_x force. Local translational σ_{tx} stress
Local T_y force. Local translational σ_{ty} stress
Local T_z force. Local translational σ_{tz} stress
Local M_x moment. Local rotational σ_{rx} stress
Local M_y moment. Local rotational σ_{ry} stress
Local M_z moment. Local rotational σ_{rz} stress

For the post file, the cbush section forces utilize existing post codes for beams. The six section forces are available in the post file through post codes 264 (Axial force for beams = Local T_x force for cbush), 267 (Shear force V_{xz} for beams = Local T_y force for cbush), 268 (Shear force V_{yz} for beams = Local T_z force for cbush), 269 (Torque for beams = Local M_x moment for cbush), 265 (Moment M_{xx} for beams = Local M_y moment for cbush), 266 (Moment M_{yy} for beams = Local M_z moment for cbush), respectively. The stress components are available in the post file through post codes 11-16, 41-46, 311, or 341. Real harmonic stress components are available in the post file through post codes 51-56 or 351. Imaginary harmonic stress components are available in the post file through post codes 61-66 or 361.

The direction cosines of the local Z axis of the cbush coordinate system are available in post codes 261 - 263.

Transformation

Displacements and rotations can be transformed to local directions.

Output Points

Centroidal section of the element.

Large Displacement Formulation

When the local Cbush coordinate system is attached to the element, a co-rotational formulation is used for large displacement analysis. The direction cosines relating the local coordinate system to the global coordinate system are constantly updated based on the current axis joining the two end-node positions.

When the local Cbush coordinate system is specified through the global coordinate system or a user-specified COORD SYSTEM option, the cbush coordinate system is constant and is not updated for large displacements.

The offset vectors at each end node are updated for large displacement analysis based on the rotations at the end nodes.

No stress stiffness matrix is calculated for large displacements.

Field Analysis

In a heat transfer or coupled thermal-mechanical analysis, the thermal conductance matrix is calculated based on the provided thermal data on the PBUSH option. Similarly, for a Joule heating analysis, the electrical resistance matrix is calculated based no the provided electrical data on the PBUSH option. No distributed fluxes or films are supported.

Element 196

Three-node, Bilinear Heat Transfer Membrane

Element 196 is a three-node, isoparametric, triangle written for heat transfer membrane applications. The element supports conduction over the element, but there is a uniform temperature through the thickness. If a thermal gradient is required in the thickness direction, one should use element type 50.

This element may also be used for electrostatic applications.

The stiffness of this element is formed using one-point integration.

This element is often used to input thermal boundary conditions only, with no conduction, similar to the MSC Nastran CHBDYG option. In that case, the thickness should be entered as zero.

Geometric Basis

The first two element base vectors lie in the plane of the three corner nodes (Figure 3-283). The first one (V_1) points from node 1 to node 2; the second one (V_2) is perpendicular to V_1 and lies on the same side as node 3. The third one (V_3) is determined such that V_1, V_2, and V_3 form a right-hand system.

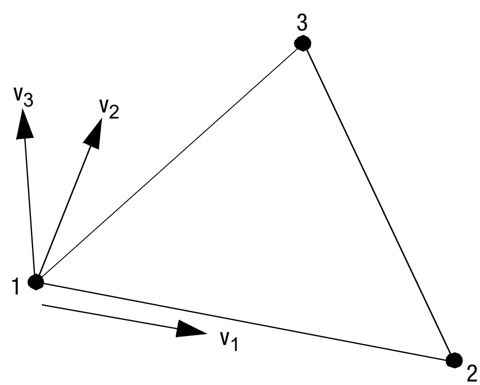


Figure 3-283 Form of Element 196

There is one integration point in the plane of the element at the centroid.

Quick Reference

Type 196

Three-node thermal membrane element in three-dimensional space.

Connectivity

Three node per element (see Figure 3-283).

Geometry

The thickness is input in the first data field (`EGEOM1`). The other two data fields are not used. If no thickness is entered, the element is only used to apply boundary conditions.

Coordinates

Three global Cartesian coordinates x, y, and z.

Degrees of Freedom

1 = temperature
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)

Fluxes

There are two type of distributed fluxes:

Volumetric Fluxes

Flux Type	Description
1	Uniform flux per unit volume on whole element.
3	Nonuniform flux per unit volume on the whole element; magnitude of flux is defined in the FLUX user subroutine.

Surface Fluxes

Flux Type	Description
2	Uniform flux per unit surface on bottom surface.
4	Nonuniform flux volume on the bottom surface; magnitude of flux is defined in the FLUX user subroutine.
5	Uniform flux per unit surface on top surface.
6	Nonuniform flux volume on the top surface; magnitude of flux is defined in the FLUX user subroutine.

Note: For conventional flux, it does not matter if one specified the top or bottom surface as the equivalent nodal flux are specified at the single degree of freedom. The specification of the top or bottom surface is relevant when the QVECT (with TABLE Input - Model Definition) option is used or when the top/bottom surface is specified for tradition simulations.

Point fluxes can also be applied at nodal degrees of freedom.

Films

Same specification as Fluxes .

Tying
Use the UFORMSN user subroutine.

Joule Heating
Capability is available.

Electrostatic
Capability is available.

Magnetostatic
Capability is not available.

Charge
Same specification as Fluxes .

Current
Same specification as Fluxes .

Output Points
Centroid.

Element 197

Six-node, Biquadratic Heat Transfer Membrane

Element 197 is a six-node, isoparametric, curved triangle written for heat transfer membrane applications. The element supports conduction over the element, but there is a uniform temperature through the thickness.

This element may also be used for electrostatic applications.

The stiffness of this element is formed using seven-point integration.

This element is often used to input thermal boundary conditions only, with no conduction, similar to the MSC Nastran CHBDYG option. In that case, the thickness should be entered as zero.

Geometric Basis

The first two element base vectors lie in the plane of the three corner nodes. The first one (V_1) points from node 1 to node 2; the second one (V_2) is perpendicular to V_1 and lies on the same side as node 3. The third one (V_3) is determined such that V_1, V_2, and V_3 form a right-hand system (Figure 3-284).

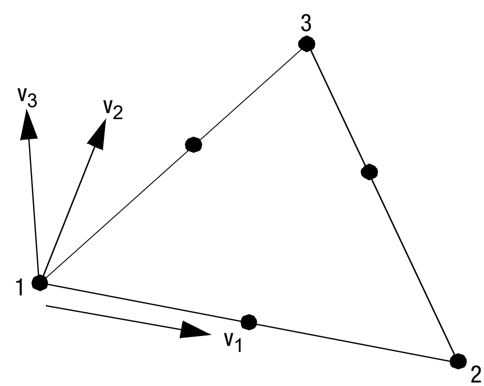


Figure 3-284 Local Coordinate System of Element 197

There are seven integration points in the plane of the element (Figure 3-285).

Quick Reference

Type 197

Six-node thermal membrane element in three-dimensional space.

Connectivity

Six nodes per element (see Figure 3-284).

The normal direction is based upon the counterclockwise numbering direction. The corner nodes are numbered first. The fourth node is located between nodes 1 and 2, the fifth node is located between nodes 2 and 3, etc.

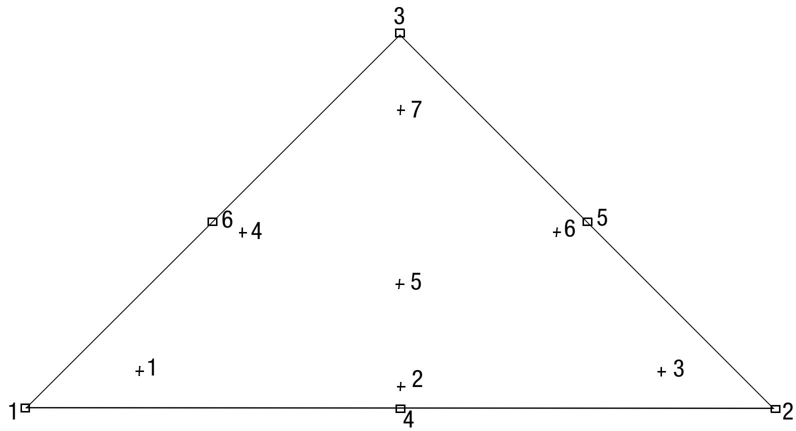


Figure 3-285 Integration Point Locations

Geometry

The thickness is input in the first data field (EGEOM1). The other two data fields are not used. If no thickness is entered, the element is only used to apply boundary conditions.

Coordinates

Three global Cartesian coordinates x, y, and z.

Degrees of Freedom

1 = temperature
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)

Fluxes

There are two types of distributed fluxes:

Volumetric Fluxes

Flux Type	Description
1	Uniform flux per unit volume on whole element.
3	Nonuniform flux per unit volume on the whole element; magnitude of flux is defined in the FLUX user subroutine.

Surface Fluxes

Flux Type	Description
2	Uniform flux per unit surface on bottom surface.
4	Nonuniform flux volume on the bottom surface; magnitude of flux is defined in the FLUX user subroutine.
5	Uniform flux per unit surface on top surface.
6	Nonuniform flux volume on the top surface; magnitude of flux is defined in the FLUX user subroutine.

Note: For conventional flux, it does not matter if one specified the top or bottom surface as the equivalent nodal flux are specified at the single degree of freedom. The specification of the top or bottom surface is relevant when the QVECT (with TABLE Input - Model Definition) option is used or when the top/bottom surface is specified for tradition simulations.

Point fluxes can also be applied at nodal degrees of freedom.

Films

Same specification as Fluxes .

Tying

Use the UFORMSN user subroutine.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Charge

Same specification as Fluxes .

Current

Same specification as Fluxes .

Output Points

Seven integration points or centroid.

Element 198 — Four-node, Isoparametric Heat Transfer Element

Element type 198 is a four-node, isoparametric, arbitrary quadrilateral written for heat transfer membrane applications. This element supports conduction over the element, but there is a uniform temperature through the thickness. If a thermal gradient is required in the thickness direction, one should use element type 85.

This element can also be used for electrostatic applications.

The stiffness of this element is formed using four-point Gaussian integration.

This element is often used to input thermal boundary conditions only, with no conduction, similar to the MSC Nastran CHBDYG option. In that case, the thickness should be entered as zero.

Geometric Basis

The element is defined geometrically by the (x,y,z) coordinates of the four corner nodes. Due to the bilinear interpolation, the surface forms a hyperbolic paraboloid which is allowed to degenerate to a plate. The element thickness is specified in the GEOMETRY option.

The thermal gradient output is given in local orthogonal surface directions V_1, V_2, and V_3 (Figure 3-286) which for the centroid are defined in the following way.

At the centroid, the vectors tangent to the curves with constant isoparametric coordinates are normalized.

$$t_1 = \frac{\partial x}{\partial \xi} / \left|\frac{\partial x}{\partial \eta}\right|, \; t_2 = \frac{\partial x}{\partial \xi} / \left|\frac{\partial x}{\partial \eta}\right|$$

Now a new basis is being defined as:

$$s = t_1 + t_2, \; d = t_1 + t_2$$

After normalizing these vectors by:

$$\bar{s} = s / \sqrt{2}|s|, \; \bar{d} = d / \sqrt{2}|d|$$

The local orthogonal directions are then obtained as:

$$V_1 = \bar{s} + \bar{d}, \; V_2 = \bar{s} - \bar{d}, \text{ and } V_3 = V_1 \times V_2$$

In this way, the vectors $\frac{\partial x}{\partial \xi}$, $\frac{\partial x}{\partial \eta}$ and V_1, V_2 have the same bisecting plane.

The local directions at the Gaussian integration points are found by projection of the centroid directions. Hence, if the element is flat, the directions at the Gauss points are identical to those at the centroid.

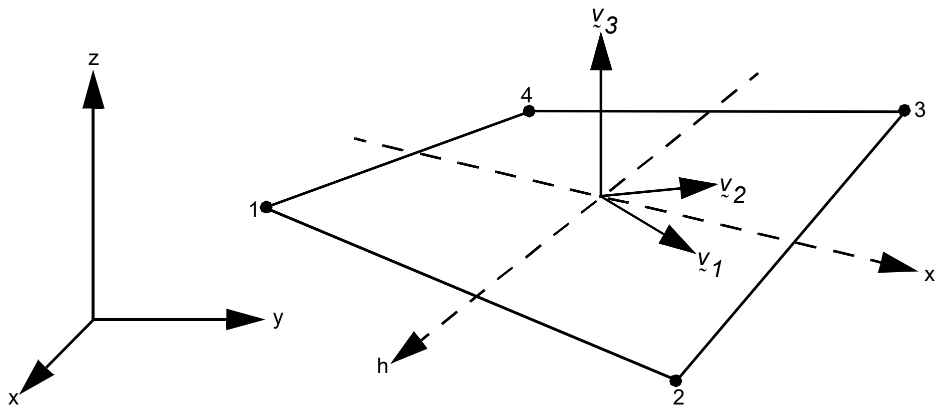


Figure 3-286 Form of Element 198

Quick Reference

Type 198

Four-node thermal membrane element (straight edge) in three-dimensional space.

Connectivity

Four nodes per element (see Figure 3-286).

The normal direction is based upon the counterclockwise numbering direction.

Geometry

The thickness is input in the first data field (EGEOM1). The other two data fields are not used. If no thickness is input, the element is only used to apply boundary conditions.

Coordinates

Three global Cartesian coordinates, x, y, z.

Degrees of Freedom

1 = temperature
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)

Fluxes

Two types of distributed fluxes:

Volumetric Fluxes

Flux Type	Description
1	Uniform flux per unit volume on whole element.
3	Nonuniform flux per unit volume on the whole element; magnitude of flux is defined in the FLUX user subroutine.

Surface Fluxes

Flux Type	Description
2	Uniform flux per unit surface on bottom surface.
4	Nonuniform flux volume on the bottom surface; magnitude of flux is defined in the FLUX user subroutine.
5	Uniform flux per unit surface on top surface.
6	Nonuniform flux volume on the top surface; magnitude of flux is defined in the FLUX user subroutine.

Note: For conventional flux, it does not matter if one specified the top or bottom surface as the equivalent nodal flux are specified at the single degree of freedom. The specification of the top or bottom surface is relevant when the QVECT option is used or when the top/bottom surface is specified for tradition simulations.

Point fluxes can be applied at nodal degrees of freedom.

Films

Same specification as Fluxes .

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Charge

Same specifications as Fluxes .

Current

Same specifications as Fluxes .

Output Points

Centroid or four Gaussian integration points (see Figure 3-287).

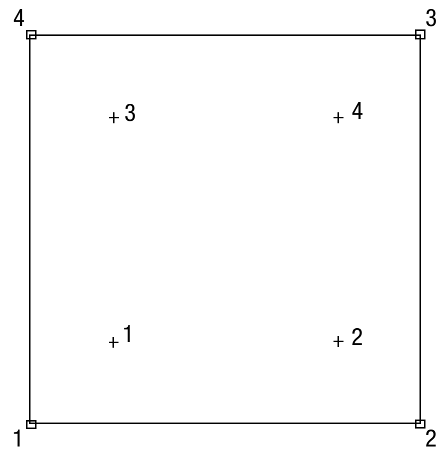


Figure 3-287 Gaussian Integration Points for Element 198

Element 199

Eight-node, Biquadratic Heat Transfer Membrane

Element 199 is an eight-node, isoparametric, arbitrary quadrilateral written for heat transfer membrane applications. The element supports conduction over the element, but there is a uniform temperature throughout the thickness. If a thermal gradient is required in the thickness direction, one should use element type 86.

This element may also be used for electrostatic applications.

The stiffness of this element is formed using nine-point Gaussian integration.

The element used biquadratic interpolation functions to represent the coordinates and the temperatures; hence, the thermal gradients have a linear variation. This allows for accurate representation of the temperature field.

This element is often used to input thermal boundary conditions only, with no conduction, similar to the MSC Nastran CHBDYG option. If that case, the thickness should be entered as zero.

Geometric Basis

The element is defined geometrically by the (x, y, z) coordinates of the eight corner nodes (Figure 3-288). The element thickness is specified in the GEOMETRY option.

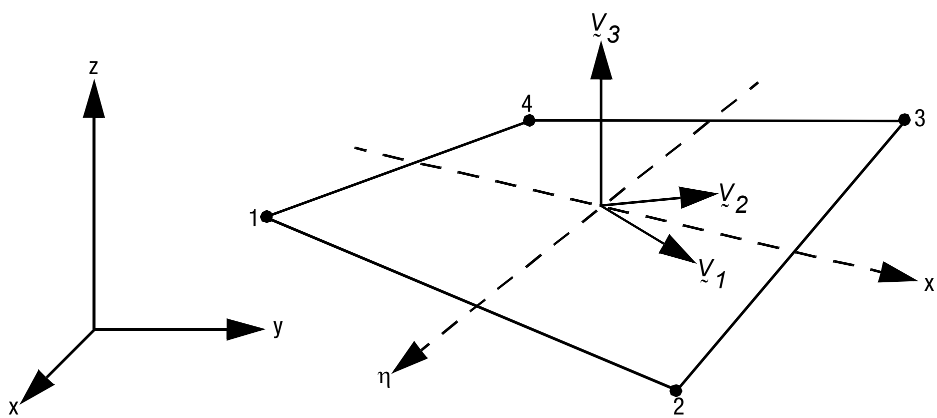


Figure 3-288 Local Coordinate System for Eight-node Thermal Membrane Element

The thermal gradient output is given in local orthogonal surface directions $\underset{\sim}{V}_1$, $\underset{\sim}{V}_2$, and $\underset{\sim}{V}_3$ (Figure 3-289) which for the centroid are defined in the following way.

At the centroid, the vectors tangent to the curves with constant isoparametric coordinates are normalized.

$$\underset{\sim}{t}_1 = \frac{\partial x}{\partial \xi} / \left|\frac{\partial x}{\partial \eta}\right|, \ \underset{\sim}{t}_2 = \frac{\partial x}{\partial \xi} / \left|\frac{\partial x}{\partial \eta}\right|$$

Now a new basis is being defined as:

$$\underset{\sim}{s} = \underset{\sim}{t}_1 + \underset{\sim}{t}_2, \ \underset{\sim}{d} = \underset{\sim}{t}_1 + \underset{\sim}{t}_2$$

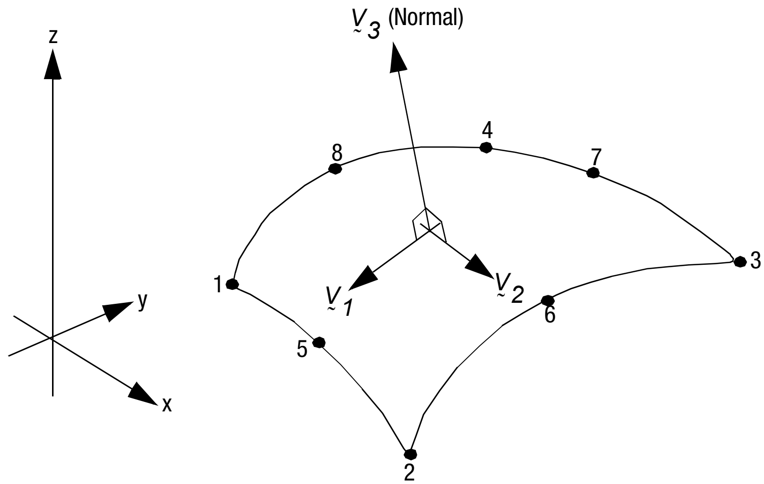


Figure 3-289 Form of Eight-node Thermal Membrane Element

After normalizing these vectors by:

$$\bar{s} = s / \sqrt{2}|s|\,,\ \bar{d} = d / \sqrt{2}|d|$$

The local orthogonal directions are then obtained as:

$$V_1 = \bar{s} + \bar{d}\,,\ V_2 = \bar{s} - \bar{d}\,,\text{ and } V_3 = V_1 \times V_2$$

In this way, the vectors $\frac{\partial x}{\partial \xi}$, $\frac{\partial x}{\partial \eta}$ and V_1, V_2 have the same bisecting plane.

The local directions at the Gaussian integration points are found by projection of the centroid directions. Hence, if the element is flat, the directions at the Gauss points are identical to those at the centroid.

Quick Reference

Type 199

Eight-node thermal membrane element in three-dimensional space.

Connectivity

Eight nodes per element (see Figure 3-289). The corner nodes are numbered first. The fifth node is located between 1 and 2, the sixth node is located between nodes 2 and 3, etc.

The normal direction is based upon the counterclockwise numbering direction.

Geometry

The thickness is input in the first data field (`EGEOM1`). The other two data fields are not used. If no thickness is entered, the element is only used to apply boundary conditions.

Coordinates

Three global Cartesian coordinates x, y, z.

Degrees of Freedom

1 = Temperature (heat transfer)
1 = Voltage, temperature (Joule Heating)
1 = Potential (electrostatic)

Fluxes

Two types of distributed fluxes.

Volumetric Fluxes

Flux Type	Description
1	Uniform flux per unit volume on whole element.
3	Nonuniform flux per unit volume on the whole element; magnitude of flux is defined in the FLUX user subroutine.

Surface Fluxes

Flux Type	Description
2	Uniform flux per unit surface on bottom surface.
4	Nonuniform flux volume on the bottom surface; magnitude of flux is defined in the FLUX user subroutine.
5	Uniform flux per unit surface on top surface.
6	Nonuniform flux volume on the top surface; magnitude of flux is defined in the FLUX user subroutine.

Note: For conventional flux, it does not matter if one specified the top or bottom surface as the equivalent nodal flux are specified at the single degree of freedom. The specification of the top or bottom surface is relevant when the QVECT option is used or when the top/bottom surface is specified for tradition simulations.

Point fluxes can also be applied at nodal degrees of freedom.

Films

Same specification as Fluxes .

Tying

Use the UFORMSN user subroutine.

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Current

Same specifications as Fluxes .

Charge

Same specifications as Fluxes .

Output Points

Centroid or nine Gaussian integration points (see Figure 3-290).

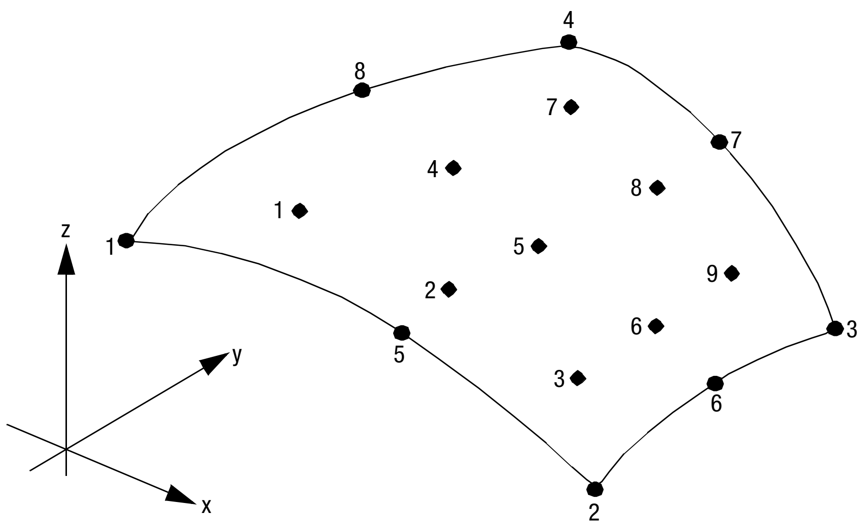


Figure 3-290 Gaussian Integration Points for Element 199

Element 200

Six-node, Biquadratic Isoparametric Membrane

Element 200 is a six-node, isoparametric, curved triangle written for membrane applications. As a membrane has no bending stiffness, the element is very unstable.

The stiffness of this element is formed using seven Gaussian integration points.

All constitutive models can be used with this element.

This element is usually used with the LARGE DISP parameter, in which case the (tensile) initial stress stiffness increases the rigidity of the element.

Geometric Basis

The first two element base vectors lie in the plane of the three corner nodes. The first one (V_1) points from node 1 to node 2; the second one (V_2) is perpendicular to V_1 and lies on the same side as node 3. The third one (V_3) is determined such that V_1, V_2, and V_3 form a right-hand system (Figure 3-291).

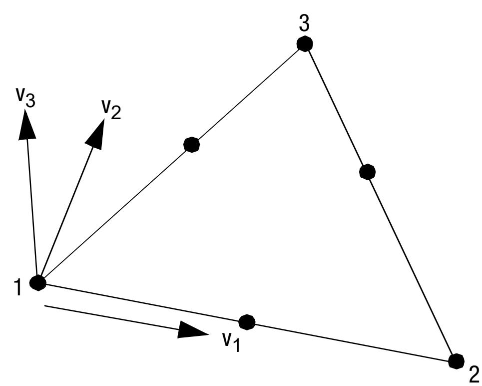


Figure 3-291 Local Coordinate System of Element 200

There are seven integration points in the plane of the element (Figure 3-292).

Quick Reference

Type 200

Six-node membrane element in three-dimensional space.

Connectivity

The normal direction is based upon the counterclockwise numbering direction. The corner nodes are numbered first. The fourth node is located between nodes 1 and 2, the fifth node is located between nodes 2 and 3, etc.

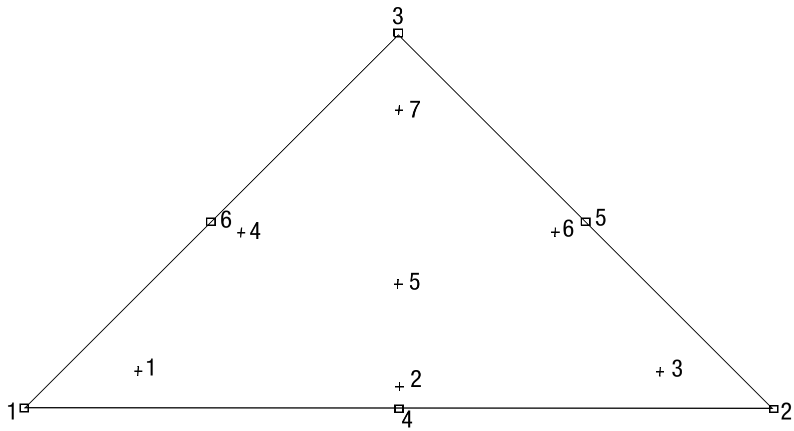


Figure 3-292 Integration Point Locations

Geometry

The thickness is input in the first data field (`EGEOM1`). The other two data fields are not used.

Coordinates

Three global Cartesian coordinates x, y, and z.

Degrees of Freedom

Three global degrees of freedom u, v, and w.

Tractions

Four distributed load types are available depending on the load type definition:

Load Type	Description
1	Gravity load, proportional to surface area, in negative global z-direction.
2	Pressure (load per unit area) positive when in the direction of normal. V_3
3	Nonuniform gravity, load proportional to surface area, in negative global z-direction. Use the FORCEM user subroutine.
4	Nonuniform pressure, proportional to surface area, positive when in direction of normal. Use the FORCEM user subroutine.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.

Load Type	Description
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Output of Strains and Stresses

Output of stress (σ_{11}, σ_{22}, τ_{12}) and strain (ε_{11}, ε_{22}, γ_{12}) is in the local (V_1, V_2) directions defined above.

Transformation

Nodal degrees of freedom can be transformed to local degrees of freedom.

Output Points

Centroid.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 197. See Element 197 for a description of the conventions used for entering the flux and film data for this element.

Element 201

Two-dimensional Plane Stress Triangle

Element 201 is a three-node, isoparametric, triangular element written for plane stress applications. This element uses bilinear interpolation functions. The stresses are constant throughout the element. This results in a poor representation of shear behavior.

In general, you need more of these lower-order elements than the higher-order elements such as element type 124. Hence, use a fine mesh.

The stiffness of this element is formed using one point integration at the centroid. The mass matrix of this element is formed using four-point Gaussian integration.

Quick Reference

Type 201

Two-dimensional, plane stress, three-node triangle.

Connectivity

Node numbering must be counterclockwise (see Figure 3-293).

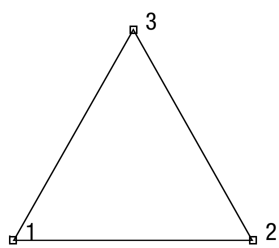


Figure 3-293 Plane-Stress Triangle

Geometry

Thickness stored in first data field (EGOM1). Default thickness is unity. Other fields are not used.

Coordinates

Two coordinates in the global x and y directions.

Degrees of Freedom

1 = u displacement
2 = v displacement

Tractions

Load types for distributed loads are as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force per unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element; magnitude supplied through the FORCEM user subroutine.
4	Nonuniform body force per unit volume in first coordinate direction; magnitude supplied through the FORCEM user subroutine.
5	Nonuniform body force per unit volume in second coordinate direction; magnitude supplied through the FORCEM user subroutine.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element; magnitude supplied through the FORCEM user subroutine.
8	Uniform pressure on 3-1 face of the element.
9	Nonuniform pressure on 3-1 face of the element; magnitude supplied through the FORCEM user subroutine.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at the centroid of the element is as follows:

$1 = \varepsilon_{xx}$

$2 = \varepsilon_{yy}$

$4 = \gamma_{xy}$

Output of Stresses

Same as Output of Strains .

Transformation

Nodal degrees of freedom can be transformed to local degrees of freedom. Corresponding nodal loads must be applied in local direction.

Tying

Use the UFORMSN user subroutine.

Output Points

Only available at the centroid. Element must be small for accuracy.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available. Output of stress in global coordinate directions.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 37. See Element 37 for a description of the conventions used for entering the flux and film data for this element.

Element 202

Three-dimensional Fifteen-node Pentahedral

Element type 202 is a fifteen-node, isoparametric, arbitrary pentahedral. This element uses triquadratic interpolation functions, which gives good behavior.

The stiffness of this element is formed using 21-point Gaussian integration.

This element can be used for all constitutive relations. When using incompressible rubber materials (for example, Mooney and Ogden), the element must be used within the Updated Lagrange framework.

Quick Reference

Type 202

Three-dimensional, twenty-one-node, second-order, isoparametric element (arbitrarily distorted pentahedral).

Connectivity

Fifteen nodes per element. Node numbering must follow the scheme below (see Figure 3-294):

> Nodes 1, 2, and 3 are corners of one face, given in counterclockwise order when viewed from inside the element. Node 4 has the same edge as node 1. Node 5 has the same edge as node 2. Node 6 has the same edge as node 3. The midside nodes follow as shown:

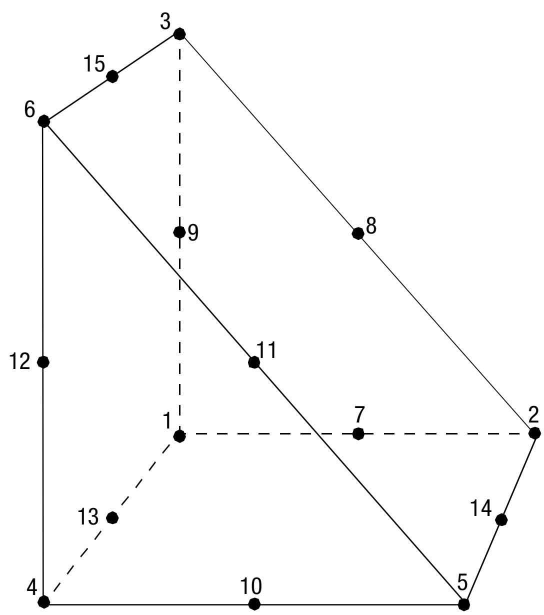


Figure 3-294 Pentahedral

Geometry

Not required.

Coordinates

Three coordinates in the global x-, y-, and z-directions.

Degrees of Freedom

Three global degrees of freedom u, v, and w per node.

Distributed Loads

Distributed loads chosen by value of IBODY are as follows:

Load Type	Description
1	Uniform pressure on 1-2-5-4 face.
2	Nonuniform pressure on 1-2-5-4 face; magnitude and direction supplied through the FORCEM user subroutine.
3	Uniform pressure on 2-3-6-5 face.
4	Nonuniform pressure on 2-3-6-5 face; magnitude and direction supplied through the FORCEM user subroutine.
5	Uniform pressure on 3-1-4-6 face.
6	Nonuniform pressure on 3-1-4-6 face; magnitude and direction supplied through the FORCEM user subroutine.
7	Uniform pressure on 1-3-2 face.
8	Nonuniform pressure on 1-3-2 face; magnitude and direction supplied through the FORCEM user subroutine.
9	Uniform pressure on 4-5-6 face.
10	Nonuniform pressure on 4-5-6 face; magnitude and direction supplied through the FORCEM user subroutine.
11	Uniform volumetric load in x-direction.
12	Nonuniform volumetric load in x-direction; magnitude and direction supplied through the FORCEM user subroutine.
13	Uniform volumetric load in y-direction.
14	Nonuniform volumetric load in y-direction; magnitude and direction supplied through the FORCEM user subroutine.
15	Uniform volumetric load in z-direction.
16	Nonuniform volumetric load in z-direction; magnitude and direction supplied through the FORCEM user subroutine.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Load Type	Description
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Pressure forces are positive into element face.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

1 = ε_{xx}
2 = ε_{yy}
3 = ε_{zz}
4 = γ_{xy}
5 = γ_{yz}
6 = γ_{zx}

Output of Stresses

Output of stresses is the same as for Output of Strains .

Transformation

Standard transformation of three global degrees of freedom to local degrees of freedom.

Tying

No special tying available.

Output Points

Centroid or the 21 integration points as shown in Figure 3-295.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 203. See Element 203 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux generated by dissipated plastic energy is specified with type 101.

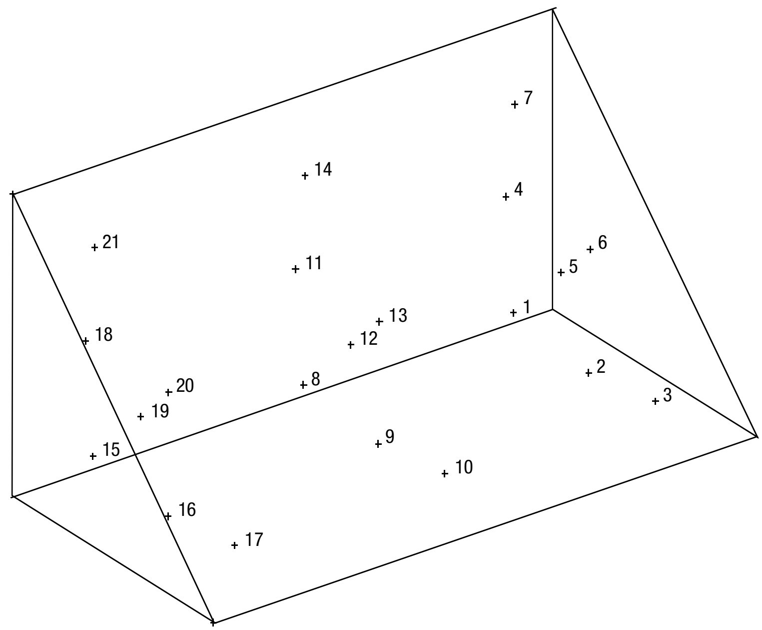


Figure 3-295 Twenty-one-Point Gauss Integration Scheme for Element 202

Element 203 Three-dimensional Fifteen-node Pentahedral (Heat Transfer Element)

Element type 203 is a fifteen-node, isoparametric, arbitrary pentahedral written for three-dimensional heat transfer applications. This element can also be used for electrostatic applications.

As this element uses triquadratic interpolation functions, the thermal gradients have a linear variation throughout the element. This allows for accurate representation of the temperature field.

The conductivity of this element is formed using twenty-one-point Gaussian integration.

Quick Reference

Type 203

Fifteen-node, three-dimensional, second-order isoparametric heat transfer element.

Connectivity

Fifteen nodes per element.

Nodes 1, 2, and 3 are the corners on one face, numbered in a counterclockwise direction when viewed from inside the element. Nodes 4, 5, and 6 are the nodes on the other face, with node 4 opposite node 1, and so on. The midside nodes follow as shown (see Figure 3-296).

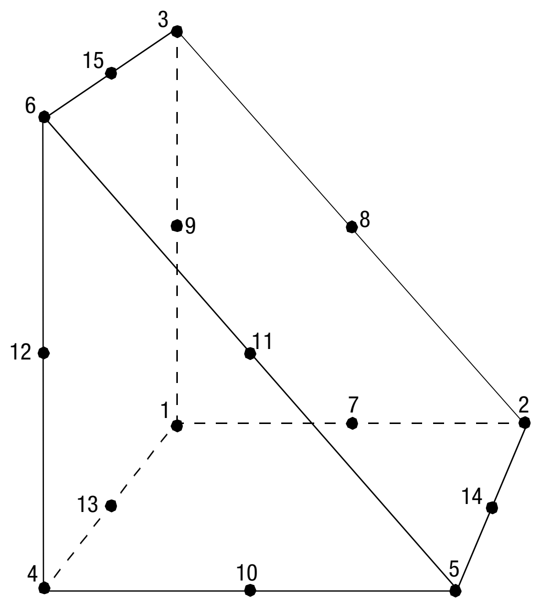


Figure 3-296 Arbitrarily Distorted Heat Transfer Pentahedral

Geometry

Not applicable.

Coordinates

Three coordinates in the global x, y, and z directions.

Degrees of Freedom

1 = temperature (heat transfer)
1 = voltage, temperature (Joule Heating)
1 = potential (electrostatic)

Fluxes

Fluxes are distributed according to the appropriate selection of a value of IBODY. Surface fluxes are assumed positive when directed into the element and are evaluated using a four-point integration scheme, where the integration points have the same location as the nodal points.

Load Type (IBODY)	Description
1	Uniform flux on 1-2-5-4 face.
2	Nonuniform flux on 1-2-5-4 face; magnitude and direction supplied through the FLUX user subroutine.
3	Uniform flux on 2-3-6-5 face.
4	Nonuniform flux on 2-3-6-5 face; magnitude and direction supplied through the FLUX user subroutine.
5	Uniform flux on 3-1-4-6 face.
6	Nonuniform flux on 3-1-4-5 face; magnitude and direction supplied through the FLUX user subroutine.
7	Uniform flux on 1-3-2 face.
8	Nonuniform flux on 1-3-2 face; magnitude and direction supplied through the FLUX user subroutine.
9	Uniform flux on 4-5-6 face.
10	Nonuniform flux on 4-5-6 face; magnitude and direction supplied through the FLUX user subroutine.
11	Uniform volumetric flux.
12	Nonuniform volumetric flux; magnitude and direction supplied through the FLUX user subroutine.

Films

Same specification as Fluxes .

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Current

Same specification as Fluxes .

Charge

Same specifications as Fluxes .

Output Points

Centroid or 21 Gaussian integration points (see Figure 3-297).

Note:	As in all three-dimensional analysis, a large nodal bandwidth results in long computing times. Use the optimizers as much as possible.

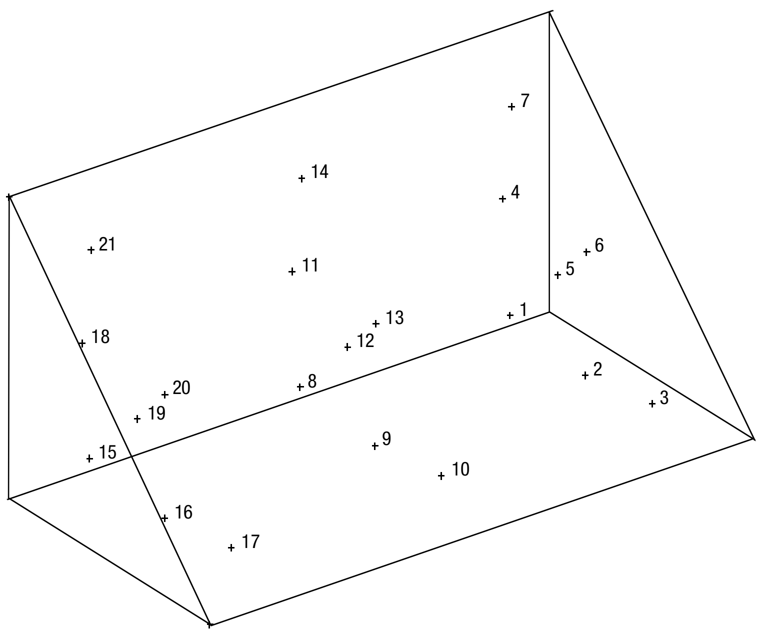


Figure 3-297 Integration Points for Element 203

Element 204

Three-dimensional Magnetostatic Pentahedral

Element type 204 is a six-node, isoparametric, magnetostatic pentahedral. As this element uses trilinear interpolation functions, the magnetic induction tends to be constant throughout the element; hence, high gradients are not captured.

The coefficient matrix of this element is formed using six-point Gaussian integration.

Quick Reference

Type 204

Three-dimensional, six-node, first-order, isoparametric, magnetostatic element.

Connectivity

Six nodes per element. Node numbering must follow the scheme below (see Figure 3-298):

Nodes 1, 2, and 3 are corners of one face, given in counterclockwise order when viewed from inside the element. Node 4 has the same edge as node 1. Node 5 has the same edge as node 2. Node 6 has the same edge as node 3.

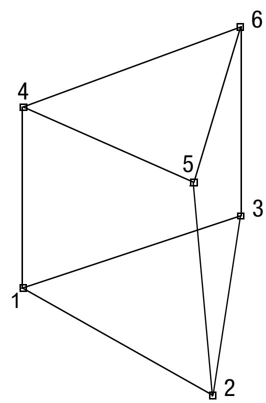


Figure 3-298 Pentahedral

Geometry

The value of the penalty factor is given in the second field `EGEOM7` (default is 0.0001).

Coordinates

Three coordinates in the global x-, y-, and z-directions.

Degrees of Freedom

1 = x component of vector potential
2 = y component of vector potential
3 = z component of vector potential

Distributed Currents

Distributed currents chosen by value of IBODY are as follows:

Load Type	Description
1	Uniform current on 1-2-5-4 face.
2	Nonuniform current on 1-2-5-4 face; magnitude and direction supplied through the FORCEM user subroutine.
3	Uniform current on 2-3-6-5 face.
4	Nonuniform current on 2-3-6-5 face; magnitude and direction supplied through the FORCEM user subroutine.
5	Uniform current on 3-1-4-6 face.
6	Nonuniform current on 3-1-4-6 face; magnitude and direction supplied through the FORCEM user subroutine.
7	Uniform current on 1-3-2 face.
8	Nonuniform current on 1-3-2 face; magnitude and direction supplied through the FORCEM user subroutine.
9	Uniform current on 4-5-6 face.
10	Nonuniform current on 4-5-6 face; magnitude and direction supplied through the FORCEM user subroutine.
11	Uniform volumetric current in x-direction.
12	Nonuniform volumetric current in x-direction; magnitude and direction supplied through the FORCEM user subroutine.
13	Uniform volumetric current in y-direction.
14	Nonuniform volumetric current in y-direction; magnitude and direction supplied through the FORCEM user subroutine.
15	Uniform volumetric current in z-direction.
16	Nonuniform volumetric current in z-direction; magnitude and direction supplied through the FORCEM user subroutine.

Currents are positive into element face.

Joule Heating

Capability is not available.

Electrostatic

Capability is not available.

Magnetostatic

Capability is available.

Output at Integration Points

Potential

Magnetic flux density

1 = x
2 = y
3 = z

Magnetic field vector

1 = x
2 = y
3 = z

Tying

Use the UFORMSN user subroutine.

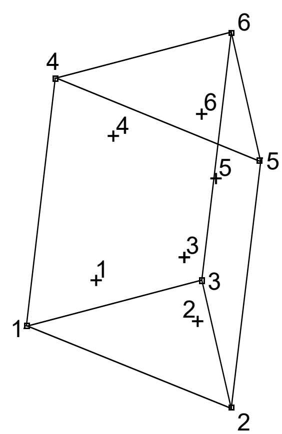


Figure 3-299 Six-Point Gauss Integration Scheme for Element 204

Element 205

Three-dimensional Fifteen-node Magnetostatic Pentahedral

Element type 205 is a fifteen-node, isoparametric, arbitrary pentahedral written for three-dimensional magnetostatic applications.

This element uses triquadratic interpolation functions. This allows for an accurate representation of the magnetic induction.

The conductivity of this element is formed using twenty-one point Gaussian integration.

Quick Reference

Type 205

Fifteen-node, three-dimensional, second-order isoparametric magnetostatic element.

Connectivity

Fifteen nodes per element.

Nodes 1, 2, and 3 are the corners on one face, numbered in a counterclockwise direction when viewed from inside the element. Nodes 4, 5, and 6 are the nodes on the other face, with node 4 opposite node 1, and so on. The midside nodes follow as shown (see Figure 3-300).

Geometry

The value of the penalty factor is given in the second field `EGEOM8` (default is 1.0). The values of the magnetic potential and the magnetic induction are highly dependent upon the value of the penalty factor. For this reason, lower-order elements are recommended.

Coordinates

Three coordinates in the global x, y, and z directions.

Degrees of Freedom

1 = x component of vector potential
2 = y component of vector potential
3 = z component of vector potential

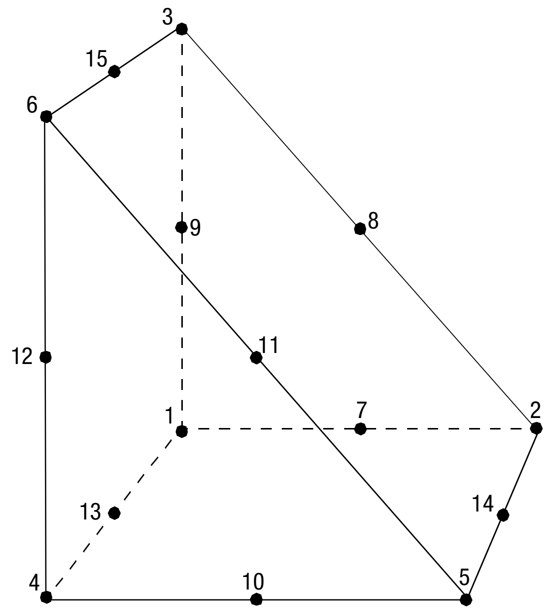


Figure 3-300 Arbitrarily Magnetostatic Pentahedral

Currents

Currents are distributed according to the appropriate selection of a value of IBODY. surface currents are assumed positive when directed into the element.

Load Type (IBODY)	Description
1	Uniform current on 1-2-5-4 face.
2	Nonuniform current on 1-2-5-4 face; magnitude and direction supplied through the FORCEM user subroutine.
3	Uniform current on 2-3-6-5 face.
4	Nonuniform current on 2-3-6-5 face; magnitude and direction supplied through the FORCEM user subroutine.
5	Uniform current on 3-1-4-6 face.
6	Nonuniform current on 3-1-4-5 face; magnitude and direction supplied through the FORCEM user subroutine.
7	Uniform current on 1-3-2 face.
8	Nonuniform current on 1-3-2 face; magnitude and direction supplied through the FORCEM user subroutine.
9	Uniform current on 4-5-6 face.

Load Type (IBODY)	Description
10	Nonuniform current on 4-5-6 face; magnitude and direction supplied through the FORCEM user subroutine.
11	Uniform volumetric current.
12	Nonuniform volumetric current; magnitude and direction supplied through the FORCEM user subroutine.

Joule Heating

Capability is not available.

Electrostatic

Capability is not available.

Magnetostatic

Capability is available.

Output Points

Centroid or twenty-one Gaussian integration points (see Figure 3-301).

Output at Integration Points

Potential

Magnetic flux density

1 = x
2 = y
3 = z

Magnetic field vector

1 = x
2 = y
3 = z

Tying

Use the UFORMSN user subroutine.

Note:	As in all three-dimensional analysis, a large nodal bandwidth results in long computing times. Use the optimizers as much as possible.

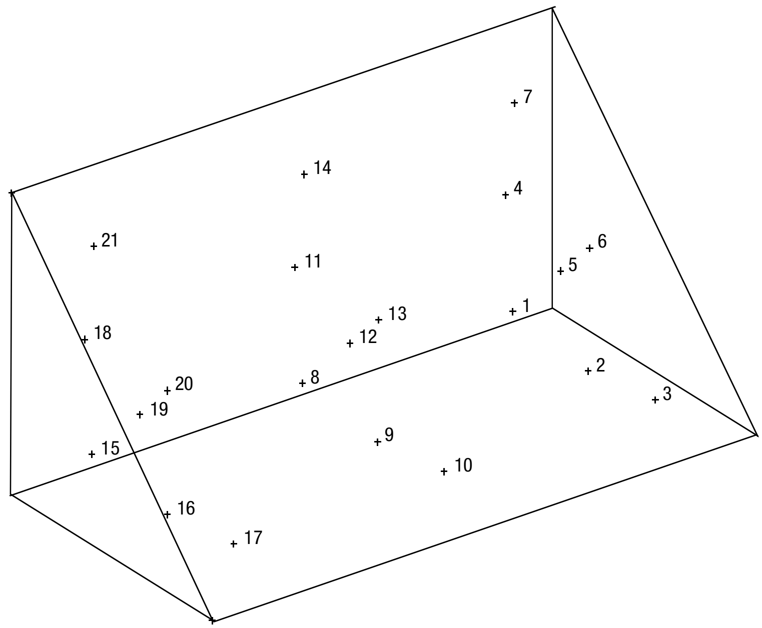


Figure 3-301 Integration Points for Element 205

Element 206 — Twenty-node 3-D Magnetostatic Element

This is an twenty-node 3-D magnetostatic element with quadratic interpolation functions. It is similar to element type 44. The coefficient matrix is numerically integrated using twenty-seven integration points.

Quick Reference

Type 206

Twenty-node 3-D magnetostatic element.

Connectivity

Twenty nodes per element.

See Figure 3-302 for numbering. Nodes 1-4 are the corners on one face, numbered in a counterclockwise direction when viewed from inside the element. Nodes 5-8 are the nodes on the other face, with node 5 opposite node 1, and so on.

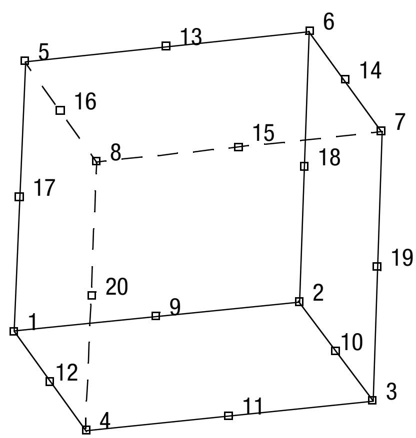


Figure 3-302 Twenty-node 3-D Magnetostatic Element

Coordinates

Three coordinates in the global x-, y-, and z-directions.

Geometry

The value of the penalty factor is given in the second field EGEOM8 (default is 1.0). The values of the magnetic potential and the magnetic induction are highly dependent upon the value of the penalty factor. For this reason, lower-order elements are recommended.

Degrees of Freedom

1 = x component of vector potential
2 = y component of vector potential
3 = z component of vector potential

Distributed Currents

Distributed currents are listed in the table below:

Current Type	Description
0	Uniform current on 1-2-3-4 face.
1	Nonuniform current on 1-2-3-4 face; magnitude is supplied through the FORCEM user subroutine.
2	Uniform body force per unit volume in -z-direction.
3	Nonuniform body force per unit volume (e.g. centrifugal force); magnitude and direction is supplied through the FORCEM user subroutine.
4	Uniform current on 6-5-8-7 face.
5	Nonuniform current on 6-5-8-7 face.
6	Uniform current on 2-1-5-6 face.
7	Nonuniform current on 2-1-5-6 face.
8	Uniform current on 3-2-6-7 face.
9	Nonuniform current on 3-2-6-7 face.
10	Uniform pressure on 4-3-7-8 face.
11	Nonuniform current on 4-3-7-8 face.
12	Uniform current on 1-4-8-5 face.
13	Nonuniform current on 1-4-8-5 face.
20	Uniform current on 1-2-3-4 face.
21	Nonuniform current on 1-2-3-4 face.
22	Uniform body current per unit volume in -z-direction.
23	Nonuniform body current per unit volume (for example, centrifugal force); magnitude and direction is supplied through the FORCEM user subroutine.

For all nonuniform currents, body forces per unit volume and loads, the magnitude and direction is supplied via the FORCEM user subroutine.

Currents are positive into element face.

Joule Heating

Capability is not available.

Electrostatic

Capability is not available.

Magnetostatic

Capability is available.

Output Points

Centroid or the twenty-seven integration points shown in Figure 3-303

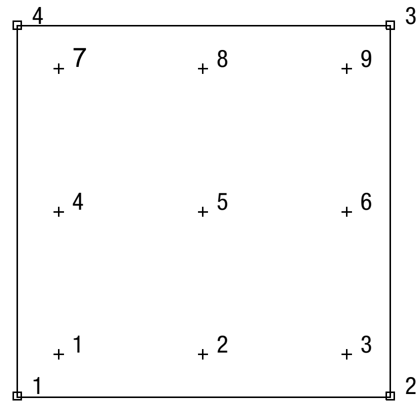


Figure 3-303 Element 206 Integration Plane

Output at Integration Points

Potential

Magnetic flux density

1 = x
2 = y
3 = z

Magnetic field vector

1 = x
2 = y
3 = z

Tying

Use the UFORMSN user subroutine.

Element 207

Not Available

Not available at this time.

Element 208

Not Available

Not available at this time.

Element 209

Not Available

Not available at this time.

Element 210

Not Available

Not available at this time.

Element 211

Not Available

Not available at this time.

Element 212

Not Available

Not available at this time.

Element 213

Not Available

Not available at this time.

Element 214

Not Available

Not available at this time.

Element 215

Not Available

Not available at this time.

Element 216 Three-dimensional Five-node Pyramid

Element type 216 is a five-node, isoparametric, arbitrary pyramid. As this element uses trilinear interpolation functions, the strains tend to be constant throughout the element. This results in a poor representation of shear behavior.

The stiffness of this element is formed using five-point Gaussian integration.

For nearly incompressible behavior, including plasticity or creep, it is advantageous to use an alternative integration procedure. This constant dilatation method which eliminates potential element locking is flagged through the GEOMETRY option.

This element can be used for all constitutive relations. When using incompressible rubber materials (for example, Mooney and Ogden), the element must be used within the Updated Lagrange framework.

Quick Reference

Type 216

Three-dimensional, five-node, first-order, isoparametric element (arbitrarily distorted pyramid).

Connectivity

Five nodes per element. Node numbering must follow the scheme below (see Figure 3-304).

Nodes 1, 2,3, and 4 form the base of the pyramid, given in counter clockwise order when viewed from inside the element. Node 5 is on the apex of the pyramid.

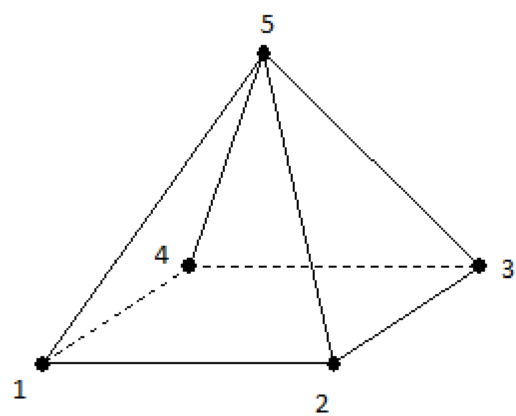


Figure 3-304 Arbitrary distorted pyramid element

Geometry

If a nonzero value is entered in the second data field (`EGEOM2`), the volumetric strain is constant throughout the element. This is particularly useful for analysis of approximately incompressible materials, and for analysis of structures in the fully plastic range. It is also recommended for creep problems in which it is attempted to obtain the steady state solution.

Coordinates

Three coordinates in the global x-, y- and z- directions.

Degrees of Freedom

Three global degrees of freedom u, v, and w per node.

Distributed Loads

Distributed loads chosen by value of IBODY are as follows:

Load Type	Description
1	Uniform pressure on 1-2-5 face.
2	Nonuniform pressure on 1-2-5 face; magnitude and direction supplied through the FORCEM user subroutine.
3	Uniform pressure on 2-3-5 face.
4	Nonuniform pressure on 2-3-5 face; magnitude and direction supplied through the FORCEM user subroutine.
5	Uniform pressure on 3-4-5 face.
6	Nonuniform pressure on 3-4-5 face; magnitude and direction supplied through the FORCEM user subroutine.
7	Uniform pressure on 4-1-5 face.
8	Nonuniform pressure on 4-1-5 face; magnitude and direction supplied through the FORCEM user subroutine.
9	Uniform pressure on 1-2-3-4 face.
10	Nonuniform pressure on 1-2-3-4 face; magnitude and direction supplied through the FORCEM user subroutine.
11	Uniform volumetric load in x-direction.
12	Nonuniform volumetric load in x-direction; magnitude and direction supplied through the FORCEM user subroutine.
13	Uniform volumetric load in y-direction.
14	Nonuniform volumetric load in y-direction; magnitude and direction supplied through the FORCEM user subroutine.
15	Uniform volumetric load in z-direction.
16	Nonuniform volumetric load in z-direction; magnitude and direction supplied through the FORCEM user subroutine.

Load Type	Description
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in global x, y, z direction respectively.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Pressure forces are positive into element face.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

1 = ε_{xx}
2 = ε_{yy}
3 = ε_{zz}
4 = γ_{xy}
5 = γ_{yz}
6 = γ_{zx}

Output of Stresses

Output of stresses is same as for Output of Strains .

Transformation

Standard transformation of three global degrees of freedom to local degrees of freedom.

Output Points

Centroid or the 5 integration points as shown in Figure 3-305

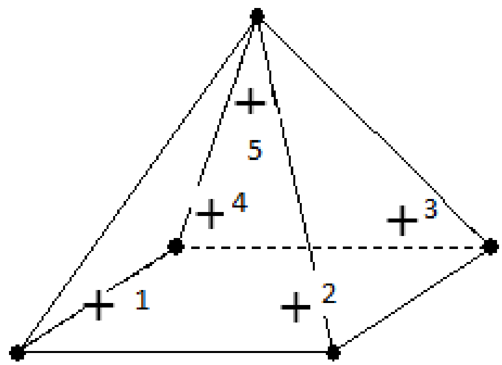


Figure 3-305 Five-Point Gauss Integration Scheme for Element 216

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 217. See Element 217 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux generated by dissipated plastic energy is specified with type 101.

Element 217 Three-dimensional Five-node Pyramid (Heat Transfer Element)

Element type 217 is a five-node, isoparametric, arbitrary pyramid written for three-dimensional heat transfer applications. This element can be used for electrostatic applications.

As this element uses trilinear interpolation functions, the thermal gradients tend to be constant throughout the element. This results in a poor representation of shear behavior.

The conductivity of this element is formed using five-point Gaussian integration

Quick Reference

Type 217

Three-dimensional, five-node, first-order, isoparametric heat transfer element.

Connectivity

Five nodes per element. Node numbering must follow the scheme below (see Figure 3-306).

Nodes 1, 2,3, and 4 form the base of the pyramid, given in counter clockwise order when viewed from inside the element. Node 5 is on the apex of the pyramid.

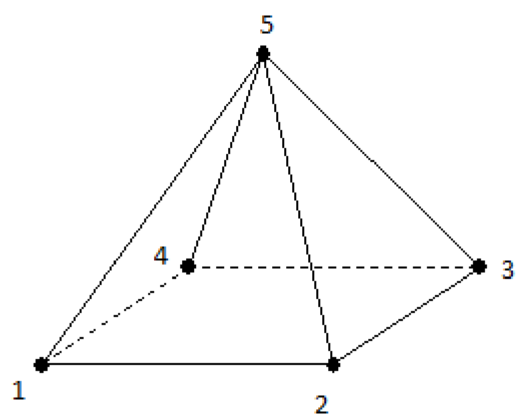


Figure 3-306 Arbitrary distorted pyramid element

Geometry

If a nonzero value is entered in the fourth data field (EGEOM4), the temperatures at the integration points obtained from interpolation of nodal temperatures are constant throughout the element.

Coordinates

Three coordinates in the global x-, y- and z- directions.

Degrees of Freedom

1 = temperature (heat transfer)

1 = voltage, temperature (Joule Heating)

1 = potential (electrostatic)

Fluxes

Fluxes are distributed according to the appropriate selection of a value of `IBODY`. Surface fluxes are assumed positive when directed into the element and are evaluated using a 4-point integration scheme, where the integration points have the same location as the nodal points.:

Load Type	Description
1	Uniform flux on 1-2-5 face.
2	Nonuniform flux on 1-2-5 face; magnitude and direction supplied through the FLUX user subroutine.
3	Uniform flux on 2-3-5 face.
4	Nonuniform flux on 2-3-5 face; magnitude and direction supplied through the FLUX user subroutine.
5	Uniform flux on 3-4-5 face.
6	Nonuniform flux on 3-4-5 face; magnitude and direction supplied through the FLUX user subroutine.
7	Uniform flux on 4-1-5 face.
8	Nonuniform flux on 4-1-5 face; magnitude and direction supplied through the FLUX user subroutine.
9	Uniform flux on 1-2-3-4 face.
10	Nonuniform flux on 1-2-3-4 face; magnitude and direction supplied through the FLUX user subroutine.
11	Uniform volumetric flux; magnitude and direction supplied through the FLUX user subroutine.
12	Nonuniform volumetric flux; magnitude and direction supplied through the FLUX user subroutine.

Films

Same specification as Fluxes .

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Current

Same specifications as Fluxes .

Charge

Same specifications as Fluxes .

Output Points

Centroid or five Gaussian integration points (see Figure 3-307)

Note:	As in all three-dimensional analyses, a large nodal bandwidth results in long computing times. Use the optimizer as much as possible.

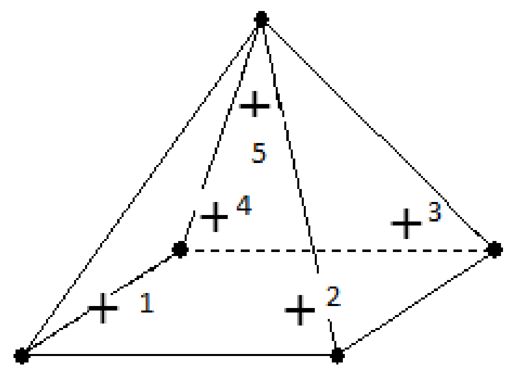


Figure 3-307 Five-Point Gauss Integration Scheme for Element 217

Element 218

Three-dimensional Thirteen-node Pyramid

Element type 218 is a thirteen-node, isoparametric, arbitrary pyramid. This element uses triquadratic interpolation functions to represent the coordinates and displacements; hence, the strains have a linear variation. This allows for an accurate representation of the strain fields in elastic analyses.

The stiffness of this element is formed using eight-point Gaussian integration. The mass matrix of this element is formed using twenty-seven point Gaussian integration

This element can be used for all constitutive relations. When using incompressible rubber materials (for example, Mooney and Ogden), the element must be used within the Updated Lagrange framework.

Quick Reference

Type 218

Three-dimensional, thirteen-node, second-order, isoparametric element (arbitrarily distorted pyramid).

Connectivity

Thirteen nodes per element. Node numbering must follow the scheme below (see Figure 3-308):

Nodes 1-2-3-4-6-7-8-9 form the base of the pyramid, given in counter clockwise order when viewed from inside the element. Node 5 is on the apex of the pyramid. The midside nodes follow as shown

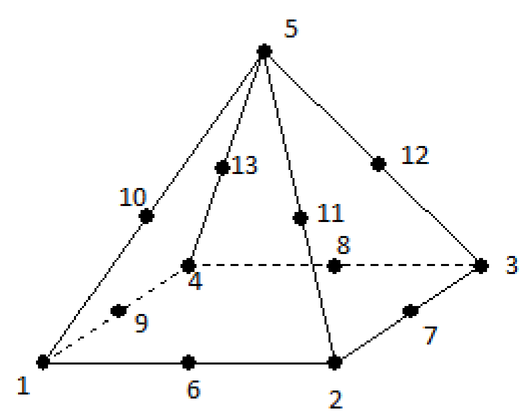


Figure 3-308 Arbitrary distorted pyramid element

Geometry

Not applicable.

Coordinates

Three coordinates in the global x-, y- and z- directions.

Degrees of Freedom

Three global degrees of freedom u, v, and w per node.

Distributed Loads

Distributed loads chosen by value of `IBODY` are as follows:

Load Type	Description
1	Uniform pressure on 1-2-5 face.
2	Nonuniform pressure on 1-2-5 face; magnitude and direction supplied through the FORCEM user subroutine.
3	Uniform pressure on 2-3-5 face.
4	Nonuniform pressure on 2-3-5 face; magnitude and direction supplied through the FORCEM user subroutine.
5	Uniform pressure on 3-4-5 face.
6	Nonuniform pressure on 3-4-5 face; magnitude and direction supplied through the FORCEM user subroutine.
7	Uniform pressure on 4-1-5 face.
8	Nonuniform pressure on 4-1-5 face; magnitude and direction supplied through the FORCEM user subroutine.
9	Uniform pressure on 1-2-3-4 face.
10	Nonuniform pressure on 1-2-3-4 face; magnitude and direction supplied through the FORCEM user subroutine.
11	Uniform volumetric load in x-direction.
12	Nonuniform volumetric load in x-direction; magnitude and direction supplied through the FORCEM user subroutine.
13	Uniform volumetric load in y-direction.
14	Nonuniform volumetric load in y-direction; magnitude and direction supplied through the FORCEM user subroutine.
15	Uniform volumetric load in z-direction.
16	Nonuniform volumetric load in z-direction; magnitude and direction supplied through the FORCEM user subroutine.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in global x, y, z directions, respectively.

Load Type	Description
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

Pressure forces are positive into element face.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

1 = ε_{xx}
2 = ε_{yy}
3 = ε_{zz}
4 = γ_{xy}
5 = γ_{yz}
6 = γ_{zx}

Output of Stresses

Output of stresses is same as for Output of Strains .

Transformation

Standard transformation of three global degrees of freedom to local degrees of freedom.

Tying

No special tying available.

Output Points

The 8 integration points as shown in Figure 3-309

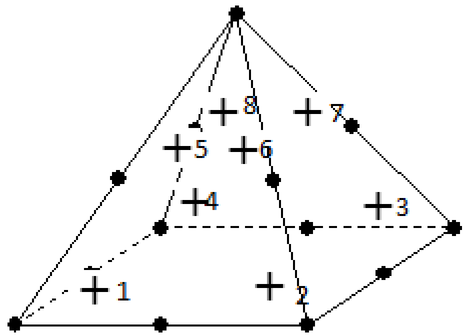


Figure 3-309 Arbitrary distorted pyramid element

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 219. See Element 219 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux generated by dissipated plastic energy is specified with type 101.

Element 219

Three-dimensional Thirteen-node Pyramid (Heat Transfer Element)

Element type 219 is a thirteen-node, isoparametric, arbitrary pyramid written for three-dimensional heat transfer applications. This element can be used for electrostatic applications.

As this element uses triquadratic interpolation functions, the thermal gradients have a linear variation throughout the element. This allows for accurate representation of the temperature field.

The conductivity of this element is formed using eight-point Gaussian integration and the heat capacitance of this element is formed using twenty-seven point Gaussian integration.

Quick Reference

Type 219

Three-dimensional, thirteen-node, second-order, isoparametric heat transfer element.

Connectivity

Thirteen nodes per element. Node numbering must follow the scheme below (see Figure 3-310).

Nodes 1, 2,3, and 4 form the base of the pyramid, given in counter clockwise order when viewed from inside the element. Node 5 is on the apex of the pyramid.

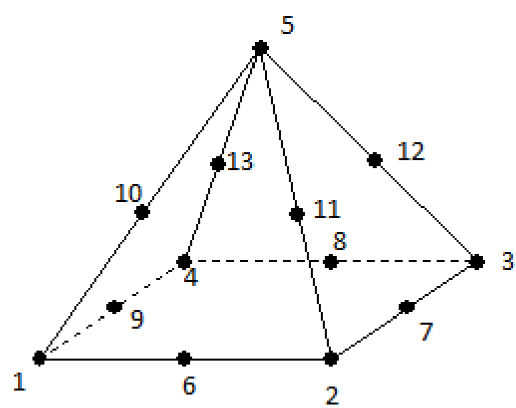


Figure 3-310 Arbitrary distorted pyramid element

Geometry

Not applicable.

Coordinates

Three coordinates in the global x-, y- and z- directions.

Degrees of Freedom

1 = temperature (heat transfer)

1 = voltage, temperature (Joule Heating)

1 = potential (electrostatic)

Fluxes

Fluxes are distributed according to the appropriate selection of a value of `IBODY`. Surface fluxes are assumed positive when directed into the element and are evaluated using a 4-point integration scheme, where the integration points have the same location as the nodal points.

Load Type	Description
1	Uniform flux on 1-2-5 face.
2	Nonuniform flux on 1-2-5 face; magnitude and direction supplied through the FLUX user subroutine.
3	Uniform flux on 2-3-5 face.
4	Nonuniform flux on 2-3-5 face; magnitude and direction supplied through the FLUX user subroutine.
5	Uniform flux on 3-4-5 face.
6	Nonuniform flux on 3-4-5 face; magnitude and direction supplied through the FLUX user subroutine.
7	Uniform flux on 4-1-5 face.
8	Nonuniform flux on 4-1-5 face; magnitude and direction supplied through the FLUX user subroutine.
9	Uniform flux on 1-2-3-4 face.
10	Nonuniform flux on 1-2-3-4 face; magnitude and direction supplied through the FLUX user subroutine.
11	Uniform volumetric flux; magnitude and direction supplied through the FLUX user subroutine.
12	Nonuniform volumetric flux; magnitude and direction supplied through the FLUX user subroutine.

Films

Same specification as Fluxes .

Joule Heating

Capability is available.

Electrostatic

Capability is available.

Magnetostatic

Capability is not available.

Current

Same specifications as Fluxes .

Charge

Same specifications as Fluxes .

Output Points

Centroid or eight Gaussian integration points (see Figure 3-311)

Note:	As in all three-dimensional analyses, a large nodal bandwidth results in long computing times. Use the optimizer as much as possible.

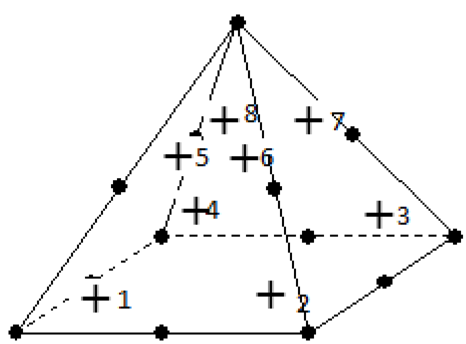


Figure 3-311 Eight-Point Gauss Integration Scheme for Element 219

Element 220

Four-node Planar Heat Transfer Interface Element

Element type 220 is a thermal four-node planar interface element, which is typically used in conjunction with element type 186 to simulate the onset and progress of delamination.

The connectivity of the element is shown in Figure 3-304. The element is typically used to model the interface between different materials, where nodes 1 and 2 correspond to one side of the interface (called the bottom) and nodes 3 and 4 to the other (called the top). The thermal gradients are expressed with respect to the local coordinate system $(\underset{\sim}{v}_1, \underset{\sim}{v}_2)$, indicated in Figure 3-304. The element is allowed to be infinitely thin, in which case the edges 1-2 and 3-4 coincide.

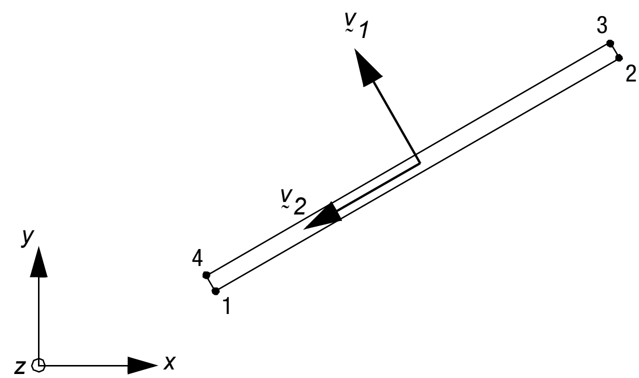


Figure 3-304 Element Type 220: Connectivity and Local Coordinate System

The constitutive behavior of the element is defined via the COHESIVE (Thermal) model definition option.

Because the element is considered to be thin, there is no specific heat or latent heat associated with this element. For similar reasons, there is no heat generated due to inelastic processes such as plasticity or curing included.

Geometric Basis

The first element base vector $\underset{\sim}{v}_1$ is obtained by rotating the direction vector from the middle of edge 1-4 to the middle of edge 2-3 counterclockwise over 90 degrees. Together with the first element base vector and the global z-axis, the second element base vector $\underset{\sim}{v}_2$ forms a right-hand system (see Figure 3-304).

The element stiffness matrix is integrated numerically using a two-point integration scheme. By default, the position of the integration points corresponds to the Gauss integration scheme. As an alternative, the Newton-Cotes/Lobatto scheme can be selected via the GEOMETRY option, in which case the integration points are located at the middle of the edges 1-4 and 2-3 (see Figure 3-305).

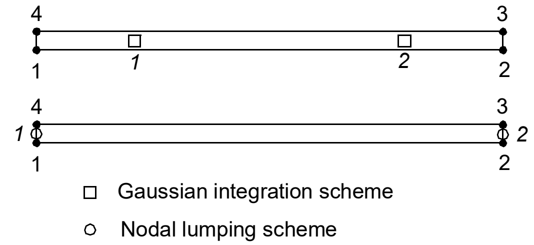


Figure 3-305 Element 220: Mid-line and Location of Integration Points

Quick Reference

Type 220

Linear, four-node heat transfer planar interface element.

Connectivity

Four nodes per element. Node numbering must be counterclockwise (see Figure 3-304), so that the two sides of the interface are the element edges 1-2 and 3-4.

Geometry

The thickness is entered in the first data field. Default thickness is one. If the fifth data field is set to one, the Newton-Cotes/Lobatto integration scheme is used instead of the default Gaussian integration scheme.

Coordinates

Two coordinates in the global x- and y-direction.

Degrees of Freedom

1 = temperature (heat transfer)

Fluxes

There are no distributed flux types available for this element.

Films

There are no distributed films for this element.

Output of Thermal Gradients

The thermal gradients are given at the element integration points. They are calculated in the local coordinate system based upon the difference in temperature between the bottom and top surfaces. However, they are output in the global coordinate system.

Tying

Use the TYING model definition option or the UFORMSN user subroutine.

Joule Heating

Capability is unavailable.

Magnetostatic

Capability is unavailable.

Electrostatic

Capability is unavailable.

Element 221

Eight-node Planar Heat Transfer Interface Element

Element type 221 is a thermal eight-node planar interface element, which is typically used in conjunction with element type 187 to simulate the onset and progress of delamination.

The connectivity of the element is shown in Figure 3-306. The element is typically used to model the interface between different materials, where nodes 1, 5, and 2 correspond to one side of the interface (called the bottom) and nodes 3, 7, and 4 to the other (called the top). The thermal gradients are expressed with respect to the local coordinate system $(\underset{\sim}{v}_1, \underset{\sim}{v}_2)$, indicated in Figure 3-306.The element is allowed to be infinitely thin, in which case the edges 1-5-2 and 3-7-4 coincide.

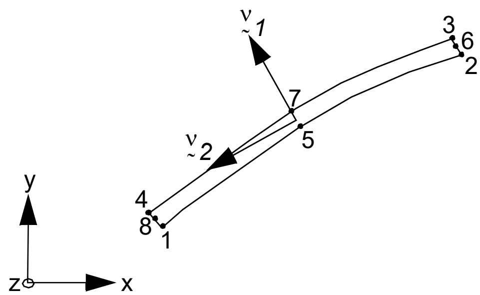


Figure 3-306 Element Type 221: Connectivity and Local Coordinate System

The constitutive behavior of the element is defined via the COHESIVE model definition option.

Because the element is considered to be thin, there is no specific heat or latent heat associated with this element. For similar reasons, there is no heat generated due to inelastic processes such as plasticity or curing included.

Geometric Basis

In each integration point, the first element base vector $\underset{\sim}{v}_1$ is obtained by rotating the tangent vector to the line through the points halfway the nodes 1 and 4, 5 and 7, and 2 and 3, counterclockwise over 90 degrees. Together with the first element base vector and the global z-axis, the second element base vector system $\underset{\sim}{v}_2$ forms a right-hand system (see Figure 3-306).

The element stiffness matrix is integrated numerically using a three-point integration scheme. By default, the position of the integration points corresponds to the Gauss integration scheme. As an alternative, the Newton-Cotes/Lobatto scheme can be selected via the GEOMETRY option, in which case the integration points are located at the middle of the edges 1-4 and 2-3 and at the element centroid (see Figure 3-307).

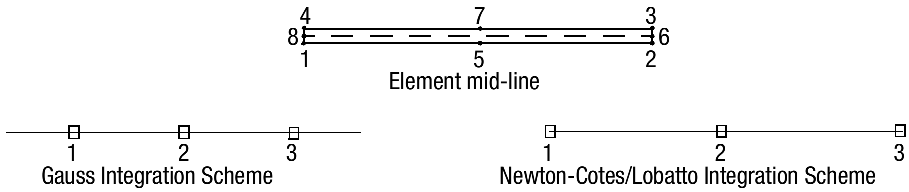


Figure 3-307 Element 221: Mid-line and Location of Integration Points

Quick Reference

Type 221

Quadratic, eight-node heat transfer planar interface element.

Connectivity

Eight nodes per element. Node numbering must be counterclockwise (see Figure 3-306), so that the two sides of the interface are the element edges 1-5-2 and 3-7-4.

Note:	Nodes 6 and 8 are actually not needed in the element formulation; they appear only to make the element compatible with eight-node quadrilateral planar elements.

Geometry

The thickness is entered in the first data field. Default thickness is one. If the fifth data field is set to one, the Newton-Cotes/Lobatto integration scheme is used instead of the default Gauss integration scheme.

Coordinates

Two coordinates in the global x- and y-direction.

Degrees of Freedom

1 = temperature (heat transfer)

Fluxes

There are no distributed flux types available for this element.

Films

There are no distributed films for this element.

Output of Thermal Gradients

The thermal gradients are given at the element integration points. They are calculated in the local coordinate system based upon the difference in temperature between the bottom and top surfaces. However, they are output in the global coordinate system.

Tying

Use the TYING model definition option or the UFORMSN user subroutine.

Joule Heating

Capability is unavailable.

Magnetostatic

Capability is unavailable.

Electrostatic

Capability is unavailable.

Element 222 — Eight-node Three-dimensional Heat Transfer Interface Element

Element type 222 is a thermal eight-node 3-D interface element, which is typically used in conjunction with element type 188 to simulate the onset and progress of delamination.

The connectivity of the element is shown in Figure 3-308. The element is typically used to model the interface between different materials, where nodes 1, 2, 3 and 4 correspond to one side (called the bottom) of the interface and nodes 5, 6, 7 and 8 to the other (called the top). The thermal gradients are expressed with respect to the local coordinate system $(\underset{\sim}{v}_1, \underset{\sim}{v}_2, \underset{\sim}{v}_3)$, indicated in Figure 3-308. The element is allowed to be infinitely thin, in which case the faces 1-2-3-4 and 5-6-7-8 coincide.

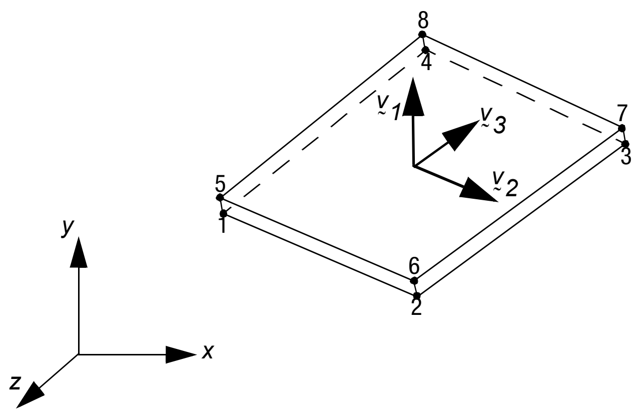


Figure 3-308 Element Type 222: Connectivity and Local Coordinate System

The constitutive behavior of the element is defined via the COHESIVE model definition option.

Because the element is considered to be thin, there is no specific heat or latent heat associated with this element. For similar reasons, there is no heat generated due to inelastic processes such as plasticity or curing included.

Geometric Basis

In order to define the local base vectors, at each integration point normalized tangent vectors to the curves with constant isoparametric coordinates are introduced (see Figure 3-309):

$$\underset{\sim}{t}_1 = \frac{\partial \underset{\sim}{x}}{\partial \xi} \Big/ \left\| \frac{\partial \underset{\sim}{x}}{\partial \xi} \right\| \text{ and } \underset{\sim}{t}_2 = \frac{\partial \underset{\sim}{x}}{\partial \eta} \Big/ \left\| \frac{\partial \underset{\sim}{x}}{\partial \eta} \right\|$$

where $\underset{\sim}{x}$ is the position vector of a point on the element mid-plane.

Now the first element base vector $\underset{\sim}{v}_1$ is the local normal vector and is given by:

$$\underset{\sim}{v}_1 = \frac{\underset{\sim}{t}_1 \times \underset{\sim}{t}_2}{\left\| \underset{\sim}{t}_1 \times \underset{\sim}{t}_2 \right\|}.$$

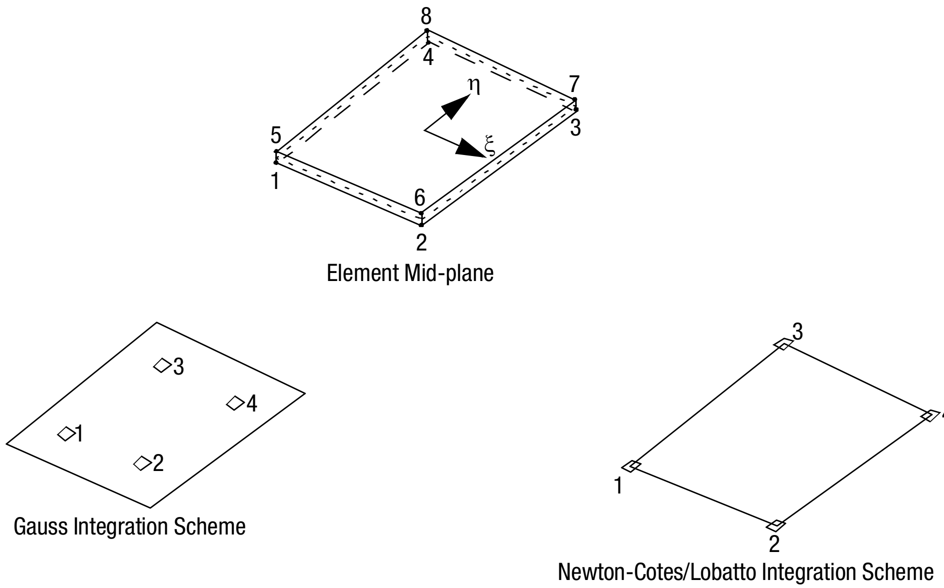


Figure 3-309 Element 222: Mid-plane and Location of Integration Points

The in-plane element base vector v_3 is constructed to be perpendicular to the plane spanned by the local normal vector v_1 and the first mid-plane edge vector e_1:

$$v_3 = \frac{v_1 \times e_1}{\| v_1 \times e_1 \|},$$

in which e_1 is defined as:

$$e_1 = \frac{1}{2}(x^2 + x^6) - \frac{1}{2}(x^1 + x^5),$$

where the superscript refers to a node number of the element.

Finally, the in-plane element base vector v_2 is parallel to the plane of the local normal vector and the first min-plane edge and follows from the cross product of the former two element base vectors:

$$v_2 = v_3 \times v_1 .$$

The element stiffness matrix is integrated numerically using a four-point integration scheme. By default, the position of the integration points corresponds to the Gauss integration scheme. As an alternative, the Newton/Cotes/Lobatto integration scheme can be selected via the GEOMETRY option, in which case the integration points are located at the middle of the edges 1-5, 2-6, 3-7, and 4-8 (see Figure 3-309).

Quick Reference

Type 222

Linear, eight-node heat transfer 3-D interface element.

Connectivity

Eight nodes per element. Node numbering must be according to Figure 3-308, so that the two sides of the interface are the element faces 1-2-3-4 and 5-6-7-8.

Geometry

If the fifth data field is set to one, the Newton-Cotes/Lobatto integration scheme is used instead of the default Gauss integration scheme.

Coordinates

Three coordinates in the global x-, y- and z-direction.

Degrees of Freedom

1 = temperature (heat transfer)

Fluxes

There are no distributed flux types available for this element.

Films

There are no distributed films for this element.

Output of Thermal Gradients

The thermal gradients are given at the element integration points. They are calculated in the local coordinate system based upon the difference in temperature between the bottom and top surfaces. However, they are output in the global coordinate system.

Tying

Use the TYING model definition option or the UFORMSN user subroutine.

Joule Heating

Capability is unavailable.

Magnetostatic

Capability is unavailable.

Electrostatic

Capability is unavailable.

Element 223

Twenty-node Three-dimensional Heat Transfer Interface Element

Element type 223 is a thermal twenty-node 3-D interface element, which is typically used in conjunction with element type 189 to simulate the onset and progress of delamination. The connectivity of the element is shown in Figure 3-310. The element is typically used to model the interface between different materials, where nodes 1, 9, 2, 10, 3, 11, 4, and 12 correspond to one side (called the bottom) of the interface and nodes 5, 13, 6, 14, 7, 15, 8, and 16 to the other (called the top). The thermal gradients are expressed with respect to the local coordinate system $(\underset{\sim}{v}_1, \underset{\sim}{v}_2, \underset{\sim}{v}_3)$, indicated in Figure 3-310.The constitutive behavior of the element is defined via the COHESIVE model definition option.

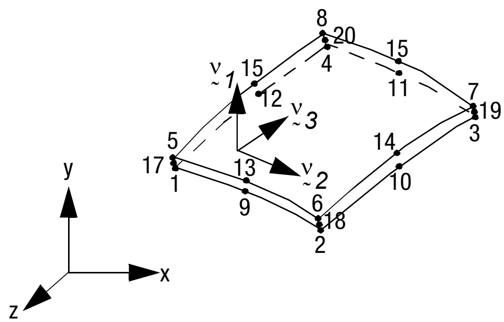


Figure 3-310 Element Type 223: Connectivity and Local Coordinate System

Because the element is considered to be thin, there is no specific heat or latent heat associated with this element. For similar reasons, there is no heat generated due to inelastic processes such as plasticity or curing included.

Geometric Basis

In order to define the local base vectors, at each integration point normalized tangent vectors to the curves with constant isoparametric coordinates are introduced (see Figure 3-311):

$$\underset{\sim}{t}_1 = \frac{\partial \underset{\sim}{x}}{\partial \xi} \bigg/ \left\| \frac{\partial \underset{\sim}{x}}{\partial \xi} \right\| \quad \text{and} \quad \underset{\sim}{t}_2 = \frac{\partial \underset{\sim}{x}}{\partial \eta} \bigg/ \left\| \frac{\partial \underset{\sim}{x}}{\partial \eta} \right\|$$

where $\underset{\sim}{x}$ is the position vector of a point on the element mid-plane.

Now the first element base vector $\underset{\sim}{v}_1$ is the local normal vector and is given by:

$$\underset{\sim}{v}_1 = \frac{\underset{\sim}{t}_1 \times \underset{\sim}{t}_2}{\left\| \underset{\sim}{t}_1 \times \underset{\sim}{t}_2 \right\|} .$$

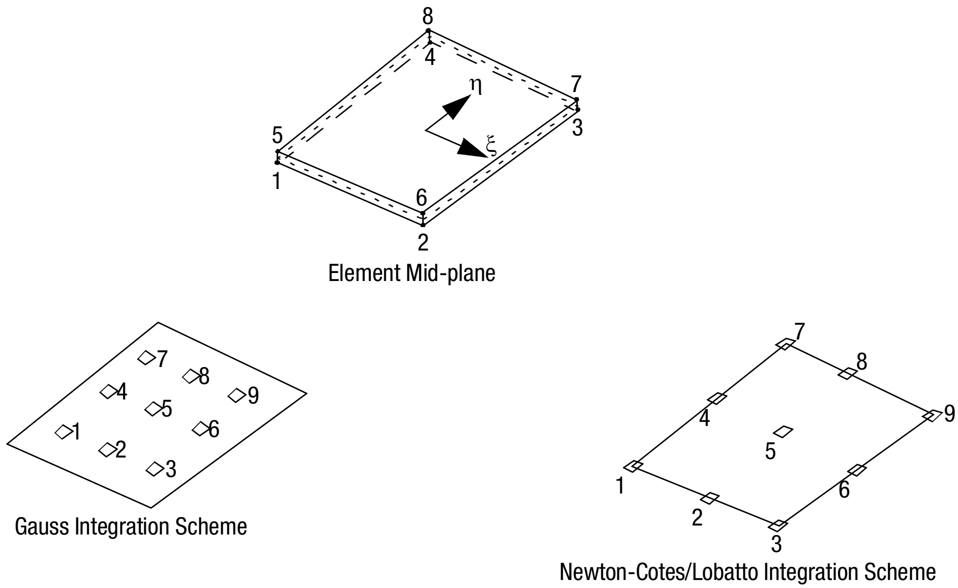


Figure 3-311 Element 223: Mid-plane and Location of Integration Points

The in-plane element base vector v_3 is constructed to be perpendicular to the plane spanned by the local normal vector v_1 and the first mid-plane edge vector e_1:

$$v_3 = \frac{v_1 \times e_1}{\|v_1 \times e_1\|},$$

in which e_1 is defined as:

$$e_1 = \frac{1}{2}(x^2 + x^6) - \frac{1}{2}(x^1 + x^5),$$

where the superscript refers to a node number of the element.

Finally, the in-plane element base vector v_2 is parallel to the plane of the local normal vector and the first min-plane edge and follows from the cross product of the former two element base vectors:

$$v_2 = v_3 \times v_1 .$$

The element stiffness matrix is integrated numerically using a nine-point integration scheme. By default, the position of the integration points corresponds to the Gauss integration scheme. As an alternative, the Newton/Cotes/Lobatto integration scheme can be selected via the GEOMETRY option, in which case the integration points are located at the middle of the edges 1-5, 2-6, 3-7, and 4-8; between nodes 9-13, 10-14, 11-15, 12-16, and at the centroid xx (see Figure 3-311).

Quick Reference

Type 223

Quadratic, twenty-node heat transfer 3-D interface element.

Connectivity

Twenty nodes per element. Node numbering must be according to Figure 3-310, so that the two sides of the interface are the element faces 1-9-2-10-3-11-4-12 and 5-13-6-14-7-15-8-16.

Note:	Nodes 17, 18, 19, and 20 are actually not needed in the element formulation; they only appear to make the element compatible with twenty-node hexahedral volume elements.

Geometry

If the fifth data field is set to one, the Newton-Cotes/Lobatto integration scheme is used instead of the default Gauss integration scheme.

Coordinates

Three coordinates in the global x-, y- and z-direction.

Degrees of Freedom

1 = temperature (heat transfer)

Fluxes

There are no distributed flux types available for this element.

Films

There are no distributed films for this element.

Output of Thermal Gradients

The thermal gradients are given at the element integration points. They are calculated in the local coordinate system based upon the difference in temperature between the bottom and top surfaces. However, they are output in the global coordinate system.

Tying

Use the TYING model definition option or the UFORMSN user subroutine.

Joule Heating

Capability is unavailable.

Magnetostatic

Capability is unavailable.

Electrostatic

Capability is unavailable.

Element 224 — Four-node Axisymmetric Heat Transfer Interface Element

Element type 224 is a thermal four-node axisymmetric interface element, which is typically used in conjunction with element type 190 to simulate the onset and progress of delamination.

The connectivity of the element is shown in Figure 3-312. The element is typically used to model the interface between different materials, where nodes 1 and 2 correspond to one side of the interface (called the bottom) and nodes 3 and 4 to the other (called the top). The thermal gradients are expressed with respect to the local coordinate system $(\underset{\sim}{v}_1, \underset{\sim}{v}_2)$, indicated in Figure 3-312. The element is allowed to be infinitely thin, in which case the edges 1-2 and 3-4 coincide.

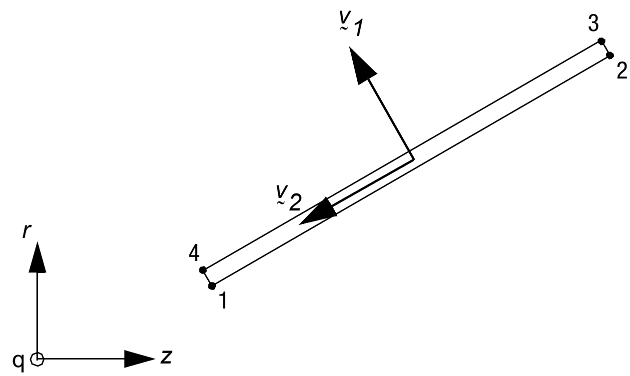


Figure 3-312 Element Type 224: Connectivity and Local Coordinate System

The constitutive behavior of the element is defined via the COHESIVE model definition option.

Because the element is considered to be thin, there is no specific heat or latent heat associated with this element. For similar reasons, there is no heat generated due to inelastic processes such as plasticity or curing included.

Geometric Basis

The first element base vector $\underset{\sim}{v}_1$ is obtained by rotating the direction vector from the middle of edge 1-4 to the middle of edge 2-3 counterclockwise over 90 degrees. Together with the first element base vector and the global θ-axis, the second element base vector $\underset{\sim}{v}_2$ forms a right-hand system (see Figure 3-312). The element stiffness matrix is integrated numerically using a two-point integration scheme. By default, the position of the integration points corresponds to the Gauss integration scheme. As an alternative, a nodal lumping scheme can be selected via the GEOMETRY option, in which case the integration points are located at the middle of the edges 1-4 and 2-3 (see Figure 3-313).

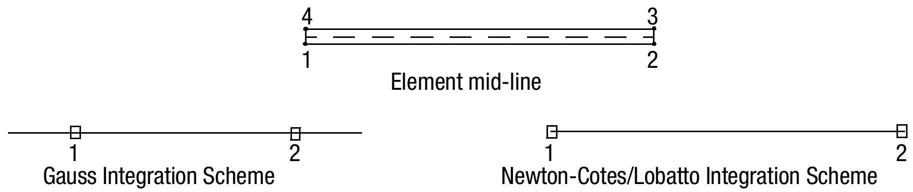


Figure 3-313 Element 224: Mid-line and Location of Integration Points

Quick Reference

Type 224

Linear, four-node heat transfer axisymmetric interface element.

Connectivity

Four nodes per element. Node numbering must be counterclockwise (see Figure 3-312), so that the two sides of the interface are the element edges 1-2 and 3-4.

Geometry

If the fifth data field is set to one, the Newton-Cotes/Lobatto integration scheme is used instead of the default Gauss integration scheme.

Coordinates

Two coordinates in the global z- and r-direction.

Degrees of Freedom

1 = temperature (heat transfer)

Fluxes

There are no distributed flux types available for this element.

Films

There are no distributed films for this element.

Output of Thermal Gradients

The thermal gradients are given at the element integration points. They are calculated in the local coordinate system based upon the difference in temperature between the bottom and top surfaces. However, they are output in the global coordinate system.

Tying

Use the TYING model definition option or the UFORMSN user subroutine.

Joule Heating

Capability is unavailable.

Magnetostatic

Capability is unavailable.

Electrostatic

Capability is unavailable.

Element 225

Eight-node Axisymmetric Heat Transfer Interface Element

Element type 225 is a thermal eight-node axisymmetric interface element, which is typically used in conjunction with element type 191 to simulate the onset and progress of delamination.

The connectivity of the element is shown in Figure 3-314. The element is typically used to model the interface between different materials, where nodes 1, 5, and 2 correspond to one side of the interface (called the bottom) and nodes 3, 7, and 4 to the other (called the top). The thermal gradients are expressed with respect to the local coordinate system $(\underset{\sim}{v}_1, \underset{\sim}{v}_2)$, indicated in Figure 3-314. The element is allowed to be infinitely thin, in which case the edges 1-5-2 and 3-7-4 coincide.

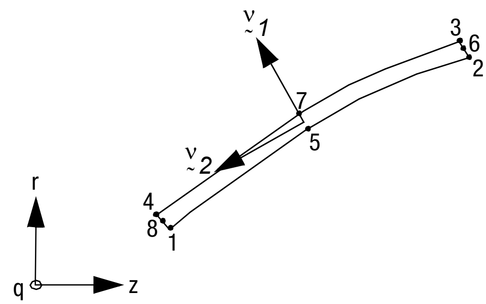


Figure 3-314 Element Type 225: Connectivity and Local Coordinate System

Because the element is considered to be thin, there is no specific heat or latent heat associated with this element. For similar reasons, there is no heat generated due to inelastic processes such as plasticity or curing included.

The constitutive behavior of the element is defined via the COHESIVE model definition option.

Geometric Basis

In each integration point, the first element base vector $\underset{\sim}{v}_1$ is obtained by rotating the tangent vector to the line through the points halfway the nodes 1 and 4, 5 and 7, and 2 and 3, counterclockwise over 90 degrees. Together with the first element base vector and the global θ-axis, the second element base vector $\underset{\sim}{v}_2$ forms a right-hand system (see Figure 3-314).

The element stiffness matrix is integrated numerically using a three-point integration scheme. By default, the position of the integration points corresponds to the Gauss integration scheme. As an alternative, the Newton- Cotes/Lobatto scheme can be selected via the GEOMETRY option, in which case the integration points are located at the middle of the edges 1-4 and 2-3 and at the element centroid (see Figure 3-315).

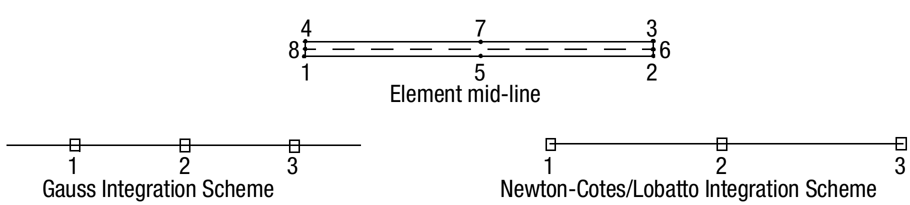


Figure 3-315 Element 225: Mid-line and Location of Integration Points

Quick Reference

Type 225

Quadratic, eight-node heat transfer axisymmetric interface element.

Connectivity

Eight nodes per element. Node numbering must be counterclockwise (see Figure 3-314), so that the two sides of the interface are the element edges 1-5-2 and 3-7-4.

Note:	Nodes 6 and 8 are actually not needed in the element formulation; they appear only to make the element compatible with eight-node quadrilateral axisymmetric elements.

Geometry

If the fifth data field is set to one, the Newton-Cotes/Lobatto integration scheme is used instead of the default Gauss integration scheme.

Coordinates

Two coordinates in the global z- and r-direction.

Degrees of Freedom

1 = temperature (heat transfer)

Fluxes

There are no distributed flux types available for this element.

Films

There are no distributed films for this element.

Output of Thermal Gradients

The thermal gradients are given at the element integration points. They are calculated in the local coordinate system based upon the difference in temperature between the bottom and top surfaces. However, they are output in the global coordinate system.

Tying

Use the TYING model definition option or the UFORMSN user subroutine.

Joule Heating

Capability is unavailable.

Magnetostatic

Capability is unavailable.

Electrostatic

Capability is unavailable.

Element 226

Six-node Three-dimensional Heat Transfer Interface Element

Element type 226 is a thermal six-node 3-D interface element, which is typically used in conjunction with element type 191 to simulate the onset and progress of delamination.

The connectivity of the element is shown in Figure 3-316. The element is typically used to model the interface between different materials, where nodes 1, 2, and 3 correspond to one side of the interface (called the bottom) and nodes 4, 5, and 6 to the other (called the top). The thermal gradients are expressed with respect to the local coordinate system $(\underset{\sim}{v}_1, \underset{\sim}{v}_2, \underset{\sim}{v}_3)$, indicated in Figure 3-316. The element is allowed to be infinitely thin, in which case the faces 1-2-3 and 4-5-6 coincide.

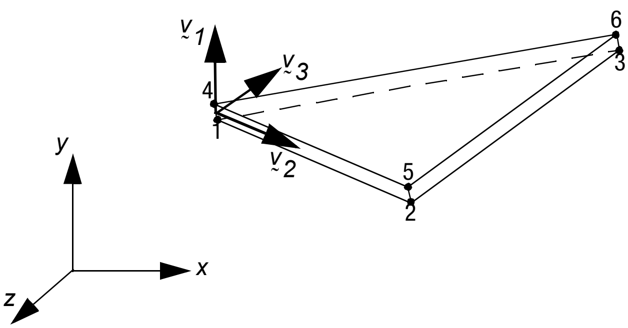


Figure 3-316 Element Type 226: Connectivity and Local Coordinate System

The constitutive behavior of the element is defined via the COHESIVE model definition option.

Because the element is considered to be thin, there is no specific heat or latent heat associated with this element. For similar reasons, there is no heat generated due to inelastic processes such as plasticity or curing included.

Geometric Basis

In order to define the local base vectors, at each integration point normalized tangent vectors to the curves with constant isoparametric coordinates are introduced (see Figure 3-317):

$$\underset{\sim}{t}_1 = \frac{\partial \underset{\sim}{x}}{\partial \xi} \bigg/ \left\| \frac{\partial \underset{\sim}{x}}{\partial \xi} \right\| \text{ and } \underset{\sim}{t}_2 = \frac{\partial \underset{\sim}{x}}{\partial \eta} \bigg/ \left\| \frac{\partial \underset{\sim}{x}}{\partial \eta} \right\|$$

where $\underset{\sim}{x}$ is the position vector of a point on the element mid-plane.

Now the first element base vector $\underset{\sim}{v}_1$ is the local normal vector and is given by:

$$\underset{\sim}{v}_1 = \frac{\underset{\sim}{t}_1 \times \underset{\sim}{t}_2}{\left\| \underset{\sim}{t}_1 \times \underset{\sim}{t}_2 \right\|}.$$

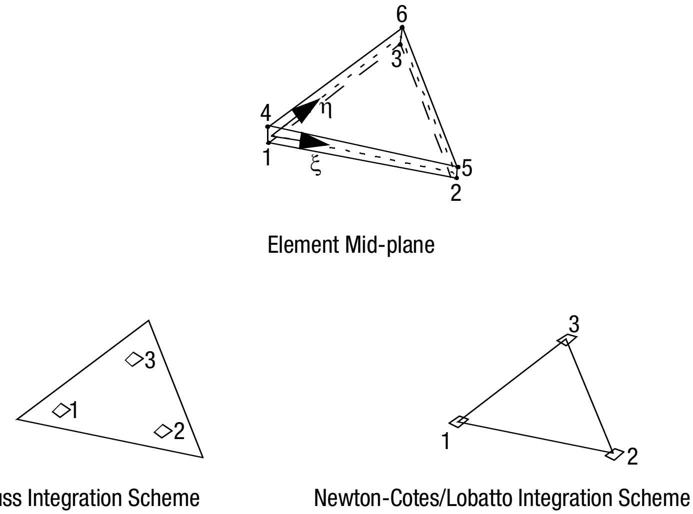


Figure 3-317 Element 226: Mid-plane and Location of Integration Points

The in-plane element base vector v_3 is constructed to be perpendicular to the plane spanned by the local normal vector v_1 and the first mid-plane edge vector e_1:

$$v_3 = \frac{v_1 \times e_1}{\|v_1 \times e_1\|},$$

in which e_1 is defined as:

$$e_1 = \frac{1}{2}(x^2 + x^6) - \frac{1}{2}(x^1 + x^5),$$

where the superscript refers to a node number of the element.

Finally, the in-plane element base vector v_2 is parallel to the plane of the local normal vector and the first min-plane edge and follows from the cross product of the former two element base vectors:

$$v_2 = v_3 \times v_1 .$$

The element stiffness matrix is integrated numerically using a three-point integration scheme. By default, the position of the integration points corresponds to the Gauss integration scheme. As an alternative, the Newton/Cotes/Lobatto integration scheme can be selected via the GEOMETRY option, in which case the integration points are located at the middle of the edges 1-4, 2-5, and 3-6 (see Figure 3-317).

Quick Reference

Type 226

Linear, six-node heat transfer 3-D interface element.

Connectivity

Six nodes per element. Node numbering must be according to Figure 3-316, so that the two sides of the interface are the element faces 1-2-3 and 4-5-6.

Geometry

If the fifth data field is set to one, the Newton-Cotes/Lobatto integration scheme is used instead of the default Gauss integration scheme.

Coordinates

Three coordinates in the global x-, y- and z-direction.

Degrees of Freedom

1 = temperature (heat transfer)

Fluxes

There are no distributed flux types available for this element.

Films

There are no distributed films for this element.

Output of Thermal Gradients

The thermal gradients are given at the element integration points. They are calculated in the local coordinate system based upon the difference in temperature between the bottom and top surfaces. However, they are output in the global coordinate system.

Tying

Use the TYING model definition option or the UFORMSN user subroutine.

Joule Heating

Capability is unavailable.

Magnetostatic

Capability is unavailable.

Electrostatic

Capability is unavailable.

Element 227 Fifteen-node Three-dimensional Heat Transfer Interface Element

Element type 227 is a thermal fifteen-node 3-D interface element, which is typically used in conjunction with element type 192 to simulate the onset and progress of delamination.

The connectivity of the element is shown in Figure 3-318. The element is typically used to model the interface between different materials, where nodes 1, 7, 2, 8, 3, and 9 correspond to one side of the interface (called the bottom) and nodes 4, 10, 5, 11, 6, and 12 to the other (called the top). The thermal gradients are expressed with respect to the local coordinate system $(\underset{\sim}{v}_1, \underset{\sim}{v}_2, \underset{\sim}{v}_3)$, indicated in Figure 3-318. The element is allowed to be infinitely thin, in which case the faces 1-7-2-8-3-9 and 4-10-5-11-6-12 coincide.

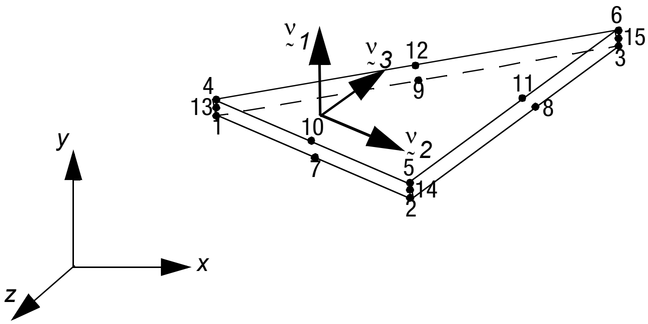


Figure 3-318 Element Type 227: Connectivity and Local Coordinate System

The constitutive behavior of the element is defined via the COHESIVE model definition option.

Because the element is considered to be thin, there is no specific heat or latent heat associated with this element. For similar reasons, there is no heat generated due to inelastic processes such as plasticity or curing included.

Geometric Basis

In order to define the local base vectors, at each integration point normalized tangent vectors to the curves with constant isoparametric coordinates are introduced (see Figure 3-319):

$$\underset{\sim}{t}_1 = \frac{\partial x}{\partial \xi} \Big/ \left\| \frac{\partial x}{\partial \xi} \right\| \text{ and } \underset{\sim}{t}_2 = \frac{\partial x}{\partial \eta} \Big/ \left\| \frac{\partial x}{\partial \eta} \right\|$$

where $\underset{\sim}{x}$ is the position vector of a point on the element midplane.

Now the first element base vector $\underset{\sim}{v}_1$ is the local normal vector and is given by:

$$\underset{\sim}{v}_1 = \frac{\underset{\sim}{t}_1 \times \underset{\sim}{t}_2}{\left\| \underset{\sim}{t}_1 \times \underset{\sim}{t}_2 \right\|} .$$

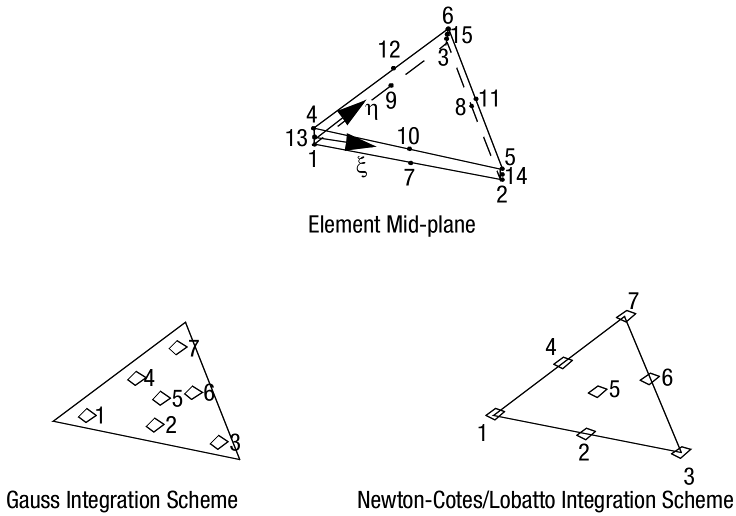


Figure 3-319 Element 227: Mid-plane and Location of Integration Points

The in-plane element base vector v_3 is constructed to be perpendicular to the plane spanned by the local normal vector v_1 and the first mid-plane edge vector ε_1:

$$v_3 = \frac{v_1 \times \varepsilon_1}{\| v_1 \times \varepsilon_1 \|},$$

in which ε_1 is defined as:

$$\varepsilon_1 = \frac{1}{2}(x^2 + x^6) - \frac{1}{2}(x^1 + x^5),$$

where the superscript refers to a node number of the element.

Finally, the in-plane element base vector v_2 is parallel to the plane of the local normal vector and the first min-plane edge and follows from the cross product of the former two element base vectors:

$$v_2 = v_3 \times v_1 .$$

The element stiffness matrix is integrated numerically using a seven-point integration scheme. By default, the position of the integration points corresponds to the Gauss integration scheme. As an alternative, the Newton/Cotes/Lobatto integration scheme can be selected via the GEOMETRY option, in which case the integration points are located at the middle of the edges 1 and 4, 7 and 10, 2 and 5, 8 and 11, 3 and 6, 12 and 15 and at the element centroid (see Figure 3-319).

Quick Reference

Type 227

Quadratic, fifteen-node heat transfer 3-D interface element.

Connectivity

Fifteen nodes per element. Node numbering must be according to Figure 3-318, so that the two sides of the interface are the element faces 1-7-2-8-3-9 and 4-10-5-11-6-12.

Geometry

If the fifth data field is set to one, the Newton-Cotes/Lobatto integration scheme is used instead of the default Gauss integration scheme.

Coordinates

Three coordinates in the global x-, y- and z-direction.

Degrees of Freedom

1 = temperature (heat transfer)

Fluxes

There are no distributed flux types available for this element.

Films

There are no distributed films for this element.

Output of Thermal Gradients

The thermal gradients are given at the element integration points. They are calculated in the local coordinate system based upon the difference in temperature between the bottom and top surfaces. However, they are output in the global coordinate system.

Tying

Use the TYING model definition option or the UFORMSN user subroutine.

Joule Heating

Capability is unavailable.

Magnetostatic

Capability is unavailable.

Electrostatic

Capability is unavailable.

Element 228

Arbitrary Triangle Planar Magnetodynamic

Element type 228 is a three-node arbitrary triangle written for planar magnetodynamic applications. This element can be used for either transient or harmonic problems.

Quick Reference

Type 228

Planar triangle.

Connectivity

Three nodes per element.

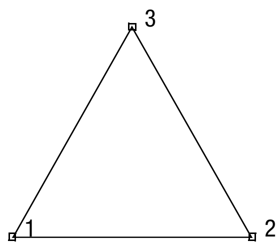


Figure 3-320 3-node Magnetodynamic Triangle

Node numbering follows right-handed convention (counterclockwise).

Geometry

The constraint $\nabla \cdot \boldsymbol{A} = 0$ is enforced with a penalty formulation. The value of the penalty factor is given in the seventh field EGEOM7 (the default is 0.0001).

Coordinates

Two coordinates in the global x- and y-directions.

Degrees Of Freedom

Magnetic Vector Potential

1 = Ax
2 = Ay
3 = Az

Electric Scalar Potential

4 = V

Distributed Current

Current types for distributed currents as follows:

0 - 9	Currents normal to element edge, or volumetric.
20 - 25	Currents in plane, tangential to element edge.
30 - 39	Currents out of plane.

Current Type	Description
0	Uniform normal current everywhere distributed on side 1-2 of the element.
1	Uniform body current in the x-direction.
2	Uniform body current in the y-direction.
3	Nonuniform normal current everywhere on side 1-2 of the element; magnitude is supplied through the FORCEM user subroutine.
4	Nonuniform body current in the x-direction.
5	Nonuniform body current in the y-direction
6	Uniform current on side 2-3 of the element.
7	Nonuniform current on side 2-3 of the element; magnitude is supplied through the FORCEM user subroutine.
8	Uniform current on side 3-1 of the element.
9	Nonuniform current on side 3-1 of the element; magnitude is supplied through the FORCEM user subroutine.
20	Uniform shear current on side 1 - 2 (positive from 1 to 2).
21	Nonuniform shear current on side 1 - 2; magnitude is supplied through the FORCEM user subroutine.
22	Uniform shear current on side 2 - 3 (positive from 2 to 3).
23	Nonuniform shear current on side 2 - 3; magnitude is supplied through the FORCEM user subroutine.
24	Uniform shear current on side 3 - 1 (positive from 3 to 4).
25	Nonuniform shear current on side 3 - 1; magnitude is supplied through the FORCEM user subroutine.
30	Uniform current per unit area of side 1-2 of the element, normal to the plane.
31	Nonuniform current per unit area on side 1-2 of the element, normal to the plane; magnitude is given in the FORCEM user subroutine.
36	Uniform current per unit area on side 2-3 of the element, normal to the plane.
37	Nonuniform current per unit area on side 2-3 of the element, normal to the plane; magnitude is given in the FORCEM user subroutine.
38	Uniform current per unit area on side 3-1 of the element normal to the plane.
39	Nonuniform current per unit area on side 3-1 of the element, normal to the plane; magnitude is given in the FORCEM user subroutine.

All currents are positive when directed into the element. In addition, point currents and charges can be applied at the nodes.

Distributed Charges

Charge types for distributed charges are as follows:

Charge Type	Description
50	Uniform charge per unit area side 1-2 of the element.
51	Uniform charge per unit volume on whole element.
52	Uniform charge per unit volume on whole element.
53	Nonuniform charge per unit area on side 1-2 of the element.
54	Nonuniform charge per unit volume on whole element.
55	Nonuniform charge per unit volume on whole element.
56	Uniform charge per unit area on side 2-3 of the element.
57	Nonuniform charge per unit area on side 2-3 of the element.
58	Uniform charge per unit area on side 3-1 of the element.
59	Nonuniform charge per unit area on side 3-1 of the element.

For all nonuniform charges, the magnitude is supplied through the FORCEM user subroutine.

All charges are positive when adding charge to the element. In addition, point charges can be applied at the nodes.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output

Three components of:

Electric field intensity	E
Electric charge density	D
Magnetic field intensity	H
Magnetic charge density	B
Current density	J

Transformation

Two global degrees of freedom (Ax, Ay) can be transformed into local coordinates.

Output Points

Output is available at the centroid.

Element 229 Arbitrary Triangle Axisymmetric Magnetodynamic Ring

Element type 229 is a three-node, isoparametric, arbitrary triangle written for axisymmetric magnetodynamic applications. This element can be used for either transient or harmonic analyses.

Quick Reference

Type 229

Axisymmetric, arbitrary ring with a triangle cross-section.

Connectivity

Three nodes per element. Node numbering follows right-handed convention (counterclockwise).

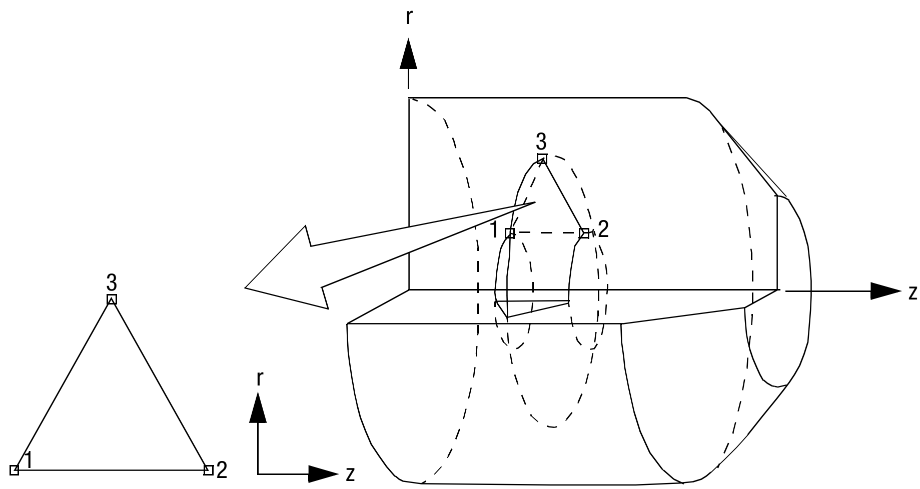


Figure 3-321 3-node Magnetodynamic Triangle Ring

Geometry

The constraint $\nabla \cdot \boldsymbol{A} = 0$ is enforced with a penalty formulation. The value of the penalty factor is given in the seventh field `EGEOM7` (the default is 0.0001).

Coordinates

Two coordinates in the global z and r directions.

Degrees of Freedom

Magnetic Vector Potential

1 = A_z

2 = A_r

3 = A_θ

Electric Scalar Potential

4 = V

Distributed Currents

Current types for distributed currents as follows:

0 - 9	Currents normal to element edge, or volumetric.
20 - 25	Currents in plane, tangential to element edge.
30 - 39	Currents in the circumferential direction.

Current Type	Description
0	Uniform normal current distributed on side 1-2 of the element.
1	Uniform body current in the z-direction.
2	Uniform body current in the r-direction.
3	Nonuniform normal current on side 1-2 of the element; magnitude is supplied through the FORCEM. user subroutine
4	Nonuniform body current in the z-direction.
5	Nonuniform body current in the r-direction.
6	Uniform normal current on side 2-3 of the element.
7	Nonuniform normal current on side 2-3 of the element; magnitude is supplied through the FORCEM user subroutine.
8	Uniform normal current on side 3-1 of the element.
9	Nonuniform normal current on side 3-1 of the element; magnitude is supplied through the FORCEM user subroutine.
20	Uniform shear current on side 1 - 2 (positive from 1 to 2).
21	Nonuniform shear current on side 1 - 2; magnitude is supplied through the FORCEM user subroutine.
22	Uniform shear current on side 2 - 3 (positive from 2 to 3).
23	Nonuniform shear current on side 2 - 3; magnitude is supplied through the FORCEM user subroutine.
24	Uniform shear current on side 3 - 1 (positive from 3 to 4).
25	Nonuniform shear current on side 3 - 1; magnitude is supplied through the FORCEM user subroutine.
30	Uniform current per unit area side 1-2 of the element in the theta direction.

Current Type	Description
33	Nonuniform current per unit area on side 1-2 of the element in the theta direction; magnitude is given in the FORCEM user subroutine.
36	Uniform current per unit area on side 2-3 of the element in the theta direction.
37	Nonuniform current per unit area on side 2-3 of the element in the theta direction; magnitude is given in the FORCEM user subroutine.
38	Uniform current per unit area on side 3-1 of the element in the theta direction.
39	Nonuniform current per unit area on side 3-1 of the element in the theta direction; magnitude is given in the FORCEM user subroutine.

All currents are positive when directed into the element. In addition, point currents can be applied at the nodes. The magnitude of point current must correspond to the current integrated around the circumference.

Distributed Charges

Charge types for distributed charges are as follows:

Charge Type	Description
50	Uniform charge per unit area side 1-2 of the element.
51	Uniform charge per unit volume on whole element.
52	Uniform charge per unit volume on whole element.
53	Nonuniform charge per unit area on side 1-2 of the element; magnitude is given in the FORCEM user subroutine.
54	Nonuniform charge per unit volume on whole element; magnitude is given in the FORCEM user subroutine.
55	Nonuniform charge per unit volume on whole element; magnitude is given in the FORCEM user subroutine.
56	Uniform charge per unit area on side 2-3 of the element.
57	Nonuniform charge per unit area on side 2-3 of the element; magnitude is given in the FORCEM user subroutine.
58	Uniform charge per unit area on side 3-1 of the element.
59	Nonuniform charge per unit area on side 3-1 of the element; magnitude is given in the FORCEM user subroutine.

All charges are positive when adding charge to the element. In addition, point charges can be applied at the nodes. The magnitude of the point charge must correspond to the charge integrated around the circumference.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output

Three components of:

Electric field intensity	E
Electric flux density	D
Magnetic field intensity	H
Magnetic flux density	B
Current density	J

Transformation

Two global degrees (Az, Ar) of freedom can be transformed into local coordinates.

Output Points

Output is available at the centroid.

Element 230 Three-dimensional Magnetodynamic Arbitrarily Distorted Tetrahedral

Element 230 is a four-node isoparametric tetrahedral element and can be used for either transient or harmonic analysis. This element uses trilinear interpolation functions for the vector and scalar potential. The stiffness of this element is formed using one-point Gaussian integration.

Note:	As in all three-dimensional analyses, a large nodal bandwidth results in long computing times. Optimize the nodal bandwidth.

Quick Reference

Type 230

Four-node 3-D first-order isoparametric element (arbitrarily distorted tetrahedral).

Connectivity

Four nodes per element (see Figure 3-322). Node numbering must follow the scheme below:

Nodes 1, 2, 3 are the corners of the first face, given in counterclockwise order when viewed from inside the element. Node 4 is on the opposing vertex. Note that in most normal cases, the elements are generated automatically via a preprocessor (such as Mentat or a CAD program) so that you need not be concerned with the node numbering scheme.

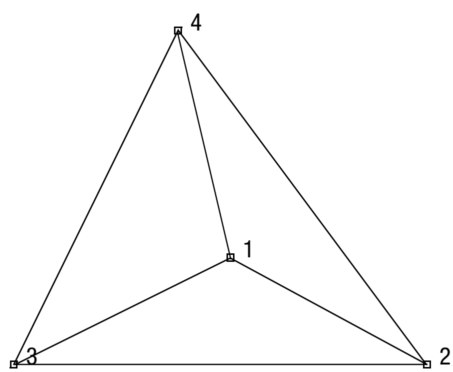


Figure 3-322 Nodes and Integration Point for Element 230

Geometry

The constraint $\nabla \cdot \boldsymbol{A} = 0$ is enforced with a penalty formulation. The value of the penalty factor is given in the seventh field `EGEOM7` (the default is 0.0001).

Coordinates

Three coordinates in the global x-, y-, and z-directions.

Degrees Of Freedom

Magnetic Vector Potential

$1 = A_x$
$2 = A_y$
$3 = A_z$

Electric Scalar Potential

$4 = V$

Distributed Currents

Distributed currents chosen by value of `IBODY` as follows:

Load Type	Description
0	Uniform normal current on 1-2-3 face.
1	Nonuniform normal current on 1-2-3 face.
2	Uniform normal current on 1-2-4 face.
3	Nonuniform normal current on 1-2-4 face.
4	Uniform normal current on 2-3-4 face.
5	Nonuniform normal current on 2-3-4 face.
6	Uniform normal current on 1-3-4 face.
7	Nonuniform normal current on 1-3-4 face.
8	Uniform volume current in the x direction.
9	Nonuniform volume current in the x direction.
10	Uniform volume current in the y direction.
11	Nonuniform volume current in the y direction.
12	Uniform volume current in the z direction.
13	Nonuniform volume current in the z direction.

For all nonuniform normal and shear currents, the magnitude is supplied through the FORCEM user subroutine.

Currents are positive into element face.

Distributed Charges

Charge types for distributed charges are listed below.

Charge Type	Description
70	Uniform current on 1-2-3 face.
71	Nonuniform charge on 1-2-3 face.

Charge Type	Description
72	Uniform charge on 1-2-4 face.
73	Nonuniform charge on 1-2-4 face.
74	Uniform charge on 2-3-4 face.
75	Nonuniform charge on 2-3-4 face.
76	Uniform charge on 1-3-4 face.
77	Nonuniform charge on 1-3-4 face
78	Uniform charge per unit volume on whole element.
79	Nonuniform charge per unit volume on whole element.

For all nonuniform charges, the magnitude is supplied through the FORCEM user subroutine.

Charges are positive into the element face.

Output

Three components of:

Electric field intensity	E
Electric flux density	D
Magnetic field intensity	H
Magnetic flux density	B
Current density	J

Transformation

Standard transformation of three global degrees of freedom to local degrees of freedom.

Output Points

Output is at the centroid.

Element 231

Arbitrary Triangle Planar Magnetodynamic

Element type 231 is a six-node arbitrary triangle written for planar magnetodynamic applications. This element can be used for either transient or harmonic problems.

Quick Reference

Type 231

Second order triangular element.

Connectivity

Six nodes per element.

Node numbering follows right-handed convention (counterclockwise).

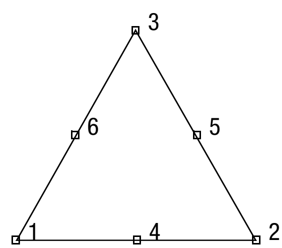


Figure 3-323 Nodes of Six-node, 2-D Element

Geometry

The constraint $\nabla \cdot \boldsymbol{A} = 0$ is enforced with a penalty formulation. The value of the penalty factor is given in the eighth field `EGEOM8` (the default is 1.0).

Coordinates

Two coordinates in the global x- and y-directions.

Degrees Of Freedom

Magnetic Vector Potential

1 = A_x

2 = A_y

3 = A_z

Electric Scalar Potential

4 = V

Distributed Current

Current types for distributed currents as follows:

0 - 11	Currents normal to element edge, or currents tangential to element edge.
12- 15	Volumetric currents.
30 - 39	Currents out of plane.

Load Type (IBODY)	Description
0	Uniform current on side 1-4-2.
1	Nonuniform current on side 1-4-2.
2	Uniform shear current on side 1-4-2.
3	Nonuniform shear current on side 1-4-2.
4	Uniform current on side 2-5-3.
5	Nonuniform current on side 2-5-3.
6	Uniform shear current on side 2-5-3.
7	Nonuniform shear current on side 2-5-3.
8	Uniform current on side 3-6-1.
9	Nonuniform current on side 3-6-1.
10	Uniform shear current on side 3-6-1.
11	Nonuniform shear current on side 3-6-1.
12	Uniform volumetric current in x-direction.
13	Nonuniform volumetric current in x-direction.
14	Uniform volumetric current in y-direction.
15	Nonuniform volumetric current in y-direction.
30	Uniform current per unit area on side 1-4-2 of the element, normal to the plane.
31	Nonuniform current per unit area on side 1-4-2 of the element, normal to the plane; magnitude is given in the FORCEM user subroutine.
34	Uniform current per unit area on side 2-5-3 of the element, normal to the plane.
35	Nonuniform current per unit area on side 2-5-3 of the element, normal to the plane; magnitude is given in the FORCEM user subroutine.
38	Uniform current per unit area on side 3-6-1 of the element, normal to the plane.
39	Nonuniform current per unit area on side 3-6-1 of the element, normal to the plane; magnitude is given in the FORCEM user subroutine.

All currents are positive when directed into the element. In addition, point currents and charges can be applied at the nodes.

Distributed Charges

Charge types for distributed charges are as follows:

Charge Type	Description
50	Uniform charge per unit area on side 1-4-2 of the element.
51	Nonuniform charge per unit area on side 1-4-2 of the element.
54	Uniform charge per unit area on side 2-5-3 of the element.
55	Nonuniform charge per unit area on side 2-5-3 of the element
58	Uniform charge per unit area on side 3-6-1 of the element.
59	Nonuniform charge per unit area on side 3-6-1 of the element.
62	Uniform charge per unit volume on whole element.
63	Nonuniform charge per unit volume on whole element.

For all nonuniform charges, the magnitude is supplied through the FORCEM user subroutine.

All charges are positive when adding charge to the element. In addition, point charges can be applied at the nodes.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output

Three components of:

Electric field intensity	E
Electric charge density	D
Magnetic field intensity	H
Magnetic charge density	B
Current density	J

Transformation

Two global degrees of freedom (Ax, Ay) can be transformed into local coordinates.

Output Points

Output is available at the three Gaussian points shown in Figure 3-324.

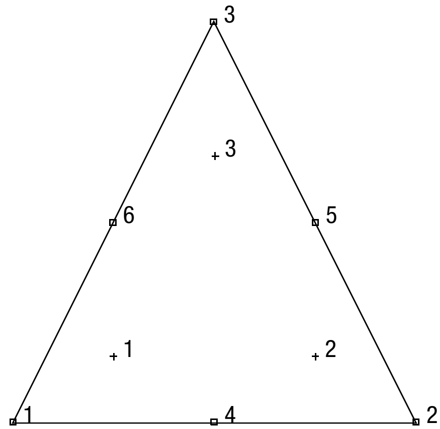


Figure 3-324 Gaussian Integration Points for Element Type 231

Element 232 Arbitrary Triangle Axisymmetric Magnetodynamic Ring

Element type 232 is a six-node, isoparametric, arbitrary triangle written for axisymmetric magnetodynamic applications. This element can be used for either transient or harmonic analyses.

Quick Reference

Type 232

Second order axisymmetric, arbitrary ring with a triangle cross-section.

Connectivity

Six nodes per element. Node numbering follows right-handed convention (counterclockwise).

Geometry

The constraint $\nabla \cdot \boldsymbol{A} = 0$ is enforced with a penalty formulation. The value of the penalty factor is given in the eighth field `EGEOM8` (the default is 1.0).

Coordinates

Two coordinates in the global z and r directions.

Degrees of Freedom

Vector Potential

$1 = A_z$

$2 = A_r$

$3 = A_\theta$

Scalar Potential

$4 = V$

Distributed Currents

Current types for distributed currents are listed below:

0 - 11 Currents normal to element edge, or currents tangential to element edge.
12- 15 Volumetric currents.
30 - 39 Currents in the circumferential direction.

Load Type (IBODY)	Description
0	Uniform current on side 1-4-2.
1	Nonuniform current on side 1-4-2.
2	Uniform shear current on side 1-4-2.

Load Type (IBODY)	Description
3	Nonuniform shear current on side 1-4-2.
4	Uniform current on side 2-5-3.
5	Nonuniform current on side 2-5-3.
6	Uniform shear current on side 2-5-3.
7	Nonuniform shear current on side 2-5-3.
8	Uniform current on side 3-6-1.
9	Nonuniform current on side 3-6-1.
10	Uniform shear current on side 3-6-1.
11	Nonuniform shear current on side 3-6-1.
12	Uniform volumetric current in z-direction.
13	Nonuniform volumetric current in z-direction.
14	Uniform current in r-direction.
15	Nonuniform volumetric current in r-direction.
30	Uniform current per unit area on side 1-4-2 of the element, normal to the plane.
31	Nonuniform current per unit area on side 1-4-2 of the element, normal to the plane; magnitude is given in the FORCEM user subroutine.
34	Uniform current per unit area on side 2-5-3 of the element, normal to the plane.
35	Nonuniform current per unit area on side 2-5-3 of the element, normal to the plane; magnitude is given in the FORCEM user subroutine.
38	Uniform current per unit area on side 3-6-1 of the element, normal to the plane.
39	Nonuniform current per unit area on side 3-6-1 of the element, normal to the plane; magnitude is given in the FORCEM user subroutine.

All currents are positive when directed into the element. In addition, point currents can be applied at the nodes. The magnitude of point current must correspond to the current integrated around the circumference.

Distributed Charges

Charge types for distributed charges are as follows:

Charge Type	Description
50	Uniform charge per unit area on side 1-4-2 of the element.
51	Nonuniform charge per unit area on side 1-4-2 of the element.
54	Uniform charge per unit area on side 2-5-3 of the element.
55	Nonuniform charge per unit area on side 2-5-3 of the element

Charge Type	Description
58	Uniform charge per unit area on side 3-6-1 of the element.
59	Nonuniform charge per unit area on side 3-6-1 of the element.
62	Uniform charge per unit volume on whole element.
63	Nonuniform charge per unit volume on whole element.

All charges are positive when adding charge to the element. In addition, point charges can be applied at the nodes. The magnitude of the point charge must correspond to the charge integrated around the circumference.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output

Three components of:

Electric field intensity	E
Electric flux density	D
Magnetic field intensity	H
Magnetic flux density	B
Current density	J

Transformation

Two global degrees (Az, Ar) of freedom can be transformed into local coordinates.

Output Points

Output is available at the three Gaussian points shown in Figure 3-325.

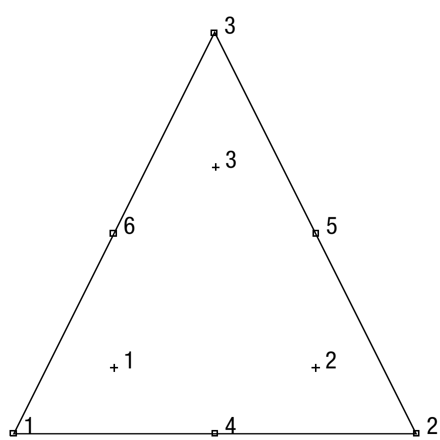


Figure 3-325 Integration Points for Element 232

Element 233 Three-dimensional Magnetodynamic Arbitrarily Distorted Tetrahedral

Element 233 is a ten-node isoparametric tetrahedral element and can be used for either transient or harmonic analysis. This element uses trilinear interpolation functions for the vector and scalar potential. The stiffness of this element is formed using four-point Gaussian integration.

Note:	As in all three-dimensional analyses, a large nodal bandwidth results in long computing times. Optimize the nodal bandwidth.

Quick Reference

Type 233

Ten-node 3-D first-order isoparametric element (arbitrarily distorted brick).

Connectivity

Ten nodes per element (see Figure 3-322). Node numbering must follow the scheme below:

Nodes 1, 2, 3 are the corners of the first face, given in counterclockwise order when viewed from inside the element. Node 4 is on the opposing vertex. Nodes 5, 6, 7 are on the first face between nodes 1 and 2, 2 and 3, 3 and 1, respectively. Nodes 8, 9, 10 are along the edges between the first face and node 4, between nodes 1 and 4, 2 and 4, 3 and 4, respectively.

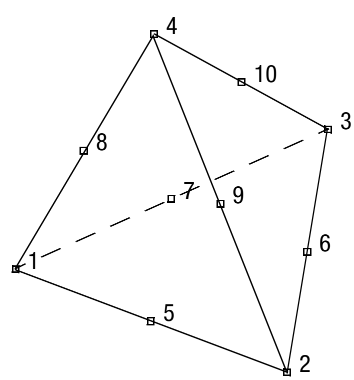


Figure 3-326 Nodes and Integration Point for Element 233

Geometry

The constraint $\nabla \cdot \boldsymbol{A} = 0$ is enforced with a penalty formulation. The value of the penalty factor is given in the eighth field EGEOM8 (the default is 1.0).

Coordinates

Three coordinates in the global x-, y-, and z-directions.

Degrees of Freedom

Magnetic Vector Potential

1 = A_x
2 = A_y
3 = A_z

Electric Scalar Potential

4 = V

Distributed Currents

Distributed currents chosen by value of `IBODY` as follows:

Load Type	Description
0	Uniform current on 1-2-3 face.
1	Nonuniform current on 1-2-3 face.
2	Uniform current on 1-2-4 face.
3	Nonuniform current on 1-2-4 face.
4	Uniform current on 2-3-4 face.
5	Nonuniform current on 2-3-4 face.
6	Uniform current on 1-3-4 face.
7	Nonuniform current on 1-3-4 face.
8	Uniform volumetric current per unit volume in x-direction.
9	Nonuniform volumetric current per unit volume in x-direction.
10	Uniform volumetric current per unit volume in y-direction.
11	Nonuniform volumetric current per unit volume in y-direction.
12	Uniform volumetric current per unit volume in z-direction.
13	Nonuniform volumetric current per unit volume in z-direction.

For all nonuniform normal and shear currents, the magnitude is supplied through the FORCEM user subroutine.

Currents are positive into element face.

Distributed Charges

Charge types for distributed charges are listed below.

Charge Type	Description
70	Uniform charge on 1-2-3 face.
71	Nonuniform charge on 1-2-3 face.

Charge Type	Description
72	Uniform charge on 1-2-4 face.
73	Nonuniform charge on 1-2-4 face.
74	Uniform charge on 2-3-4 face.
75	Nonuniform charge on 2-3-4 face.
76	Uniform charge on 1-3-4 face.
77	Nonuniform charge on 1-3-4 face
78	Uniform charge per unit volume on whole element.
79	Nonuniform charge per unit volume on whole element.

For all nonuniform charges, the magnitude is supplied through the FORCEM user subroutine.

Charges are positive into the element face.

Output

Three components of:

Electric field intensity	E
Electric flux density	D
Magnetic field intensity	H
Magnetic flux density	B
Current density	J

Transformation

Standard transformation of three global degrees of freedom to local degrees of freedom.

Output Points

Four integration points as shown in Figure 3-327.

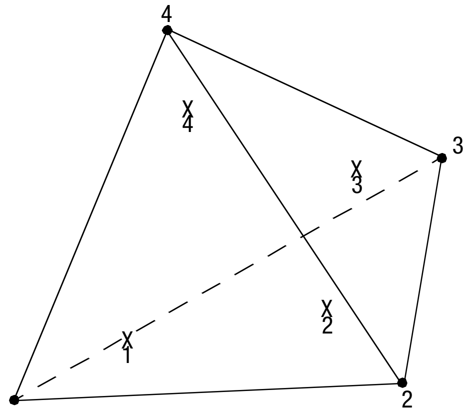


Figure 3-327 1-2 Element 233 Integration Plane

Element 234

Arbitrary Quadrilateral Planar Magnetodynamic

Element type 234 is an eight-node arbitrary quadrilateral written for planar magnetodynamic applications. This element can be used for either transient or harmonic problems.

Quick Reference

Type 234

Second order planar quadrilateral.

Connectivity

Eight nodes per element.

Node numbering follows right-handed convention (counterclockwise).

Geometry

The constraint $\nabla \cdot \boldsymbol{A} = 0$ is enforced with a penalty formulation. The value of the penalty factor is given in the eighth field `EGEOM8` (the default is 1.0).

Coordinates

Two coordinates in the global x- and y-directions.

Degrees Of Freedom

Magnetic Vector Potential

1 = A_x
2 = A_y
3 = A_z

Electric Scalar Potential

4 = V

Distributed Current

Current types for distributed currents as follows:

0 - 27 Currents normal to element edge, currents in plane tangential to element edge or volumetric currents.
30 - 41 Currents out of plane.

Load Type (IBODY)	Description
0	Uniform current on side 1-5-2.
1	Nonuniform current on side 1-5-2.

Load Type (IBODY)	Description
2	Uniform volumetric current in x-direction.
3	Nonuniform volumetric current in the x-direction.
4	Uniform volumetric current in y-direction.
5	Nonuniform volumetric current in the y-direction.
6	Uniform shear current in 1⇒5⇒2 direction on side 1-5-2.
7	Nonuniform shear current in 1⇒5⇒2 direction on side 1-5-2.
8	Uniform current on side 2-6-3.
9	Nonuniform current on side 2-6-3.
10	Uniform current on side 3-7-4.
11	Nonuniform current on side 3-7-4.
12	Uniform current on side 4-8-1.
13	Nonuniform current on side 4-8-1.
20	Uniform shear current on side 1-5-2 in the 1⇒5⇒2 direction.
21	Nonuniform shear current on side 1-5-2.
22	Uniform shear current on side 2-6-3 in the 2⇒6⇒3 direction.
23	Nonuniform shear current on side 2-6-3.
24	Uniform shear current on side 3-7-4 in the 3⇒7⇒4 direction.
25	Nonuniform shear current on side 3-7-4.
26	Uniform shear current on side 4-8-1 in the 4⇒8⇒1 direction.
27	Nonuniform shear current on side 4-8-1.
30	Uniform current on side 1-5-2 of the element, normal to the plane.
31	Nonuniform current on side 1-5-2 of the element, normal to the plane.
38	Uniform current on side 2-6-3 of the element, normal to the plane.
39	Nonuniform current on side 2-6-3 of the element, normal to the plane.
40	Uniform current on side 3-7-4 of the element, normal to the plane.
41	Nonuniform current on side 3-7-4 of the element, normal to the plane.
42	Uniform current on side 4-8-1 of the element, normal to the plane.
43	Nonuniform current on side 4-8-1 of the element, normal to the plane.

All currents are positive when directed into the element. In addition, point currents and charges can be applied at the nodes.

Distributed Charges

Charge types for distributed charges are as follows:

Charge Type	Description
50	Uniform charge per unit area on side 1-5-2 of the element.
51	Nonuniform charge per unit area on side 1-5-2 of the element.
52	Uniform charge per unit volume on whole element.
53	Nonuniform charge per unit volume on whole element.
58	Uniform charge per unit area on side 2-6-3.
59	Nonuniform charge per unit area on side 2-6-3.
60	Uniform charge per unit area on side 3-7-4.
61	Nonuniform charge per unit area on side 3-7-4.
62	Uniform charge per unit area on side 4-8-1.
63	Nonuniform charge per unit area on side 4-8-1.

For all nonuniform charges, the magnitude is supplied through the FORCEM user subroutine.

All charges are positive when adding charge to the element. In addition, point charges can be applied at the nodes.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output

Three components of:

Electric field intensity	E
Electric charge density	D
Magnetic field intensity	H
Magnetic charge density	B
Current density	J

Transformation

Two global degrees of freedom (Ax, Ay) can be transformed into local coordinates.

Output Points

Output is available at the nine Gaussian points shown in Figure 3-328.

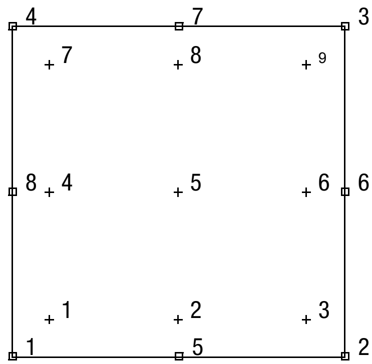


Figure 3-328 Gaussian Integration Points for Element Type 234

Element 235 Arbitrary Quadrilateral Axisymmetric Magnetodynamic Ring

Element type 235 is an eight-node, isoparametric, arbitrary quadrilateral written for axisymmetric magnetodynamic applications. This element can be used for either transient or harmonic analyses.

Quick Reference

Type 235

Second order axisymmetric, arbitrary ring with a quadrilateral cross-section.

Connectivity

Eight nodes per element. Node numbering follows right-handed convention (counterclockwise).

Geometry

The constraint $\nabla \cdot \boldsymbol{A} = 0$ is enforced with a penalty formulation. The value of the penalty factor is given in the eighth field `EGEOM8` (the default is 1.0).

Coordinates

Two coordinates in the global z and r directions.

Degrees of Freedom

Vector Potential

$1 = A_z$
$2 = A_r$
$3 = A_\theta$

Scalar Potential

$4 = V$

Distributed Currents

Current types for distributed currents are listed below:

0 - 27 Currents normal to element edge, currents in plane tangential to element edge or volumetric currents.
30 - 41 Currents in the circumferential direction.

Load Type (IBODY)	Description
0	Uniform current on side 1-5-2.
1	Nonuniform current on side 1-5-2.
2	Uniform volumetric current in z-direction.

Load Type (IBODY)	Description
3	Nonuniform volumetric current in the z-direction.
4	Uniform volumetric current in r-direction.
5	Nonuniform volumetric current in the r-direction.
6	Uniform shear current in 1⇒5⇒2 direction on side 1-5-2.
7	Nonuniform shear current in 1⇒5⇒2 direction on side 1-5-2.
8	Uniform current on side 2-6-3.
9	Nonuniform current on side 2-6-3.
10	Uniform current on side 3-7-4.
11	Nonuniform current on side 3-7-4.
12	Uniform current on side 4-8-1.
13	Nonuniform current on side 4-8-1.
20	Uniform shear current on side 1-5-2 in the 1⇒5⇒2 direction.
21	Nonuniform shear current on side 1-5-2.
22	Uniform shear current on side 2-6-3 in the 2⇒6⇒3 direction.
23	Nonuniform shear current on side 2-6-3.
24	Uniform shear current on side 3-7-4 in the 3⇒7⇒4 direction.
25	Nonuniform shear current on side 3-7-4.
26	Uniform shear current on side 4-8-1 in the 4⇒8⇒1 direction.
27	Nonuniform shear current on side 4-8-1.
30	Uniform current on side 1-5-2 of the element, normal to the plane.
31	Nonuniform current on side 1-5-2 of the element, normal to the plane.
38	Uniform current on side 2-6-3 of the element, normal to the plane.
39	Nonuniform current on side 2-6-3 of the element, normal to the plane.
40	Uniform current on side 3-7-4 of the element, normal to the plane.
41	Nonuniform current on side 3-7-4 of the element, normal to the plane.
42	Uniform current on side 4-8-1 of the element, normal to the plane.
43	Nonuniform current on side 4-8-1 of the element, normal to the plane.

All currents are positive when directed into the element. In addition, point currents can be applied at the nodes. The magnitude of point current must correspond to the current integrated around the circumference.

Distributed Charges

Charge types for distributed charges are as follows:

Charge Type	Description
50	Uniform charge per unit area on side 1-5-2 of the element.
51	Nonuniform charge per unit area on side 1-5-2 of the element.
52	Uniform charge per unit volume on whole element.
53	Nonuniform charge per unit volume on whole element.
58	Uniform charge per unit area on side 2-6-3.
59	Nonuniform charge per unit area on side 2-6-3.
60	Uniform charge per unit area on side 3-7-4.
61	Nonuniform charge per unit area on side 3-7-4.
62	Uniform charge per unit area on side 4-8-1.
63	Nonuniform charge per unit area on side 4-8-1.

All charges are positive when adding charge to the element. In addition, point charges can be applied at the nodes. The magnitude of the point charge must correspond to the charge integrated around the circumference.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output

Three components of:

Electric field intensity	E
Electric flux density	D
Magnetic field intensity	H
Magnetic flux density	B
Current density	J

Transformation

Two global degrees (A_z, A_r) of freedom can be transformed into local coordinates.

Output Points

Output is available at the nine Gaussian points shown in Figure 3-329.

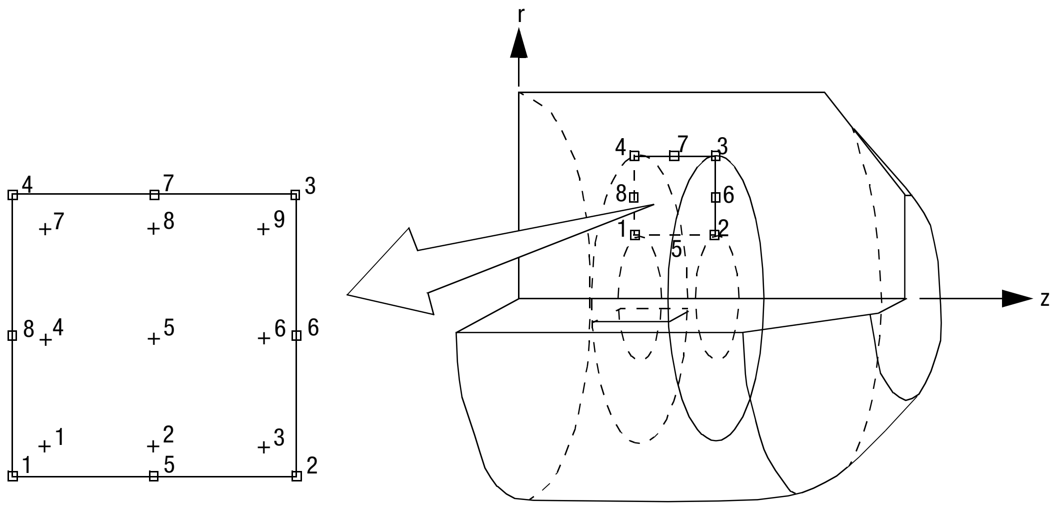


Figure 3-329 Integration Points for Element 235

Element 236 Three-dimensional Magnetodynamic Arbitrarily Distorted Brick

Element 236 is a twenty-node isoparametric brick element and can be used for either transient or harmonic analysis. This element uses trilinear interpolation functions for the vector and scalar potential. The stiffness of this element is formed using twenty-seven point Gaussian integration.

Note:	As in all three-dimensional analyses, a large nodal bandwidth results in long computing times. Optimize the nodal bandwidth.

Quick Reference

Type 236

Twenty-node 3-D first-order isoparametric element (arbitrarily distorted brick).

Connectivity

Eight nodes per element (see Figure 3-330). Node numbering must follow the scheme below:

Nodes 1, 2, 3, and 4 are corners of one face, given in counterclockwise order when viewed from inside the element. Node 5 has the same edge as node 1, node 6 as node 2, node 7 as node 3, and node 8 as node 4.

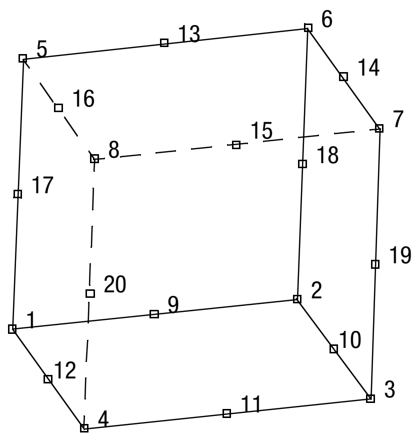


Figure 3-330 Nodes and Integration Point for Element 236

Geometry

The constraint $\nabla \cdot \boldsymbol{A} = 0$ is enforced with a penalty formulation. The value of the penalty factor is given in the eighth field EGEOM8 (the default is 1.0).

Coordinates

Three coordinates in the global x-, y-, and z-directions.

Degrees of Freedom

Magnetic Vector Potential

$1 = A_x$

$2 = A_y$

$3 = A_z$

Electric Scalar Potential

$4 = V$

Distributed Currents

Distributed currents chosen by value of `IBODY` as follows:

Current Type	Description
0	Uniform normal current on 1-2-3-4 face.
1	Nonuniform normal current on 1-2-3-4 face.
2	Uniform volume current in the z-direction.
3	Nonuniform volume current.
4	Uniform normal current on 6-5-8-7 face.
5	Nonuniform normal current on 6-5-8-7 face.
6	Uniform normal current on 2-1-5-6 face.
7	Nonuniform normal current on 2-1-5-6 face.
8	Uniform normal current on 3-2-6-7 face.
9	Nonuniform normal current on 3-2-6-7 face.
10	Uniform normal current on 4-3-7-8 face.
11	Nonuniform normal current on 4-3-7-8 face.
12	Uniform normal current on 1-4-8-5 face.
13	Nonuniform normal current on 1-4-8-5 face.
20	Uniform normal current on 1-2-3-4 face.
21	Nonuniform current on 1-2-3-4 face.
22	Uniform volume current in the z-direction.
23	Nonuniform volume current.
24	Uniform normal current on 6-5-8-7 face.
25	Nonuniform current on 6-5-8-7 face.
26	Uniform normal current on 2-1-5-6 face.
27	Nonuniform current on 2-1-5-6 face.
28	Uniform normal current on 3-2-6-7 face.

Current Type	Description
29	Nonuniform current on 3-2-6-7 face.
30	Uniform normal current on 4-3-7-8 face.
31	Nonuniform current on 4-3-7-8 face.
32	Uniform normal current on 1-4-8-5 face.
33	Nonuniform current on 1-4-8-5 face.
40	Uniform shear current 1-2-3-4 face in 1⇒2 direction.
41	Nonuniform shear current 1-2-3-4 face in 1⇒2 direction.
42	Uniform shear current 1-2-3-4 face in 2⇒3 direction.
43	Nonuniform shear current 1-2-3-4 face in 2⇒3 direction.
48	Uniform shear current 6-5-8-7 face in 5⇒6 direction.
49	Nonuniform shear current 6-5-8-7 face in 5⇒6 direction.
50	Uniform shear current 6-5-8-7 face in 6⇒7 direction.
51	Nonuniform shear current 6-5-8-7 face in 6⇒7 direction.
52	Uniform shear current 2-1-5-6 face in 1⇒2 direction.
53	Nonuniform shear current 2-1-5-6 face in 1⇒2 direction.
54	Uniform shear current 2-1-5-6 face in 1⇒5 direction.
55	Nonuniform shear current 2-1-5-6 face in 1⇒5 direction.
56	Uniform shear current 3-2-6-7 face in 2⇒3 direction.
57	Nonuniform shear current 3-2-6-7 face in 2⇒3 direction.
58	Uniform shear current 3-2-6-7 face in 2⇒6 direction.
59	Nonuniform shear current 2-3-6-7 face in 2⇒6 direction.
60	Uniform shear current 4-3-7-8 face in 3⇒4 direction.
61	Nonuniform shear current 4-3-7-8 face in 3⇒4 direction.
62	Uniform shear current 4-3-7-8 face in 3⇒7 direction.
63	Nonuniform shear current 4-3-7-8 face in 3⇒7 direction.
64	Uniform shear current 1-4-8-5 face in 4⇒1 direction.
65	Nonuniform shear current 1-4-8-5 face in 4⇒1 direction.
66	Uniform shear current 1-4-8-5 face in 1⇒5 direction.
67	Nonuniform shear current 1-4-8-5 face in 1⇒5 direction.

For all nonuniform normal and shear currents, the magnitude is supplied through the FORCEM user subroutine.

Currents are positive into element face.

Distributed Charges

Charge types for distributed charges are listed below.

Charge Type	Description
70	Uniform charge on 1-2-3-4 face.
71	Nonuniform charge on 1-2-3-4 face.
72	Uniform charge per unit volume on whole element.
73	Nonuniform charge per unit volume on whole element.
74	Uniform charge on 5-6-7-8 face.
75	Nonuniform charge on 5-6-7-8 face.
76	Uniform charge on 1-2-6-5 face.
77	Nonuniform charge on 1-2-6-5 face.
78	Uniform charge on 2-3-7-6 face.
79	Nonuniform charge on 2-3-7-6 face
80	Uniform charge on 3-4-8-7 face.
81	Nonuniform charge on 3-4-8-7 face.
82	Uniform charge on 1-4-8-5 face.
83	Nonuniform charge on 1-4-8-5 face.

For all nonuniform charges, the magnitude is supplied through the FORCEM user subroutine.

Charges are positive into the element face.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output

Three components of:

Electric field intensity	E
Electric flux density	D
Magnetic field intensity	H
Magnetic flux density	B
Current density	J

Transformation

Standard transformation of three global degrees of freedom to local degrees of freedom.

Output Points

Twenty-seven integration points as shown in Figure 3-331.

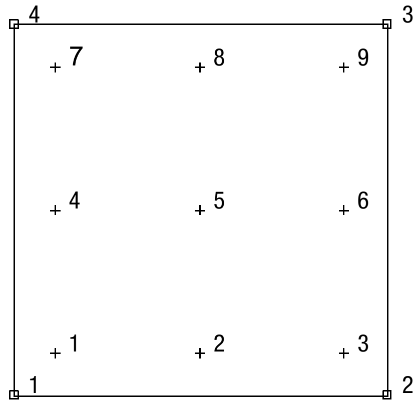


Figure 3-331 Element 236 Integration Plane

Element 237 — Three-dimensional Magnetodynamic Arbitrarily Distorted Pentahedral

Element 237 is a six-node isoparametric pentahedral element and can be used for either transient or harmonic analysis. This element uses trilinear interpolation functions for the vector and scalar potential. The stiffness of this element is formed using six point Gaussian integration.

Note: As in all three-dimensional analyses, a large nodal bandwidth results in long computing times. Optimize the nodal bandwidth.

Quick Reference

Type 237

Six-node 3-D first-order isoparametric element (arbitrarily distorted brick).

Connectivity

Six nodes per element (see Figure 3-330). Node numbering must follow the scheme below:

Nodes 1, 2, and 3 are corners of one face, given in counterclockwise order when viewed from inside the element. Node 4 has the same edge as node 1. Node 5 has the same edge as node 2. Node 6 has the same edge as node 3.

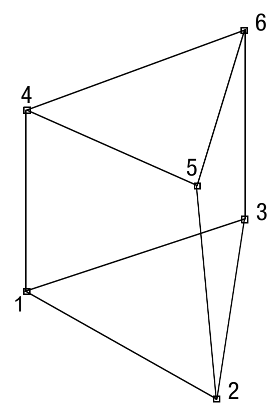


Figure 3-332 Nodes and Integration Point for Element 237

Geometry

The constraint $\nabla \cdot \boldsymbol{A} = 0$ is enforced with a penalty formulation. The value of the penalty factor is given in the seventh field `EGEOM7` (the default is 0.0001).

Coordinates

Three coordinates in the global x-, y-, and z-directions.

Degrees of Freedom

Magnetic Vector Potential

1 = A_x
2 = A_y
3 = A_z

Electric Scalar Potential

4 = V

Distributed Currents

Distributed currents chosen by value of `IBODY` as follows:

Load Type	Description
1	Uniform current on 1-2-5-4 face.
2	Nonuniform current on 1-2-5-4 face; magnitude and direction supplied through the FORCEM user subroutine.
3	Uniform current on 2-3-6-5 face.
4	Nonuniform current on 2-3-6-5 face; magnitude and direction supplied through the FORCEM user subroutine.
5	Uniform current on 3-1-4-6 face.
6	Nonuniform current on 3-1-4-6 face; magnitude and direction supplied through the FORCEM user subroutine.
7	Uniform current on 1-3-2 face.
8	Nonuniform current on 1-3-2 face; magnitude and direction supplied through the FORCEM user subroutine.
9	Uniform current on 4-5-6 face.
10	Nonuniform current on 4-5-6 face; magnitude and direction supplied through the FORCEM user subroutine.
11	Uniform volumetric current in x-direction.
12	Nonuniform volumetric current in x-direction; magnitude and direction supplied through the FORCEM user subroutine.
13	Uniform volumetric current in y-direction.
14	Nonuniform volumetric current in y-direction; magnitude and direction supplied through the FORCEM user subroutine.
15	Uniform volumetric current in z-direction.
16	Nonuniform volumetric current in z-direction; magnitude and direction supplied through the FORCEM user subroutine.

For all nonuniform normal and shear currents, the magnitude is supplied through the FORCEM user subroutine.

Currents are positive into element face.

Distributed Charges

Charge types for distributed charges are listed below.

Charge Type	Description
71	Uniform charge on 1-2-5-4 face.
72	Nonuniform charge on 1-2-5-4 face.
73	Uniform charge on 2-3-6-5 face.
74	Nonuniform charge on 2-3-6-5 face.
75	Uniform charge on 3-1-4-6 face.
76	Nonuniform charge on 3-1-4-6 face.
77	Uniform charge on 1-3-2 face.
78	Nonuniform charge on 1-3-2 face.
79	Uniform charge on 4-5-6 face.
80	Nonuniform charge on 4-5-6 face.
81	Uniform volumetric charge.
82	Nonuniform volumetric charge.

For all nonuniform charges, the magnitude is supplied through the FORCEM user subroutine.

Charges are positive into the element face.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output

Three components of:

Electric field intensity	E
Electric flux density	D
Magnetic field intensity	H
Magnetic flux density	B
Current density	J

Transformation

Standard transformation of three global degrees of freedom to local degrees of freedom.

Output Points

Six integration points as shown in Figure 3-333.

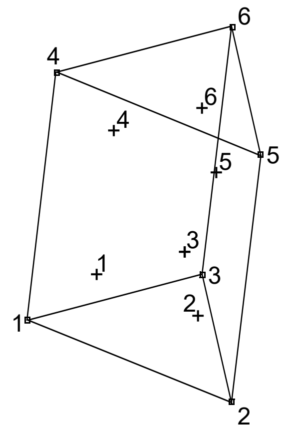


Figure 3-333 Six-Point Gauss Integration Scheme for Element 237

Element 238 Three-dimensional Magnetodynamic Arbitrarily Distorted Pentahedral

Element 238 is a ten-node isoparametric pentahedral element and can be used for either transient or harmonic analysis. This element uses trilinear interpolation functions for the vector and scalar potential. The stiffness of this element is formed using eighteen point Gaussian integration.

Note:	As in all three-dimensional analyses, a large nodal bandwidth results in long computing times. Optimize the nodal bandwidth.

Quick Reference

Type 238

Ten-node 3-D first-order isoparametric element (arbitrarily distorted pentrahedral).

Connectivity

Ten nodes per element (see Figure 3-334). Node numbering must follow the scheme below:

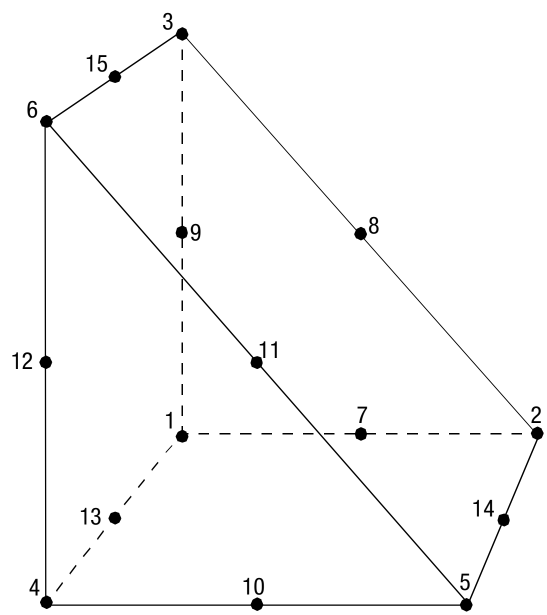


Figure 3-334 Pentahedral

Geometry

The constraint $\nabla \cdot \boldsymbol{A} = 0$ is enforced with a penalty formulation. The value of the penalty factor is given in the eighth field `EGEOM8` (the default is 1.0).

Coordinates

Three coordinates in the global x-, y-, and z-directions.

Degrees of Freedom

Magnetic Vector Potential

$1 = A_x$
$2 = A_y$
$3 = A_z$

Electric Scalar Potential

$4 = V$

Distributed Currents

Distributed currents chosen by value of `IBODY` as follows:

Load Type	Description
1	Uniform current on 1-2-5-4 face.
2	Nonuniform current on 1-2-5-4 face; magnitude and direction supplied through the FORCEM user subroutine.
3	Uniform current on 2-3-6-5 face.
4	Nonuniform current on 2-3-6-5 face; magnitude and direction supplied through the FORCEM user subroutine.
5	Uniform current on 3-1-4-6 face.
6	Nonuniform current on 3-1-4-6 face; magnitude and direction supplied through the FORCEM user subroutine.
7	Uniform current on 1-3-2 face.
8	Nonuniform current on 1-3-2 face; magnitude and direction supplied through the FORCEM user subroutine.
9	Uniform current on 4-5-6 face.
10	Nonuniform current on 4-5-6 face; magnitude and direction supplied through the FORCEM user subroutine.
11	Uniform volumetric current in x-direction.
12	Nonuniform volumetric current in x-direction; magnitude and direction supplied through the FORCEM user subroutine.
13	Uniform volumetric current in y-direction.
14	Nonuniform volumetric current in y-direction; magnitude and direction supplied through the FORCEM user subroutine.

Load Type	Description
15	Uniform volumetric current in z-direction.
16	Nonuniform volumetric current in z-direction; magnitude and direction supplied through the FORCEM user subroutine.

For all nonuniform normal and shear currents, the magnitude is supplied through the FORCEM user subroutine.

Currents are positive into element face.

Distributed Charges

Charge types for distributed charges are listed below.

Charge Type	Description
71	Uniform charge on 1-2-5-4 face.
72	Nonuniform charge on 1-2-5-4 face.
73	Uniform charge on 2-3-6-5 face.
74	Nonuniform charge on 2-3-6-5 face.
75	Uniform charge on 3-1-4-6 face.
76	Nonuniform charge on 3-1-4-6 face.
77	Uniform charge on 1-3-2 face.
78	Nonuniform charge on 1-3-2 face.
79	Uniform charge on 4-5-6 face.
80	Nonuniform charge on 4-5-6 face.
81	Uniform volumetric charge.
82	Nonuniform volumetric charge.

For all nonuniform charges, the magnitude is supplied through the FORCEM user subroutine.

Charges are positive into the element face.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output

Three components of:

Electric field intensity	E
Electric flux density	D
Magnetic field intensity	H
Magnetic flux density	B
Current density	J

Transformation

Standard transformation of three global degrees of freedom to local degrees of freedom.

Output Points

Eighteen integration points as shown in Figure 3-335.

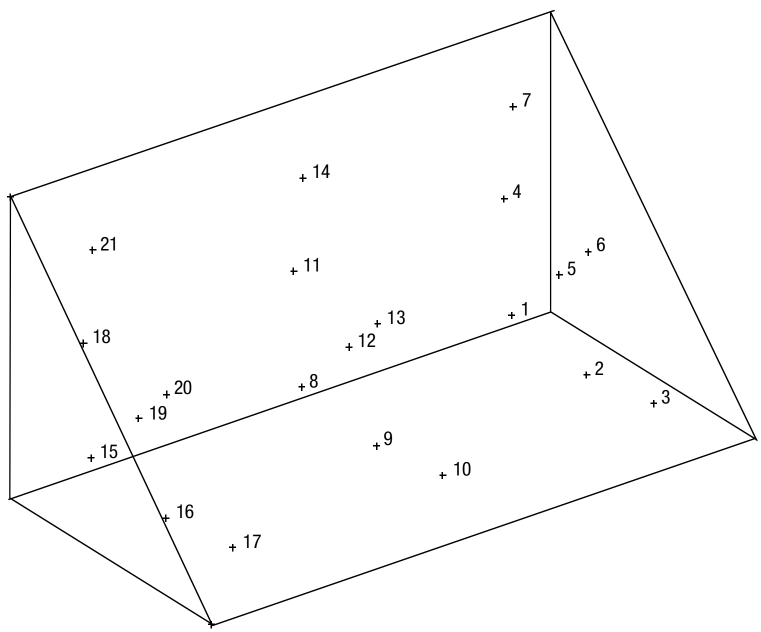


Figure 3-335 Nodes and Integration Point for Element 238

Element 239 Two-dimensional Plane Strain Triangle with Strain Smoothing

Element 239 is a 3+1-node triangular element written for plane strain applications. This element uses piecewise bilinear interpolation functions. An enhanced integration scheme is used to enforce strain smoothing over neighboring elements. This allows the element to be used for nearly incompressible behavior and it also improves the bending capability compared to the conventional element type 6.

The stiffness matrix of this element is formed using a three point integration scheme. The mass matrix (which is always lumped) of this element is formed using a four point integration scheme.

Quick Reference

Type 239

Two-dimensional, plane strain, 3 + 1-node triangle with strain smoothing.

Connectivity

Node numbering of the first three nodes must be counterclockwise (see Figure 3-336). The fourth node is located in the element centroid. If the fourth node is not provided by the user in the element connectivity, it will be automatically added by Marc.

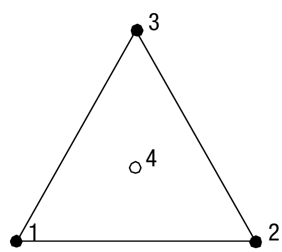


Figure 3-336 Plane Strain Triangle 239: Element Connectivity

Geometry

Thickness stored in first data field (`EGOM1`). Default thickness is unity. Other fields are not used.

Coordinates

Two coordinates in the global x and y directions.

Degrees of Freedom

1 = u displacement
2 = v displacement

Tractions

Load types for distributed loads are as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force per unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element; magnitude supplied through the FORCEM user subroutine.
4	Nonuniform body force per unit volume in first coordinate direction; magnitude supplied through the FORCEM user subroutine.
5	Nonuniform body force per unit volume in second coordinate direction; magnitude supplied through the FORCEM user subroutine.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element; magnitude supplied through the FORCEM user subroutine.
8	Uniform pressure on 3-1 face of the element.
9	Nonuniform pressure on 3-1 face of the element; magnitude supplied through the FORCEM user subroutine.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at three integration points (see Figure 3-337), is as follows:

$1 = \varepsilon_{xx}$
$2 = \varepsilon_{yy}$
$3 = \varepsilon_{zz} = 0$
$4 = \gamma_{xy}$

Output of Stresses

Output of stresses is also at the three integration points and follows the same scheme as Output of Strains . Note that σ_{zz} is usually not equal to zero.

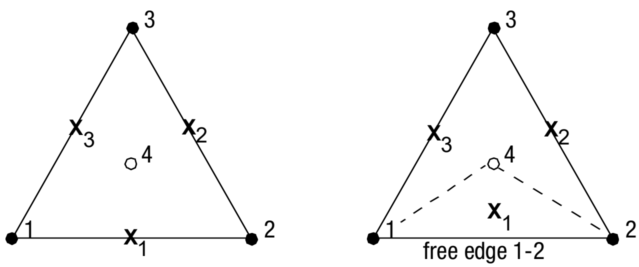


Figure 3-337 Plane Strain Triangle 239: Integration Points for Strain and Stress Output

Note:	The integration points are located at the middle of the edges of the element for edges shared by another element of type 239 with the same properties. They are located at the centroid of the corresponding sub-triangle for edges not shared by another element of type 239 with the same properties (called a *free edge* in Figure 3-337). Since the strain smoothing procedure depends on the neighboring elements, the use of symmetry boundary conditions may introduce slightly different results compared to using a full finite element mesh without the symmetry conditions.

Transformation

Nodal degrees of freedom can be transformed to local degrees of freedom. Corresponding nodal loads must be applied in local direction.

Tying

Use the UFORMSN user subroutine.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available. Output of stress and strain in global coordinate directions.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 37. See Element 37 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux due to dissipation of plastic work specified with type 101.

Element 240

Axisymmetric Triangular Ring with Strain Smoothing

Element type 240 is a 3 + 1-node triangular element written for axisymmetric applications and uses piecewise bilinear interpolation functions. An enhanced integration scheme is used to enforce strain smoothing over neighboring elements. This allows the element to be used for nearly incompressible behavior and it also improves the bending capability compared to the conventional element type 2.

The stiffness matrix of this element is formed using a three point integration scheme. The mass matrix (which is always lumped) of this element is formed using a four point integration scheme.

Quick Reference

Type 240

Axisymmetric 3 + 1-node triangular ring element with strain smoothing.

Connectivity

Node numbering of the first three nodes must be counterclockwise (see Figure 3-338). The fourth node is located in the element centroid. If the fourth node is not provided by the user in the element connectivity, it will be automatically added by Marc.

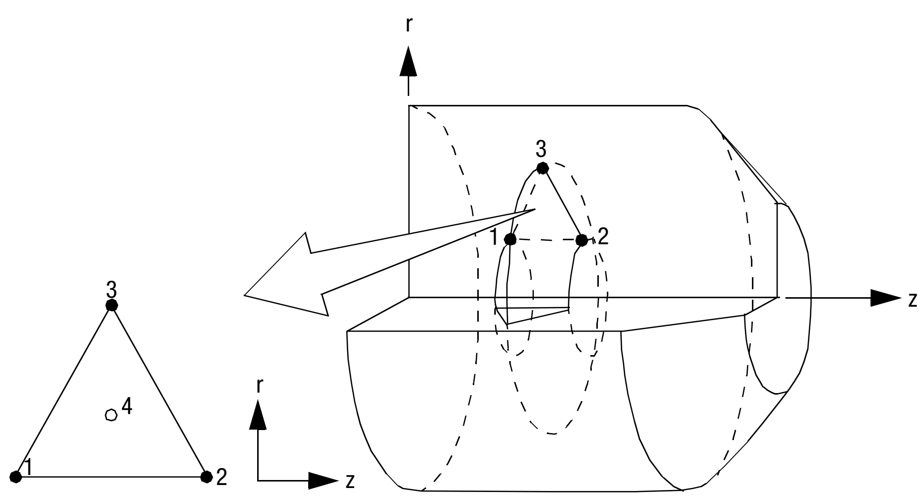


Figure 3-338 Triangular Ring Element 240: Element Connectivity

Geometry

Not required for this element.

Coordinates

1 = z
2 = r

Degrees of Freedom

1 = u = axial (parallel to symmetry axis)
2 = v = radial (normal to symmetry axis)

Tractions

Load types for distributed loads are as follows:

Load Type	Description
0	Uniform pressure distributed on 1-2 face of the element.
1	Uniform body force per unit volume in first coordinate direction.
2	Uniform body force per unit volume in second coordinate direction.
3	Nonuniform pressure on 1-2 face of the element; magnitude supplied through the FORCEM user subroutine.
4	Nonuniform body force per unit volume in first coordinate direction; magnitude supplied through the FORCEM user subroutine.
5	Nonuniform body force per unit volume in second coordinate direction; magnitude supplied through the FORCEM user subroutine.
6	Uniform pressure on 2-3 face of the element.
7	Nonuniform pressure on 2-3 face of the element; magnitude supplied through the FORCEM user subroutine.
8	Uniform pressure on 3-1 face of the element.
9	Nonuniform pressure on 3-1 face of the element; magnitude supplied through the FORCEM user subroutine.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

All pressures are positive when directed into the element. In addition, point loads can be applied at the nodes.

Concentrated loads applied at the nodes must be the value of the load integrated around the circumference.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at three integration points (see Figure 3-339), is as follows:

$1 = \varepsilon_{zz}$
$2 = \varepsilon_{rr}$
$3 = \varepsilon_{\theta\theta}$
$4 = \gamma_{zr}$

Output of Stresses

Output of stresses is also at the three integration point element and follows the same scheme as Output of Strains .

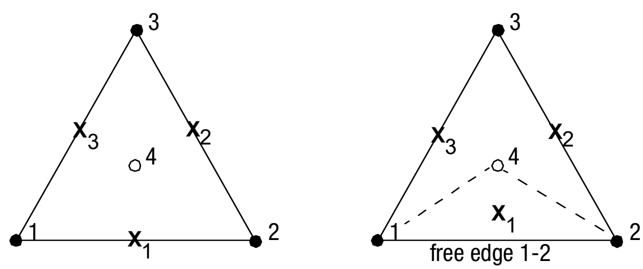


Figure 3-339 Triangular Ring Element 240: Integration Points for Strain and Stress Output

Note:	The integration points are located at the middle of the edges of the element for edges shared by another element of type 240 with the same properties. They are located at the centroid of the corresponding sub-triangle for edges not shared by another element of type 240 with the same properties (called a *free edge* in Figure 3-339). Since the strain smoothing procedure depends on the neighboring elements, the use of symmetry boundary conditions may introduce slightly different results compared to using a full finite element mesh without the symmetry conditions.

Transformation

Two global degrees of freedom can be transformed to local coordinates. In this case, the corresponding applied nodal loads should also be in the local direction.

Tying

Can be tied to axisymmetric shell type 1 by typing type 23.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available. Output of stress and strain in global coordinate directions.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 38. See Element 38 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux due to dissipation of plastic work specified with type 101.

Element 241

Three-dimensional Four-node Tetrahedron with Strain Smoothing

Element 241 is a 4+1-node three-dimensional tetrahedron (see Figure 3-340). This element uses piecewise trilinear interpolation functions. An enhanced integration scheme is used to enforce strain smoothing over neighboring elements. This allows the element to be used for nearly incompressible behavior and it also improves the bending capability compared to the conventional element type 134.

The stiffness matrix of this element is formed using a four point integration scheme. The mass matrix (which is always lumped) of this element is formed using a five point integration scheme.

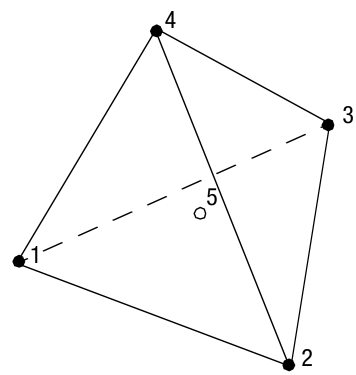


Figure 3-340 Tetrahedron Element 241: Element Connectivity

Quick Reference

Type 241

Three-dimensional 4 + 1-node tetrahedron with strain smoothing.

Connectivity

The convention for the ordering of the connectivity array is as follows. Nodes 1, 2 and 3 are the corners of the first face, given in counterclockwise order when viewed from inside the element. Node 4 is on the opposing vertex (see Figure 3-340). The fifth node is located in the element centroid. If the fifth node is not provided by the user in the element connectivity, it will be automatically added by Marc.

Geometry

Not required.

Coordinates

Three global coordinates in the x-, y-, and z-directions.

Degrees of Freedom

Three global degrees of freedom, u, v, and w.

Distributed Loads

Distributed loads chosen by value of IBODY are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3 face.
1	Nonuniform pressure on 1-2-3 face; magnitude supplied through the FORCEM user subroutine.
2	Uniform pressure on 1-2-4 face.
3	Nonuniform pressure on 1-2-4 face; magnitude supplied through the FORCEM user subroutine.
4	Uniform pressure on 2-3-4 face.
5	Nonuniform pressure on 2-3-4 face; magnitude supplied through the FORCEM user subroutine.
6	Uniform pressure on 1-3-4 face.
7	Nonuniform pressure on 1-3-4 face; magnitude supplied through the FORCEM user subroutine.
8	Uniform body force per unit volume in x direction.
9	Nonuniform body force per unit volume in x direction; magnitude supplied through the FORCEM user subroutine.
10	Uniform body force per unit volume in y direction.
11	Nonuniform body force per unit volume in y direction; magnitude supplied through the FORCEM user subroutine.
12	Uniform body force per unit volume in z direction.
13	Nonuniform body force per unit volume in z direction; magnitude supplied through the FORCEM user subroutine.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

The FORCEM user subroutine is called once per integration point when flagged. The magnitude of load defined by DIST LOADS is ignored and the FORCEM value is used instead.

For nonuniform body forces, force values must be provided for the four integration points.

For nonuniform surface pressure, force values need only be supplied for the one integration point on the face of application.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

Output of strains at three integration points (see Figure 3-341), is as follows:

1 = ε_{xx}
2 = ε_{yy}
3 = ε_{zz}
4 = γ_{xy}
5 = γ_{yz}
6 = γ_{zx}

Output of Stresses

Output of stresses is also at the four element integration points and follows the same scheme as Output of Strains .

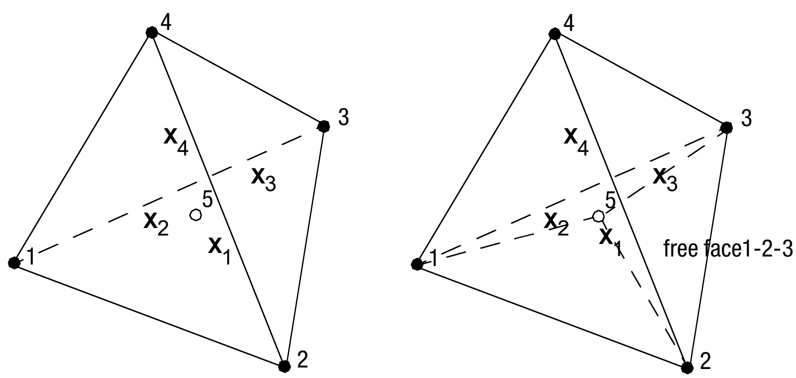


Figure 3-341 Tetrahedron Element 241: Integration Points for Strain and Stress Output

Note:	The integration points are located in the centroid of the faces of the element for faces shared by another element of type 241 with the same properties. They are located at the centroid of the corresponding sub-tetrahedron for faces not shared by another element of type 241 with the same properties (called a *free face* in Figure 3-341). Since the strain smoothing procedure depends on the neighboring elements, the use of symmetry boundary conditions may introduce slightly different results compared to using a full finite element mesh without the symmetry conditions.

Transformation

Three global degrees of freedom can be transformed to local degrees of freedom.

Tying

Use the UFORMSN user subroutine.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available – stress and strain output in global coordinate directions.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 135. See Element 135 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux due to dissipation of plastic work specified with type 101.

Element 242

Three-dimensional, 20 node Piezoelectric Arbitrary Distorted Brick

Element type 242 is a twenty-node, isoparametric, arbitrary hexahedral written for piezo-electric applications. The mechanical part of this element is based on element type 21. This element uses triquadratic interpolation functions to represent the coordinates displacements and electric potential; hence, the strains and electrical fields have a linear variations. The electrical part of this element is added at the fourth degree of freedom. This element can be used in static, modal, transient, harmonic, and buckling analysis. A description of the piezoelectric capabilities is included in Marc Volume A: Theory and User Information.

The stiffness of this element is formed using 27-point Gaussian integration.

This element can only be used with elastic constitutive relations. The electrical properties and the coupling between mechanical and electric behavior can be applied with the PIEZOELECTRIC option. The PIEZO parameter must be included.

Quick Reference

Type 242

Three-dimensional, twenty-node, second-order, isoparametric piezo-electric element (arbitrarily distorted piezoelectric brick).

Connectivity

Twenty nodes per element. Node numbering must follow the scheme below (see Figure 3-342).

Geometry

In general, not required.

Coordinates

Three coordinates in the global x-, y-, and z-directions.

Degrees of Freedom

At each corner:

1 = u (Displacement in the global x-direction)
2 = v (Displacement in the global y-direction)
3 = w (Displacement in the global z-direction)

4 = V (Electric Potential)

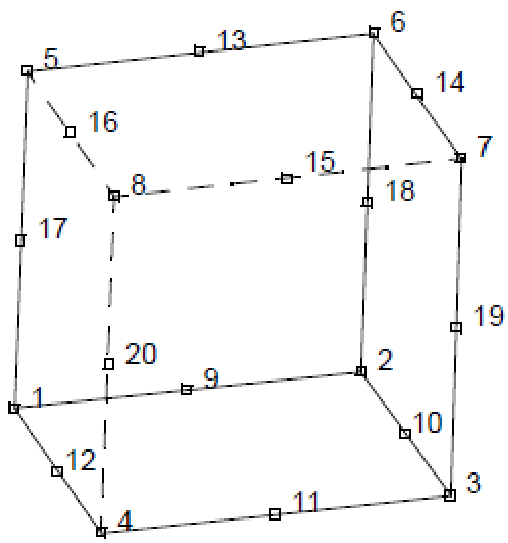


Figure 3-342 Form of element 242

Distributed Loads

Distributed loads chosen by value of `IBODY` are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3-4 face.
1	Nonuniform pressure on 1-2-3-4 face; magnitude supplied through FORCEM user subroutine.
2	Uniform body force per unit volume in -z-direction.
3	Nonuniform body force per unit volume (e.g., centrifugal force); magnitude and direction supplied through the FORCEM user subroutine.
4	Uniform pressure on 6-5-8-7 face.
5	Nonuniform pressure on 6-5-8-7 face (FORCEM user subroutine).
6	Uniform pressure on 2-1-5-6 face.
7	Nonuniform pressure on 2-1-5-6 face (FORCEM user subroutine).
8	Uniform pressure on 3-2-6-7 face.
9	Nonuniform pressure on 3-2-6-7 face (FORCEM user subroutine).
10	Uniform pressure on 4-3-7-8 face.
11	Nonuniform pressure on 4-3-7-8 face (FORCEM user subroutine).

Load Type	Description
12	Uniform pressure on 1-4-8-5 face.
13	Nonuniform pressure on 1-4-8-5 face (FORCEM user subroutine).
20	Uniform pressure on 1-2-3-4 face.
21	Nonuniform load on 1-2-3-4 face; magnitude and direction supplied in the FORCEM user subroutine.
22	Uniform body force per unit volume in -z-direction.
23	Nonuniform body force per unit volume (e.g., centrifugal force; magnitude and direction supplied through the FORCEM user subroutine).
24	Uniform pressure on 6-5-8-7 face.
25	Nonuniform load on 6-5-8-7 face; magnitude and direction supplied in the FORCEM user subroutine.
26	Uniform pressure on 2-1-5-6 face.
27	Nonuniform load on 2-1-5-6 face; magnitude and direction supplied in the FORCEM user subroutine.
28	Uniform pressure on 3-2-6-7 face.
29	Nonuniform load on 3-2-6-7 face; magnitude and direction supplied in the FORCEM user subroutine.
30	Uniform pressure on 4-3-7-8 face.
31	Nonuniform load on 4-3-7-8 face; magnitude and direction supplied in the FORCEM user subroutine.
32	Uniform pressure on 1-4-8-5 face.
33	Nonuniform load on 1-4-8-5 face; magnitude and direction supplied in the FORCEM user subroutine.
40	Uniform shear 1-2-3-4 face in 1-2 direction.
41	Nonuniform shear 1-2-3-4 face in 1-2 direction.
42	Uniform shear 1-2-3-4 face in 2-3 direction.
43	Nonuniform shear 1-2-3-4 face in 2-3 direction.
48	Uniform shear 6-5-8-7 face in 5-6 direction.
49	Nonuniform shear 6-5-8-7 face in 5-6 direction.
50	Uniform shear 6-5-8-7 face in 6-7 direction.
51	Nonuniform shear 6-5-8-7 face in 6-7 direction.
52	Uniform shear 2-1-5-6 face in 1-2 direction.
53	Nonuniform shear 2-1-5-6 face in 1-2 direction.
54	Uniform shear 2-1-5-6 face in 1-5 direction.
55	Nonuniform shear 2-1-5-6 face in 1-5 direction.
56	Uniform shear 3-2-6-7 face in 2-3 direction.
57	Nonuniform shear 3-2-6-7 face in 2-3 direction.
58	Uniform shear 3-2-6-7 face in 2-6 direction.

Load Type	Description
59	Nonuniform shear 2-3-6-7 face in 2-6 direction.
60	Uniform shear 4-3-7-8 face in 3-4 direction.
61	Nonuniform shear 4-3-7-8 face in 3-4 direction.
62	Uniform shear 4-3-7-8 face in 3-7 direction.
63	Nonuniform shear 4-3-7-8 face in 3-7 direction.
64	Uniform shear 1-4-8-5 face in 4-1 direction.
65	Nonuniform shear 1-4-8-5 face in 4-1 direction.
66	Uniform shear 1-4-8-5 face in 1-5 direction.
67	Nonuniform shear 1-4-8-5 face in 1-5 direction.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

All pressures are positive when directed into the element. In addition, point loads can be applied at the nodes.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Distributed Charges

Charge types for distributed charges are as follows:

Charge Type	Description
70	Uniform current on 1-2-3-4 face.
71	Nonuniform charge on 1-2-3-4 face.
74	Uniform charge on 5-6-7-8 face.
75	Nonuniform charge on 5-6-7-8 face.

Charge Type	Description
76	Uniform charge on 1-2-6-5 face.
77	Nonuniform charge on 1-2-6-5 face.
78	Uniform charge on 2-3-7-6 face.
79	Nonuniform charge on 2-3-7-6 face
80	Uniform charge on 3-4-8-7 face.
81	Nonuniform charge on 3-4-8-7 face.
82	Uniform charge on 1-4-8-5 face.
83	Nonuniform charge on 1-4-8-5 face.

For all nonuniform distributed charges, the magnitude is supplied through the FLUX user subroutine. All distributed charges are positive when adding charge to the element. In addition, point charges can be applied at the nodes.

Output of Strains

Output of stains at the centroid of the element in global coordinates is:

1 = ε_{xx}
2 = ε_{yy}
3 = ε_{zz}
4 = γ_{xy}
5 = γ_{yz}
6 = γ_{xz}

Output of Stresses

Same as for Output of Strains .

Output of Electric Properties

Three components of:

Electric field vector	E
Electric displacement vector	D

Transformation

Standard transformation of three global degrees of freedom to local degrees of freedom.

Tying

No special tying available.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Coupled Analysis

In a coupled thermal-piezoelectric analysis, the associated heat transfer element is type 44. See Element 44 for a description of the conventions used for entering the flux and film data for this element.

Element 243

Three-dimensional, 10 node Piezoelectric Tetrahedron

Element type 243 is a ten-node, isoparametric, arbitrary tetrahedron for piezo-electric applications. The mechanical part of this element is based on element type 127. Each edge forms a parabola such that four nodes define the corners of the element and six more nodes define the position of the midpoint of each edge (see Figure 3-343). This allows for an accurate representation of the strain field and the electric potential field in piezo-electric analyses. The electrical part of this element is added at the fourth degree of freedom. This element can be used in static, modal, transient, harmonic, and buckling analysis. A description of the piezoelectric capabilities is included in Marc Volume A: Theory and User Information.

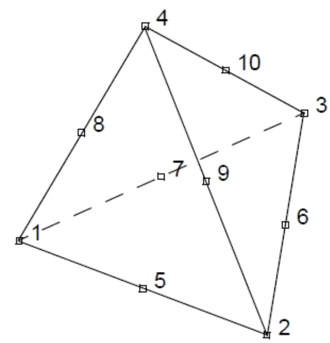


Figure 3-343 Form of Element 243

This element can only be used with elastic constitutive relations for the structural part. The electrical properties and the coupling between mechanical and electric behavior can be applied with the PIEZOELECTRIC option. The PIEZO parameter must be included.

Geometry

The geometry of the element is interpolated from the Cartesian coordinates of ten nodes.

Connectivity

The convention for the ordering of the connectivity array is as follows:

> Nodes 1, 2, 3, 4 are the corners of first face, given in a counterclockwise order when viewed from inside the element. Node 4 is the opposite vertex. Nodes 5, 6, and 7 are on the first face between nodes 1 and 2, 2 and 3, 3 and 1, respectively. Nodes 8, 9 and 10 are along the edges between the first face and node 4, between nodes 1 and 4, 2 and 4, 3 and 4, respectively.

Note that in most cases, the elements will be generated automatically via a pre-processor (such as Mentat), which will follow the node numbering scheme mentioned above.

Integration

The element is integrated numerically using four points (Gaussian quadrature). The first plane of such points is closest to 1, 2, 3 face of the element with the first node closest to the first node of the element (shown in Figure 3-344).

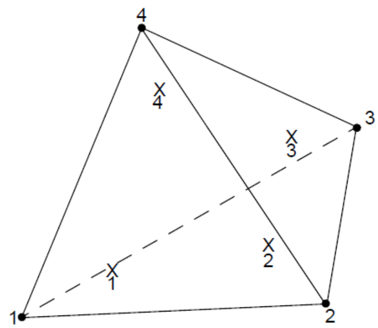


Figure 3-344 Element 243 Integration Plane

Quick Reference

Type 243

Three-dimensional, ten-nodes, second-oder, isoparametric piezo-electric tetrahedron.

Connectivity

Ten nodes numbered as described in the connectivity write-up for this element and as shown in Figure 3-343.

Geometry

Not required.

Coordinates

Three global coordinates in the x-, y-, and z-directions.

Degrees of Freedom

At each corner:

1 = u (Displacement in the global x-direction)
2 = v (Displacement in the global y-direction)
3 = w (Displacement in the global z-direction)

4 = V (Electric potential)

Distributed Loads

Distributed loads chosen by value of `IBODY` are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3 face.
1	Nonuniform pressure on 1-2-3 face.
2	Uniform pressure on 1-2-4 face.
3	Nonuniform pressure on 1-2-4 face.
4	Uniform pressure on 2-3-4 face.
5	Nonuniform pressure on 2-3-4 face.
6	Uniform pressure on 1-3-4 face.
7	Nonuniform pressure on 1-3-4 face.
8	Uniform body force per unit volume in x-direction.
9	Nonuniform body force per unit volume in x-direction.
10	Uniform body force per unit volume in y-direction.
11	Nonuniform body force per unit volume in y-direction.
12	Uniform body force per unit volume in z-direction.
13	Nonuniform body force per unit volume in z-direction.
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

All pressures are positive when directed into the element. For all nonuniform loads, the magnitude is supplied through the FORCEM user subroutine. In addition, point loads can be applied at the nodes.

The FORCEM user subroutine is called once per integration point when flagged. The magnitude of load defined by DIST LOADS is ignored and the FORCEM value is used instead.

For nonuniform body force, force values must be provided for the four integration points. For nonuniform surface pressure, force values need only be supplied for the three integration points on the face of application.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Distributed Charges

Charge types for e are listed below.

Charge Type	Description
70	Uniform charge on 1-2-3 face.
71	Nonuniform charge on 1-2-3 face.
72	Uniform charge on 1-2-4 face.
73	Nonuniform charge on 1-2-4 face.
74	Uniform charge on 2-3-4 face.
75	Nonuniform charge on 2-3-4 face.
76	Uniform charge on 1-3-4 face.
77	Nonuniform charge on 1-3-4 face

For all nonuniform distributed charges, the magnitude is supplied through the FLUX user subroutine. All distributed charges are positive when adding charge to the element. In addition, point charges can be applied at the nodes.

Output of Strains

Output of stains at the centroid of the element in global coordinates is:

$1 = \varepsilon_{xx}$
$2 = \varepsilon_{yy}$
$3 = \varepsilon_{zz}$
$4 = \gamma_{xy}$
$5 = \gamma_{yz}$
$6 = \gamma_{xz}$

Output of Stresses

Same as for Output of Strains .

Output of Electric Properties

Three components of:

Electric field vector	E
Electric displacement vector	D

Transformation

Standard transformation of three global degrees of freedom to local degrees of freedom.

Tying

Use UFORMSN user subroutine.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is not available.

Coupled Analysis

In a coupled thermal-piezoelectric analysis, the associated heat transfer element is type 133. See Element 133 for a description of the conventions used for entering the flux and film data for this element.

Element 244

2-D Planar Spring/Dashpot Element

This is a 2-D planar spring/dashpot (also called CSPRING) structural element that may exhibit nonlinear and frequency dependent behavior. The nonlinear material and geometric properties for this element are defined through the PSPRING model definition option. This element cannot be used with CONTACT.

Note:	This element is used as the substitute for SPRINGS option.

Geometric basis: None

Quick Reference

Type 244

2-D planar spring/dashpot element.

Connectivity

Two nodes.

Geometric properties

The geometric properties of the spring element are defined on the PSPRING option. They are used to define the type of spring which can be one of the following:

- the connecting DOF between the nodes
- to link a DOF of the first node to the ground
- the true direction to link the nodes

The geometric properties are also used to define the physical parameters of the spring, like stiffness in the stress pass or other value for different physical pass (thermal and electrical pass).

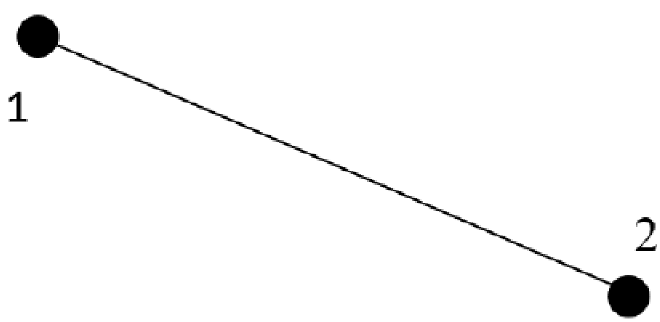


Figure 3-345 2-D Planar Spring Element

Coordinates

First three coordinates - (x,y) global Cartesian coordinates.

Degree of Freedom:.

1=	chosen degree of freedom of the first node
2=	chosen degree of freedom of the second node

Tractions

The element does not support any form of distributed loading. Point loads and moments can be applied at the nodes.

Output of Strains

Generalized strain components are as follows:

$$U_2 - U_1$$

The strain component is available in the post file through post codes 1, 121, 301 or 401.

Output of Section Forces and Stresses

Section forces and stresses are output as:

Local connected force. Local connected stress has the same value to force.

For the post file, the spring section forces utilizes post code 264 (Axial Force) for beams. The stress components are available in the post file through post codes 11, 41, 311, or 341. Real harmonic stress components are available in the post file through post codes 51 or 351. Imaginary harmonic stress components are available in the post file through post codes 61 or 361.

Transformation

Displacements and rotations can be transformed to local directions.

Output Points

Centroid section of the element.

Large Displacement Formulation

No stress stiffness matrix is calculated for large displacements.

Field Analysis

In a heat transfer or coupled thermal-mechanical analysis, the thermal conductance matrix is calculated based on the thermal data provided on the PSPRING option. Similarly, for a Joule heating analysis, the electrical resistance matrix is calculated based on the provided electrical data on the PSPRING option. No distributed fluxes or films are supported.

Element 245

Axisymmetric Spring/Dashpot Element

This is an axisymmetric spring/dashpot (also called CSPRING) structural element that may exhibit nonlinear and frequency dependent behavior. The nonlinear material and geometric properties for this element are defined through the PSPRING model definition option. This element cannot be used with CONTACT.

Note:	This element is used as the substitute for SPRINGS option.

Geometric basis: None

Quick Reference

Type 245

2-D axisymmetric spring/dashpot element.

Connectivity

Two nodes.

Geometric properties

The geometric properties of the spring element are defined on the PSPRING option. They are used to define the type of spring which can be one of the following:

- the connecting DOF between the nodes
- to link a DOF of the first node to the ground
- the true direction to link the nodes

The geometric properties are also used to define the physical parameters of the spring, like stiffness in the stress pass or other value for different physical pass (thermal and electrical pass).

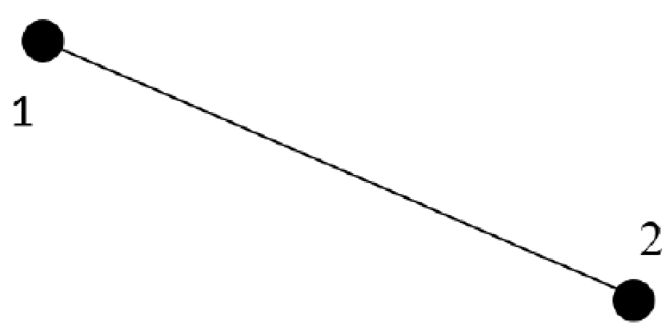


Figure 3-346 Axisymmetric Spring Element

Coordinates

First three coordinates - (x,y) global Cartesian coordinates.

Degree of Freedom:.

1=	chosen degree of freedom of the first node
2=	chosen degree of freedom of the second node

Tractions

The element does not support any form of distributed loading. Point loads and moments can be applied at the nodes.

Output of Strains

Generalized strain components are as follows:

$$U_2 - U_1$$

The strain component is available in the post file through post codes 1, 121, 301 or 401.

Output of Section Forces and Stresses

Section forces and stresses are output as:

Local connected force. Local connected stress has the same value to force.

For the post file, the spring section forces utilizes post code 264 (Axial Force) for beams. The stress components are available in the post file through post codes 11, 41, 311, or 341. Real harmonic stress components are available in the post file through post codes 51 or 351. Imaginary harmonic stress components are available in the post file through post codes 61 or 361.

Transformation

Displacements and rotations can be transformed to local directions.

Output Points

Centroid section of the element.

Large Displacement Formulation

No stress stiffness matrix is calculated for large displacements.

Field Analysis

In a heat transfer or coupled thermal-mechanical analysis, the thermal conductance matrix is calculated based on the thermal data provided on the PSPRING option. Similarly, for a Joule heating analysis, the electrical resistance matrix is calculated based on the provided electrical data on the PSPRING option. No distributed fluxes or films are supported.

Element 246

3-D Spring/Dashpot Element

This is a 3-D spring/dashpot (also called CSPRING) element that may exhibit nonlinear or frequency dependent behavior. The nominal physical parameters and geometric property values for this element are defined through the PSPRING option. This element cannot be used with CONTACT.

Note:	This element is used as the substitute for SPRINGS option.

Geometric basis: None

Quick Reference

Type 246

3-D spring/dashpot element.

Connectivity

Two nodes.

Geometric properties

The geometric properties of the spring element are defined on the PSPRING option. They are used to define the type of spring which can be one of the following:

- the connecting DOF between the nodes
- to link a DOF of the first node to the ground
- the true direction to link the nodes

The geometric properties are also used to define the physical parameters of the spring, like stiffness in the stress pass or other value for different physical pass (thermal and electrical pass).

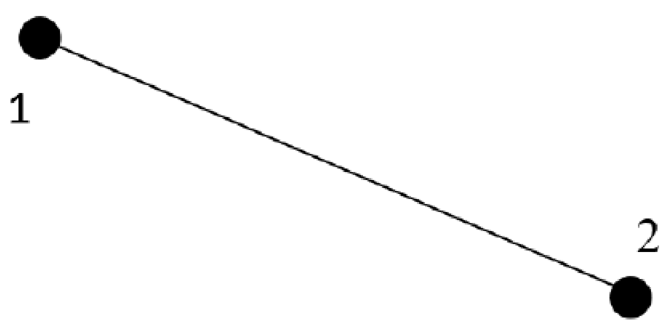


Figure 3-347 3-D Planar Spring Element

Coordinates

First three coordinates - (x,y,z) global Cartesian coordinates.

Degree of Freedom:.

1=	chosen degree of freedom of the first node
2=	chosen degree of freedom of the second node

Tractions

The element does not support any form of distributed loading. Point loads and moments can be applied at the nodes.

Output of Strains

Generalized strain components are as follows:

$$U_2 - U_1$$

The strain component is available in the post file through post codes 1, 121, 301 or 401.

Output of Section Forces and Stresses

Section forces and stresses are output as:

Local connected force. Local connected stress has the same value to force.

For the post file, the spring section forces utilizes post code 264 (Axial Force) for beams. The stress components are available in the post file through post codes 11, 41, 311, or 341. Real harmonic stress components are available in the post file through post codes 51 or 351. Imaginary harmonic stress components are available in the post file through post codes 61 or 361.

Transformation

Displacements and rotations can be transformed to local directions.

Output Points

Centroid section of the element.

Large Displacement Formulation

No stress stiffness matrix is calculated for large displacements.

Field Analysis

In a heat transfer or coupled thermal-mechanical analysis, the thermal conductance matrix is calculated based on the thermal data provided on the PSPRING option. Similarly, for a Joule heating analysis, the electrical resistance matrix is calculated based on the provided electrical data on the PSPRING option. No distributed fluxes or films are supported.

Element 247

Three-dimensional, Low-order, Tetrahedron, Herrmann Formulation

This element is a three-dimensional, isoparametric, 4-node, low-order tetrahedron with an additional pressure degree of freedom at each of the four nodes (see Figure 3-348). It is written for incompressible or nearly incompressible three-dimensional applications. The displacements, pressure and the coordinates of the element are linearly distributed over the element volume. The stiffness of this element is formed using a single Gauss integration point, with hourglass stabilization for the pressure degree of freedom.

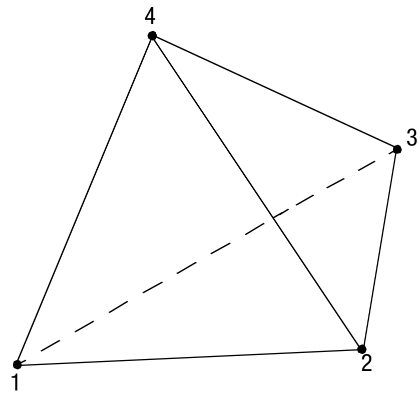


Figure 3-348 Form of Element 247

This element can be used for incompressible elasticity via the total Lagrange formulation, or for rubber elasticity and elasto-plasticity via the updated Lagrange multiplicative (F^eF^p) formulation, or for elasto-plasticity via the updated Lagrange additive formulation. To activate a large strain analysis, use the LARGE STRAIN parameter (see Marc Volume A: Theory and User Information and Marc Volume C: Program Input).

Integration

The stiffness matrix is evaluated using a single Gauss point. For the mass matrix and volumetric loads, four integration points are used.

Quick Reference

Type 247

4-node, isoparametric, three-dimensional, tetrahedron using the Herrmann formulation. Written for incompressible or nearly incompressible applications.

Connectivity

Four nodes per element (see Figure 3-348). Node numbering is the same as for element type 134; that is, nodes 1, 2, 3, being the corners of the first face in counterclockwise order when viewed from inside the element, and node 4 on the opposing vertex.

Coordinates

Three global coordinates in the x-, y- and z-directions.

Degrees of Freedom

1	= u	
2	= v	
3	= w	
4	$= \sigma_{kk}/E$	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	= -p	= negative pressure (for Mooney, Ogden, or Soil)
	= p / K	(for Herrmann elements using additive decomposition; K here is the effective bulk modulus)

$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1E_3}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_1 + \frac{\frac{\nu_{12}\nu_{23}}{E_1E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\Delta_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_2 + \frac{\frac{\nu_{31}\nu_{23}}{E_3E_2}}{\frac{\nu_{12}\nu_{31}}{E_1E_3} + \frac{\nu_{12}\nu_{23}}{E_1E_2} + \frac{\nu_{31}\nu_{23}}{E_3E_2}}\sigma_3 \right] \frac{9}{E_1 + E_2 + E_3}$$

Distributed Loads

Distributed loads chosen by value of `IBODY` are as follows:

Load Type	Description
0	Uniform pressure on 1-2-3 face.
1	Nonuniform pressure on 1-2-3 face.
2	Uniform pressure on 1-2-4 face.
3	Nonuniform pressure on 1-2-4 face.
4	Uniform pressure on 2-3-4 face.
5	Nonuniform pressure on 2-3-4 face.
6	Uniform pressure on 1-3-4 face.
7	Nonuniform pressure on 1-3-4 face.
8	Uniform body force per unit volume in x direction.
9	Nonuniform body force per unit volume in x direction.
10	Uniform body force per unit volume in y direction.
11	Nonuniform body force per unit volume in y direction.
12	Uniform body force per unit volume in z direction.
13	Nonuniform body force per unit volume in z direction.

Load Type	Description
100	Centrifugal load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
102	Gravity loading in global direction. Enter three magnitudes of gravity acceleration in respectively global x, y, z direction.
103	Centrifugal and Coriolis load, first value specifies the square of the angular velocity in radians/time and the second value specifies the angular acceleration in radians/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
104	Centrifugal load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.
105	Centrifugal and Coriolis load, first value specifies the angular velocity in cycles/time) and the second value specifies the angular acceleration in cycles/time^2. Rotation axis is specified in the ROTATION A option. Angular acceleration must be zero for axisymmetric elements.

For other types of distributed loads that are normally applicable for all types of elements, please refer to Distributed Loads in Chapter 1 of this manual.

Output of Strains

1	= global xx strain	
2	= global yy strain	
3	= global zz strain	
4	= global xy strain	
5	= global yz strain	
6	= global zx strain	
7	= σ_{kk}/E	= mean pressure variable (isotropic)
	= formula below (orthotropic)	
	= -p	= negative hydrostatic pressure (for Mooney, Ogden, or Soil)

$$H = \left[\frac{\frac{\nu_{12}\nu_{31}}{E_1 E_3}}{\frac{\nu_{12}\nu_{31}}{E_1 E_3} + \frac{\nu_{12}\nu_{23}}{E_1 E_2} + \frac{\nu_{31}\nu_{23}}{E_3 E_2}} \sigma_1 + \frac{\frac{\nu_{12}\nu_{23}}{E_1 E_2}}{\frac{\nu_{12}\nu_{31}}{E_1 E_3} + \frac{\nu_{12}\Delta_{23}}{E_1 E_2} + \frac{\nu_{31}\nu_{23}}{E_3 E_2}} \sigma_2 + \frac{\frac{\nu_{31}\nu_{23}}{E_3 E_2}}{\frac{\nu_{12}\nu_{31}}{E_1 E_3} + \frac{\nu_{12}\nu_{23}}{E_1 E_2} + \frac{\nu_{31}\nu_{23}}{E_3 E_2}} \sigma_3 \right] \frac{9}{E_1 + E_2 + E_3}$$

Output of Stresses

Similar to strains for the components 1 to 6. Component 7 should be around zero for a well-converged solution.

Transformation

Any local set (u, v, w) can be used at any node.

Updated Lagrange Procedure and Finite Strain Plasticity

Capability is available.

Coupled Analysis

In a coupled thermal-mechanical analysis, the associated heat transfer element is type 135. See Element 135 for a description of the conventions used for entering the flux and film data for this element. Volumetric flux due to dissipation of plastic work has to be specified with type 101.