

ATB Models (SOL 700)

ATBACC	Defines an acceleration field that will be applied to ATB segments.
ATBJNT	This entry can only be used together with the ATBSEG entries that this joint connects. The ATBSEG entries overwrite the position and orientation of the ATB segments as specified in the ATB input file. The ATBJNT entry can be used to create a Bulk Data file containing elements and grid points to visualize the ATB segment together with its joints. This visualization of the joints makes it possible to position the ATB model in any available preprocessor.
ATBSEG	Defines the position and orientation of the ATB segments.
RELEX	Defines a rigid ellipsoid whose properties and motion are defined by either ATB.

Contact (SOL 700)

BCBODY	Defines a flexible or rigid contact body in 2D or 3D.
BCBODY1	Defines a flexible or rigid contact body in 2D or 3D.
BCBOX	Defines a 3D contact region - all elements within the region define a contact body.
BCELIPS	Defines a list of ellipsoid names (character strings) for use of contact analysis.
BCGRID (SOL 700 only)	Grids to be included in contact analyses.
BCMATL	Defines a 3D contact region by element material. All elements with the specified materials define a contact body.
BCNECT	Defines the touching and touched contact bodies used for general contact.
BCONPRG-700	Defines geometric contact parameters of touching bodies.
BCONPPR-700	Defines physical contact parameters of touching bodies.
BCSEG	Grids which are part of an element to be used in contact analyses.
BCTABLE	Defines a contact table (old format).
BCTABL1	Defines a contact table (new format).
BSURF	Defines a contact body or surface by Element IDs.

Coordinate Systems (SOL 700)

CORD3R	Defines a moving rectangular coordinate system using three points (SOL 700 only).
------------------------	---

Dampers/Springs (SOL 700)

CDAMP1D	Defines a scalar damper connection for use in SOL 700 only.
CDAMP2D	Defines a scalar damper connection for use in SOL 700 only.
CELAS1D	Defines a scalar spring connection for use in SOL 700 only.
CELAS2D	Defines a scalar spring connection for use in SOL 700 only.

Direct Text Input (SOL 700)

ENDDYNA	Defines the end of direct text to Dytran.
TODYNA	Defines the start of direct text to Dytran.

Element Properties (SOL 700)

PBELT	Defines the properties of a belt element referenced by a CROD entry.
PELAS1	Defines a spring property designated by a force-deflection curve for SOL 700.
PSHELL1	Defines the properties of SOL 700 shell elements that are much more complicated than the shell elements defined using the PSHELL entry.
PVISC1	Defines the properties of a nonlinear damper where the damping constant varies with the velocity.

Eulerian Boundary (SOL 700)

BARRIER	Defines a barrier for transport in a Eulerian mesh.
FLOW	Defines the properties of a material for the boundaries of an Eulerian mesh.
FLOWC	Defines the properties of a material for the boundaries of a Eulerian mesh.
FLOWDEF	Definition of default Eulerian flow boundary condition.
FLOWT	Defines the material properties for the in- or outflow of material trough the boundary of an Euler mesh.
HYDSTAT	Initializes the Euler element densities in accordance to a hydrostatic pressure profile.

Eulerian Initial Conditions (SOL 700)

CYLINDR	Cylindrical shape used in the initial condition definition on the TICEUL entry.
SPHERE	Spherical shape used in the initial condition definition on the TICEUL entry.
SURFINI	Defines a surface that is used for initialization of regions of an Eulerian mesh.
TICEL	Defines the initial values of element variables at the beginning of the analysis.
TICEUL1	Defines the initial values sets for Eulerian regions. The Eulerian regions are defined by geometric shapes.
TICREG	Defines the initial values sets for Eulerian regions. The Eulerian regions are defined by geometric shapes.
TICVAL	Defines the initial values of an Eulerian geometric region.

Eulerian Materials (SOL 700)

EOSDEF	EOSDEF defines the properties of the deflagration equation of state, and the reaction rate to model the burning of solid propellants. The burning of the solid propellant produces hot gas.
EOSGAM	Defines the properties of a Gamma Law equation of state where the pressure p is defined.
EOSIG	Defines the properties of Ignition and Growth equation of state and the reaction rate equation used to model high explosives.
EOSJWL	Defines the properties of a JWL equation of state commonly used to calculate the pressure of the detonation products of high explosives.
EOSMG	Defines the properties of a Mie-Gruneisen equation of state commonly used to calculate the pressure p in high strain rate processes.
EOSNA	Defines the properties of Noble-Abel equation of state.
EOSPOL	Defines the properties of a polynomial equation of state where the pressure p is defined.
FAILJC	Defines the properties of the Johnson-Cook failure model.
FAILMPS	Defines the properties of a failure model where failure occurs when the equivalent plastic strain exceeds the specified value.
MATDEUL	Defines a complete constitutive model as a combination of an equation of state, a shear model, a yield model, a failure model, a spall model (PMIN), and corotational frame.
PMINC	Defines a spallation model where the minimum pressure is constant.
SHREL	Defines an elastic shear model with a constant shear modulus.
SHRPOL	Defines an elastic shear model with a polynomial shear modulus.
YLDHY	Defines a yield model with zero yield stress.
YLDJC	Defines a Johnson-Cook yield model where the yield stress is a function of effective plastic strain, strain rate, and temperature.
YLDMSS	Defines the yield model for snow material. This entry must be used in combination with MATDEUL, EOSPOL and SHREL.
YLDPOL	Defines a polynomial yield model where the yield stress is a function of effective plastic strain.
YLDRPL	Defines a rate power law yield model where the yield stress is a function of effective plastic strain and strain rate.
YLDSG	Defines the Steinberg-Guinan yield model where the yield stress is a function of effective plastic strain, pressure and temperature.
YLDTM	Defines the Tanimura-Mimura yield model where the yield stress is a function of effective plastic strain, strain rate and temperature.

YLDVM Defines a bilinear or piecewise-linear yield model with isotropic hardening, using the von Mises yield criterion.

YLDZA Defines the Zerilli-Armstrong yield model where the yield stress is a function of effective plastic strain, strain rate and temperature.

Eulerian Solid Elements (SOL 700)

PEULER Defines the properties of Eulerian element.

PEULER1 Eulerian element properties.

Euler/Lagrange Coupling (SOL 700)

ABINFL Defines an inflator model suited for airbag analyses.

COUCOHF Defines a cohesive friction model suited for Euler Coupled analyses. The friction model is defined as part of the coupling surface.

COHFRIC Allows friction and sticking during tensile conditions at the coupling surface.

COUOPT Defines the interaction factor and a pressure load from the covered side acting on a BSURF.

COUP1FL Defines the surrounding variables when a segment of a coupling surface fails.

COUPINT Defines the surrounding variables when a segment of a coupling surface fails.

COUPLE Defines a coupling surface that acts as the interface between an Eulerian (finite volume) and a Lagrangian (finite element) domain.

INFLCG Defines the cold gas-inflator characteristics of a COUPLE and/or GBAG subsurface.

INFLFRC Defines the gas fractions as a function of time for hybrid inflators.

INFLGAS Defines a thermically ideal gas to be used with a standard or hybrid inflator.

INFLHB Defines the hybrid-inflator characteristics of a COUPLE and/or GBAG subsurface.

INFLTNK Defines the Tanktest-inflator characteristics of a COUPLE and/or GBAG subsurface.

INFLTR Defines the inflator characteristics of a COUPLE and/or GBAG subsurface.

INITGAS Specifies the initial gas composition inside a gasbag or Euler coupling surface.

LEAKAGE Defines the porosity model to be used with GBAG or COUPLE.

PERMEAB Defines the permeability of a COUPLE and/or GBAG (sub)surface.

PERMGBG Defines a permeable area of a COUPLE and/or GBAG surface, connected to another GBAG.

PORFCPL Defines an interaction between two coupling surfaces through a hole.

PORFLOW Defines the material properties for the in- or outflow of an Eulerian mesh through a porous area of the couple surface.

PORFLWT	Defines a time dependent flow trough a porous area of the couple surface.
PORFGBG	Defines a hole in a couple and/or GBAG (sub)surface, connected to another GBAG.
PORHOLE	Defines a hole in a COUPLE and/or GBAG surface.
PORHYDS	Prescribes a hydrostatic pressure profile on a porous BSURF.

Hourglass Control (SOL 700)

HGSUPPR	Defines the hourglass suppression method and the corresponding hourglass damping coefficients.
---------	--

Initial Conditions (SOL 700)

TIC3	Allows for the definition of a velocity field of grid points consisting of a rotation and a translation specification. Used in Explicit Nonlinear (SOL 700) only.
------	---

Materials (SOL 700)

MAT1	Defines the material properties for linear isotropic materials.
MAT2	Defines the material properties for linear anisotropic materials for two-dimensional elements.
MAT8	Defines the material property for an orthotropic material for isoparametric shell elements.
MATBV	Defines the bulk viscosity for materials.
MATEP	Elasto-plastic material properties for SOL 700 only.
MATF	Specifies failure model properties.
MATFAB	Defines the properties of a bi-directional woven fabric material for shell elements.
MATHE	Specifies hyperelastic (rubber-like) material properties for nonlinear analysis.
MATORT	Specifies elastic orthotropic material properties for 3-dimensional and plane strain and shell behavior for linear and nonlinear analyses.
MATRIG	Defines the properties of a rigid body.
MATVE	Specifies isotropic visco-elastic material properties to be used for quasi-static or dynamic analysis

Miscellaneous (SOL 700)

BIAS	Specifies a variation of the mesh-size in one direction for use in the MESH entry.
CMARKB2	Defines a 2-noded marker beam element by means of connecting two grid points.
CMARKN1	Defines a 1-noded marker element on a grid point.
DETSPH	Defines the ignition point from which a spherical detonation wave travels, causing the reaction of high explosive materials.

DYFSISW	Allows activating or deactivating Fluid Structure Interaction and Eulerian solver.
EULFOR	Defines a body force loading (acceleration) on Euler elements per unit mass.
EULFOR1	Alternative way to define an acceleration within a geometric region of the Euler model regions are defined by geometric shapes which are defined by EULFREG entries.
EULFREG	Defines the acceleration field for sets of Eulerian regions, The Eulerian regions are defined by geometric shapes. For each coordinate direction a time-depended acceleration can be defined.
FFCONTR	Defines the pressure within a closed volume. Intended for the use in (partially) filled containers, where dynamic fluid effects are negligible, e.g. top loading and hot filling.
MESH	Defines a mesh.
PMARKER	Defines the behavior of the marker element in the FV domain.

Multi-Variable Parameters (SOL 700)

DYPARAM, ATBAOUT	Defines the frequency at which output is written to the main output file of ATB.
DYPARAM, ATBHOUT	A time-history file is created containing the output as requested in the ATB input file on cards H.1 to H.11.
DYPARAM, ATBTOUT	Defines the frequency at which output is written to the time-history files of ATB.
DYPARAM,AUTOCOUP	Defines the automatic coupling algorithm.
DYPARAM,AXIALSYM	Axial symmetric analysis.
DYPARAM,AXREMAP	Allows import of a 2D axial symmetric Euler archive into a 3D simulation.
DYPARAM,BULKL	Defines the default value of the linear bulk viscosity coefficient.
DYPARAM,BULKQ	Defines the default value of the quadratic bulk viscosity coefficient.
DYPARAM,BULKTYP	Defines the default type of bulk viscosity.
DYPARAM,CFULLRIG	Converts all 123456 constraints to the FULLRIG option on all entries.
DYPARAM,CLUFLIM	This DYPARAM activates a limiter that scales down the volume strain rate for clumps with a small average uncovered fraction. It can keep an instable airbag run stable, just like PARAM, VELMAX can keep runs stable.
DYPARAM,CLUMPENR	Sets the definition of the kinetic energy calculation method for Eulerian blended clumps.
DYPARAM,COHESION	Cohesion for Coulomb Friction

DYPARAM,CONM2OUT	Determines if a summary of concentrated masses and their energy and momentum is written to the output file.
DYPARAM,CONTACT	Defines certain defaults for the contact definitions.
DYPARAM,COSUBMXT	Defines the maximum number of subcycles that can occur in Euler/Lagrange coupling. During a subcycle, the geometry of the coupling surface is not updated. This number can vary in time and is given by a table.
DYPARAM,COUFRIC	Defines the Coulomb friction scheme.
DYPARAM,ELDLTH	Print initial time step sizes for elements in the first cycle.
DYPARAM,EULERCB	Divides a Euler domain into several cubes.
DYPARAM,EULTRAN	Sets the definition of the face velocity used in the transport scheme of the Multi-material solver and the single material strength solver.
DYPARAM,EUSUBCYC	Controls the growth of the subcycling interval in Euler computations.
DYPARAM,EUSUBMAX	Defines the maximum number of subcycles that can occur in the Euler solver. During a subcycle, the Euler computations are skipped.
DYPARAM,FAILDT	Defines the property of a failure model where element failure occurs when the element's time step falls below the specified limit.
DYPARAM,FAILOUT	Defines whether failed elements are written to the output file (ARCHIVES).
DYPARAM,FASTCOUP	Defines the fast coupling algorithm.
DYPARAM,FLOWMETH	Defines the method for simulating material flow between two Euler domains across open areas in coupling surfaces.
DYPARAM,GEOCHECK	This parameter forces a check of the geometry for consistent connectivity of the defined hexagonal elements and correction if needed.
DYPARAM,HGCMEM	Defines the default membrane damping coefficient for shell elements.
DYPARAM,HGCOEFF	Defines the global default hourglass damping coefficient.
DYPARAM,HGCSOL	Define the default damping coefficient for solid elements.
DYPARAM,HGCTWS	Defines the default twisting damping coefficient for shell elements.
DYPARAM,HGCWRP	Defines the default warping damping coefficient for shell elements.
DYPARAM,HGSHELL	Defines the default hourglass suppression method for shell elements.

DYPARAM,HGSOLID	Defines the default hourglass suppression method for solid elements.
DYPARAM,HGTYPE	Defines the default type of hourglass suppression method.
DYPARAM,HICGRAV	Defines the value of the gravity to be used by the HIC calculations.
DYPARAM,HVLFAIL	Defines element failure on the hydrodynamic volume limit
DYPARAM,HYDROBOD	Defines a body force for single hydro material in Euler.
DYPARAM,IMM	The option allows to specify the IMM method to be used.
DYPARAM,INFOBJ	Additional information about the BJOIN and spotweld connectivity will be listed in the output file.
DYPARAM,INISTEP	Defines the time step used at the start of the analysis.
DYPARAM,JWLDET	Specifies whether the blast wave of one explosive can ignite another explosive. Here it assumed that the explosives are modeled by a combination of EOSJWL and DETSPH entries.
DYPARAM,LIMCUB	Defines the maximum number of cubes used to sort the grid points in a contact definition.
DYPARAM,LIMITER	Defines the type and the spatial accuracy of scheme used in the Euler solver based on the ideas of Prof. Philip Roe.
DYPARAM,MATRMERG	Merges MATRIG and/or RBE2-FULLRIG rigid bodies into a new FULLRIG assembly.
DYPARAM,MATRMRG1	Merges MATRIG and/or RBE2-FULLRIG rigid bodies into one existing MATRIG or RBE2-FULLRIG assembly with predefined properties.
DYPARAM,MAXSTEP	Defines the maximum allowable time step.
DYPARAM,MESHPLN	Defines Mesh density for covering rigid planes.
DYPARAM,MINSTEP	Defines the minimum time step that causes the analysis to terminate.
DYPARAM,MIXGAS	Specifies whether the gas constants of the Euler material or of gas bags are updated based on the gas composition and temperature.
DYPARAM,NZEROVEL	Set the velocity of a node to zero in case all attached elements have failed.
DYPARAM,OLDLAGT	Activate the collapsed hexahedron scheme as default for lagrangian CTETRA elements.
DYPARAM,PARALLEL	The option allows you to gather information on the parallel section.
DYPARAM,PLCOVCUT	Defines time when PLCOVER is cut off.
DYPARAM,PMINFAIL	Defines Lagrangian solid element failure on reaching the spall limit.

DYPARAM,RBE2INFO	The grid points attached to MATRIG and RBE2 assemblies are listed to the output file.
DYPARAM,RHOCUT	Defines the minimum density for all Eulerian elements.
DYPARAM,RJSTIFF	Defines the stiffness of a rigid joint.
DYPARAM,SHELLFRM	Sets the default for the shell formulation for quadrilateral elements.
DYPARAM,SHELMYSYS	Defines the shell element system for the BLT shells.
DYPARAM,SHPLAST	Specifies the type of calculation used to determine the plane stress plasticity method for shells.
DYPARAM,SHSTRDEF	Specifies the default coordinate system for the stress and strain output of composite shells.
DYPARAM,SHTHICK	Specifies whether or not the thickness of the shell changes with membrane straining.
DYPARAM,SLELM	Defines whether shell sublayer variables are to be stored in the element arrays.
DYPARAM,SMP,BATCHSIZ	Define batch size and number of CPU loops, per entity type.
DYPARAM,SMP,CPUINFO	Define the CPU information per entity type.
DYPARAM,SNDLIM	Defines the minimum value for the speed of sound.
DYPARAM,SPHERSYM	Enables an efficient and accurate 1D spherical symmetric solution for Eulerian materials
DYPARAM,STRNOUT	Saves the total strains and equivalent effective stress (von Mises stress) at shell sublayers for output.
DYPARAM,TOLCHK	To check the direction of an Euler face, the face normal vector is projected onto the closest coordinate direction. If this projection is 1, the normal is exactly in the coordinate direction. When this projection is within a sufficient small tolerance of 1, the face can be handled by fast coupling. The tolerance used is TOLCHK. If the projection is smaller than 1-TOLCHK, the face cannot be handled by fast coupling and the analysis terminates. Options are then to slightly increase TOLCHK, write out double precision format in PATRAN, use general coupling, or use the MESH entry. Increasing TOLCHK too much can make the coupling surface computation less accurate. To keep the computation accurate, the maximal allowed value of TOLCHK is 1e-6. If DYPARAM,TOLCHK is not used, the tolerance used is 1e-14.
DYPARAM,VDAMP	Controls the global damping in the dynamic relaxation.
DYPARAM,VELMAX	Defines the maximum velocity in Eulerian meshes.

DYPARAM,VELMAX1	Defines the maximum translational and angular velocity in Eulerian and Lagrangian meshes
DYPARAM,VISCPLAS	Activate the overstress formula to update strain-rate dependent plasticity. This formula is normally used for viscous-plastic material.

Rigid Elements (SOL 700)

BJOIN	Defines (multiple) pairs of grid points of one-dimensional and/or shell elements to be joined during the analysis.
RBJOINT	Defines a joint between two rigid bodies.
WALL	Defines a rigid plane through which specified Lagrangian grid points cannot penetrate. Used in Explicit Nonlinear (SOL 700) only.

Time Step Control (SOL 700)

TSTEPNL	Defines parametric controls and data for nonlinear transient structural or heat transfer analysis.
---------	--

User Defined Subroutines (SOL 700)

COMPUDS	Defines an orthotropic failure model for shell composites specified by a user subroutine.
EOSUDS	Defines the frequency dependent properties for an isotropic poroelastic material.
FAILUDS	User defined simple failure of Eulerian materials.
FLOWUDS	Defines a flow boundary on an Eulerian mesh specified by a user subroutine.
FORCUDS	Defines enforced motion at grid points specified by a user subroutine.
NLOUTUD	User defined output requests for elements or Lagrangian grid points.
PORUDS	Defines a porosity model of a COUPLE surface through a user-written subroutine.
SHRUDS	Specifies that a user subroutine is being used to define the shear modulus.
TABLUDS	Specifies that a user routine is being used to define an arbitrary function.
TICEUDS	User defined simple failure of Eulerian materials.
YLDUDS	Specifies that a user subroutine is being used to define a simple yield model.

Solution Control

Aerodynamic Matrix Generation

- MKAERO1** Provides a table of Mach numbers (m) and reduced frequencies (k) for aerodynamic matrix calculation.
- MKAERO2** Provides a list of Mach numbers (m) and reduced frequencies (k) for aerodynamic matrix calculation.

Aerodynamic Parameters

- AERO** Gives basic aerodynamic parameters for unsteady aerodynamics.
- AEROS** Defines basic parameters for static aeroelasticity.

Aeroelastic Response Analysis

- AEDW** Defines a downwash vector associated with a particular control vector of the associated aerodynamic configuration (AECONFIG).
- AEFORCE** Defines a vector of absolute or “per unit dynamic pressure” forces associated with a particular control vector.
- AEPARM** Defines a general aerodynamic trim variable degree-of-freedom (aerodynamic extra point).
- AEPRESS** Defines a vector of pressure/unit dynamic pressure associated with a particular control vector.
- AESCALE** Defines reference lengths to scale aerodynamic grid points.
- GUST** Defines a stationary vertical gust for use in aeroelastic response analysis.
- TABRNDG** Defines the power spectral density (PSD) of a gust for aeroelastic response analysis.
- TRIM** Specifies constraints for aeroelastic trim variables. The SPLINE1 and SPLINE4 entries need to be here for the finite plate spline.
- UXVEC** Specification of a vector of aerodynamic control point values.

Aeroelastic Stability Analysis

- DIVERG** Defines Mach numbers (m) for a static aeroelastic divergence analysis.
- FLFACT** Used to specify density ratios, Mach numbers, reduced frequencies, and velocities for flutter analysis.
- FLUTTER** Defines data needed to perform flutter analysis.

Buckling Analysis

- EIGB** Defines data needed to perform buckling analysis.
- EIGRL** Defines data needed to perform real eigenvalue (vibration or buckling) analysis with the Lanczos method.

Cyclic Symmetry

CYSYM Defines parameters for cyclic symmetry analysis.

Eigenvalue Analysis

EIGC Defines data needed to perform complex eigenvalue analysis.

EIGR Defines data needed to perform real eigenvalue analysis.

EIGRL Defines data needed to perform real eigenvalue (vibration or buckling) analysis with the Lanczos method.

EIGP Defines poles that are used in complex eigenvalue extraction by the Determinant method.

RVDOF,
RVDOF1 Degrees-of-freedom specification for residual vector calculation.

Monte-Carlo simulation

MONCARL Defines the parameters for Monte-Carlo simulations.

Fatigue Analysis

DTI,UNITS Defines units necessary for conversion during the analysis for the Nastran/ADAMS interface and Nastran fatigue analysis.

FTGDEF Defines elements and their associated fatigue properties to be considered for fatigue analysis.

FTGPARM Defines parameters for a fatigue analysis

FTGSEQ Defines the loading sequence for pseudo-static fatigue analysis using SOL 101 or modal transient fatigue analysis using SOL 103 or SOL 112 or random vibration fatigue analysis using SOL 108 or SOL 111.

FTGEVNT Groups simultaneously applied loads into loading events for pseudo-static fatigue analysis using SOL 101 or modal transient fatigue analysis using SOL 103 or random vibration fatigue analysis using SOL 108 or SOL 111 by referencing FTGLOAD entries.

FTGLOAD Defines cyclic loading variation for pseudo-static fatigue analysis using SOL 101 or modal transient fatigue analysis using SOL 103 or random vibration fatigue analysis using SOL 108 or SOL 111.

MATFTG Defines fatigue material properties.

PFTG Defines fatigue elemental properties.

RANDPS Defines PSDs and cross PSDs for frequency domain fatigue (random vibration) analysis using SOL 108 or SOL 111.

TABLFTG Defines tabular data for specifying fatigue cyclic loading variation for time domain fatigue analysis using SOL 101 or SOL 103.

TABLRPC	Tabular functions for generating dynamic loads by reading the tabular data from an external channel data file.
TABRND1	Defines tabular data for specifying PSD random vibration loading for frequency domain fatigue (random vibration) analysis using SOL 108 or SOL 111.
TIM2PSD	Defines parameters and triggers conversion of time history channel data into corresponding Power Spectral Density (PSD) functions for use in random vibration fatigue analysis using SOL 108 or SOL 111.
UDNAME	Provides the name of a file that can be referenced from other bulk data entries such as FTGLOAD.

Frequency Response

FREQ	Defines a set of frequencies to be used in the solution of frequency response problems.
FREQ1	Defines a set of frequencies for problem solution.
TABDMP1	Defines modal damping as a tabular function of natural frequency.

Nonlinear Static Analysis

ITER	Defines options for the iterative solver in SOLs 101, 106, 108, 111, 153, 200 and 400.
NLADAPT	Defines additional parameters for automatic load or time stepping used with enhanced nonlinear in SOL 400.
NLPARM	Defines a set of parameters for nonlinear static analysis iteration strategy.
NLPCI	Defines a set of parameters for the arc-length incremental solution strategies in nonlinear static analysis (SOL 106).
NLSTEP	Describes the Control Parameters for Mechanical, Thermal and Coupled Analysis in SOL 400 and for Contact Analysis in SOL 101.

Optimization (SOL 200 Only)

BEADVAR	Defines design region for topography (bead or stamp) optimization.
BNDGRID	Specifies a list of grid point identification numbers on design boundaries or surfaces for shape optimization (SOL 200).
DCONADD	Defines the design constraints for a subcase as a union of DCONSTR entries.
DCONSTR	Defines design constraints.
DDVAL	Define real, discrete design variable values for discrete variable optimization.
DEQATN	Defines a design variable for design optimization.
DESVAR	Defines a design variable for design optimization.
DLINK	Relates one design variable to one or more other design variables.
DOPTPRM	Overrides default values of parameters used in design optimization.

DRESP1	Defines a set of structural responses that is used in the design either as constraints or as an objective.
DRESP2	Defines equation responses that are used in the design, either as constraints or as an objective.
DRESP3	Defines an external response using user-supplied routine.
DSCREEN	Defines screening data for constraint deletion.
DTABLE	Defines a table of real constants that are used in equations (see DEQATN entry).
DTABLE2	Defines real constants from a field of property, material or connections bulk data entries which can then be invoked by a DVxREL2, DRESP2, or DRESP3 entry.
DVBSHAP	Associates a design variable identification number to a linear combination of boundary shape vectors from a particular auxiliary model.
DVCREL1	Defines the relation between a connectivity property and design variables.
DVCREL2	Defines the relation between a connectivity property and design variables with a user-supplied equation.
DVGRID	Defines the relationship between design variables and grid point locations.
DVLREL1	Defines the linear relation between analysis model loading and design variables
DVMREL1	Defines the relation between a material property and design variables.
DVMREL2	Defines the relation between a material property and design variables with a user-supplied equation.
DVPREL1	Defines the relation between an analysis model property and design variables.
DVPREL2	Defines the relation between an analysis model property and design variables with a user-supplied equation.
DVPSURF	Design Variable to Control Surface Setting Relation
DVSHAP	Defines a shape basis vector by relating a design variable identification number (DVID) to columns of a displacement matrix.
MODTRAK	Specifies parameters for mode tracking in design optimization (SOL 200).
SEDLINK	Multiple Design Variable Linking Across PART SE Boundary
SEDRSP2	Design Sensitivity Equation Response Quantities for PART SE
SEDRSP3	Defines External Response with User-Supplied Routines
STOCHAS	Randomization of Model Parameters
TOMVAR	Defines a design region for topometry optimization (element-by-element optimization).
TOPVAR	Defines a topology design region for topology optimization.

p-element and p-Adaptivity Analysis

- ADAPT** Defines controls for p-version adaptive analysis.
- PSET** Describes polynomial order distribution and is selected by the ADAPT Case Control command.
- PVAL** Describes polynomial order distribution and is selected by the ADAPT Bulk Data entry.

Random Response

- RANDPS** Defines load set power spectral density factors for use in random analysis.
- RANDT1** Defines time lag constants for use in random analysis autocorrelation function calculation.
- RCROSS** Cross-power spectral density and cross-correlation function output.
- TABRND1** Defines power spectral density as a tabular function of frequency for use in random analysis. Referenced by the RANDPS entry.

Rotordynamics

- RGYRO** Specifies synchronous or asynchronous analysis, reference rotor, and rotation speed of the reference rotor.
- ROTHYBD** Hybrid damping for rotors
- ROTOR** Rotor Model Definition
- ROTORAX** Axisymmetric Model Rotor Definition
- ROTORG** Specifies grids that compose the rotor line model.
- ROTORSE** Specifies grids that compose the rotor line model.
- RSPINR** Specifies the relative spin rates between rotors for complex eigenvalue, frequency response, and static analysis.
- RSPINT** Specifies rotor spin rates for nonlinear transient analysis.
- UNBALNC** Specifies an unbalanced load for transient analysis in terms of a cylindrical system with the rotor rotation axis as the z-axis.

Transient Response

- TIC** Defines values for the initial conditions of variables used in structural transient analysis.
- TSTEP** Defines time step intervals at which a solution will be generated and output in transient analysis.
- TSTEPNL** Defines parametric controls and data for nonlinear transient structural or heat transfer analysis. is intended for SOLs 129, 159, and 99.

Entries A - B

\$	Comment
----	---------

Used to insert comments into the input file. Comment statements may appear anywhere within the input file.

Format:

\$ followed by any characters out to column 80.

Example:

```
$ TEST FIXTURE-THIRD MODE
```

Remarks:

1. Comments are ignored by the program.
2. Comments will appear only in the unsorted echo of the Bulk Data.
3. For free field entries (See Format of Bulk Data Entries above), a \$ can be used to terminate the field entries and any required remaining fields will be modulo replicated as blank.
4. For the default NASTRAN SYSTEM (IFPSTAR) = YES, input using FIXED field, a \$ can be used to terminate the field entries and any required remaining fields will be modulo replicated as blank.
The OLD IFP (NASTRAN SYSTEM (IFPSTAR) = NO) will fail with this option.

/

Control Input Stream

Allows the user control over field width, and on restart entry deletion, insert and update, and process control.
(This entry's execution is dependent on whether NASTRAN SYSTEM(444)= 0 or 1.)

Format:

1	2	3	4	5	6	7	8	9	10
/	Alpha numeric text as described below								

Examples:

/ 0 12

/delete all

/ Delete FORCE1,PARAM MATVP

Descriptor	Meaning
/	Must be first character and occur somewhere in fields one through 8. (Character)
<i>If NASTRAN SYSTEM(444)=0, then the n1, n2 directive below is the only valid form:</i>	
n1,n2	Two integers indicating following number of lines of previous input are to be deleted. (Integer, n1 ≥ 10, n2 > n1). /, n1,n2 directive must be entered before any new data is added to the input stream.
<i>If NASTRAN SYSTEM(444)=1, then the above n1, n2 directive will not work and only the following forms are valid:</i>	
DELETE	Keyword indicating following listed items are to be deleted. (Character) /DELETE directive must be entered before any new data is added to the input stream.
ALL	Keyword indicating following a /DELETE directive. All previous input items will be deleted on RESTART. (Character)
<i>List of names</i>	A list of Bulk Data names to be deleted following a /DELETE directive. All previous input Bulk entries with the names indicated will be deleted on RESTART. (Character)
SFW	A Keyword that is used for fixed field entries and allows the user to define the field width of single field entries for Field 2 thru Field 9. See Remark 1. (Character)
n	Single Field Width value n used with SFW. This is used for fixed field entries and allows the user to define the field width of single field entries for Field 2 thru Field 9 to the value n.(Integer 8 ≤ n ≤ 14, Default = 8). If double field is selected by use of the * option, then Field 2 thru Field 9 are of 2n width. (Field 1 and Field 10 always remain of width 8). See Remark 1.

Descriptor	Meaning
SKIPON	Input data following this keyword would not be processed in the current execution. (Character). This command is terminated by one of three methods: 1) The occurrence of a SKIPOFF. 2) The occurrence of a BEGIN (Such as BEGIN SUPER=10) 3) The occurrence of an ENDDATA.
SKIPOFF	Keyword turning off the SKIPON command. (Character)
INSERT	Keyword that implies that entries when placed into the input stream must not be previously present. See Remark 1. (Character; Default for INITIAL runs.)
UPDATE	Keyword that implies that entries when placed into the input stream are new or are to replace existing entries. See Remark 1. (Character; Default for RESTART runs.)
DUPMETH	Keyword that allows reversion to the original IFP, DUPTOL tolerance selection. /DUPMETH=OLD selects original IFP tolerance selection. See the DPBLKTOL(402) system cell description and Remark 1. (Character; Default NEW for IFPSTAR tolerance selection).
DUPTOL	Keyword that allows a closeness tolerance value when searching for removal of duplicate entries. (Character)
v	Tolerance value, used with DUPTOL, when searching for removal of duplicate entries. See Remark 1. (Real ≥ 0 ; Default 1.0-5.)
LISTON	Keyword to indicate a "list of data to follow" See Remark 10. (Character)
LISTOFF	Keyword to indicate a "list of data is terminated" See Remark 10. (Character)

Remarks:

1. If the SFW, INSERT, UPDATE, DUPMETH, DUPTOL Keywords and any associated value (n, v) occur in the first BEGIN BULK Section the values set will carry forward to any other BEGIN section included. If a user wishes to have some later BEGIN have different control values, then the BEGIN BULK values can be overridden for that particular BEGIN Section by the inclusion of / entries particular to that specific BEGIN Section. The DUPMETH must come before the DUPTOL entry. Currently IFPSTAR will not recognize the settings from the command line or RC file due to default usage issues at the application state and real value passing through the component level. When DUPMETH is OLD and DPBLKTOL or DUPTOL is zero or positive, remove grids with duplicated id without checking CP, CD, PS ,SE fields and distance tolerance.

Example:

```
/DUPMETH=OLD
/DUPTOL=0.1
```

With the following NASTRAN entry required:
NASTRAN SYSTEM(402)=0.1

2. The input stream into Nastran is now part of a Common Data Model component and entries are committed to a data base as soon as they are interpreted in a single unsorted pass. When the user requests a sorted ECHO, the ECHO represents a binary value converted back into a column character location. It does not represent the actual binary entry value! Unsorted ECHO respects the request list of the ECHO command.
3. If an improperly formatted bulk data entry is encountered, the true file line number of the error is printed along with reference to the Parent entry. If the offending entry is in an include file the file is identified.
4. Usage of the continuation mnemonic is no longer needed and its usage is not recommended.
5. Default Fixed Fields are 80 characters (see SFW). Free field supports 128 characters.
6. Because the Common Data Model component entries are committed to a data base as soon as they are interpreted in a single unsorted pass, the /DELETE syntax is now required to be used on restart.
7. If an /INSERT is present on RESTART the Common Data Model component will signal an error condition EXCEPT when certain objects such as GRID entries have been classified as Remove Duplicate or when objects such as SPC that allow superposition are encountered.
8. Entries that allow superposition are always in /INSERT.
9. If NASTRAN SYSTEM(444)=1, then the Nastran Replication feature is not recommended.
10. For convenience in entering an ordered list of elements (tuples) data or, such as table value inputs, the following is allowed for example:

tabled1,111

/liston

1.0 1.0

2.0 1.0

3.0 2.0

4.0 2.0

5.0 1.0

6.0 1.0

/listoff

,endt

or for example if user likes commas:

tabled1,112

/liston

1.0, 1.0

2.0, 1.0

3.0, 2.0

4.0, 2.0

5.0, 1.0
6.0, 1.0
/listoff
,endt

ABINFL

Inflator Model to be Used With GBAG or COUPLE Entries

Defines an inflator model suited for airbag analyses. The inflator model is defined as part of the GBAG or COUPLE surface. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
ABINFL	CID	INFID	SUBID	INFTYPE	INFTYPID	COEFF	COEFFV		

Example:

ABINFL	201	1	120	INFLHB	11		0.012		
--------	-----	---	-----	--------	----	--	-------	--	--

Descriptor	Meaning
CID	Unique number of a ABINFL entry. (Integer > 0; Required)
INFID	Number of a set of ABINFL entries NFID must be referenced from a GBAG or COUPLE entry. (Integer > 0; Required)
SUBID	Number of a BSURF, BCBOX, BCPROP, BCMATL or BCSEG, which must be a part of the as defined on the GBAG or COUPLE entry. (Integer > 0; Required)
INFTYPE	Defines the type of inflator. (Character; Required)
	INFLTR The INFLTR logic is used to model inflators in an air bag.
	INFLHB The INFLHB logic is used to model hybrid inflators in an air bag.
	INFLCG The INFLCG logic models a cold gas inflator.
	INFLTNK The INFLTNK logic models the inflator properties (mass flow rate and inflator gas temperature) calculated from the empirical results.
INFTYPID	Number of the entry selected under INFTYPE, for example, INFLTR,INFTYPID. (Integer > 0; Required)
COEFF	Method of defining the area coefficient. (Character, CONSTANT)
	CONSTANT The area coefficient is constant and specified on COEFFV.
	TABLE The area coefficient varies with time. COEFFV is the number of a TABLED1 entry giving the variation with time.
COEFFV	The area coefficient or the number of a TABLED1 entry depending on the COEFF entry. (0.0 < Real < 1.0 or 1 > 0)

Remarks:

1. The INFLTR or INFLHB inflator geometry and location is defined by a BSURF, BCBOX, BCPROP, BCMATL or BCSEG. The area of the hole through which the gas enters is equal to the area of the subsurface multiplied by COEFFV. A value of COEFFV = 1.0 will open up the complete subsurface area, while a value of COEFFV = 0.0 will result in a closed inflator area with no inflow.

2. This allows for setting up the exact same model for either a uniform pressure model or an Euler Coupled model. This makes it possible to set up the model using the switch from full gas dynamics to uniform pressure (GBAGCOU).

ABSNMVB

Meta data group Boolean name/value pairs

Associate Boolean Name/Value pairs to meta data collectors (ABSTRACT)

Format:

1	2	3	4	5	6	7	8	9	10
ABSNMVB	ABSTID		NAME1	VAL1	NAME2	VAL2	-etc.-	...	

Example:

1	2	3	4	5	6	7	8	9	10
ABSNMVB	50		STIFFNER	TRUE	VARSECT	FALSE			

Descriptor	Meaning
ABSTID	Unique identification number for meta data group (ABSTRACT). (Integer>0).
NAMEi	Up to eight characters defining the name of this Name/Value pair. (Character).
VALi	Value of this Name/Value pair -Boolean- data (Character) TRUE or FALSE.

Remarks:

1. This entry has no impact on the solution.
2. The only allowed character values are “TRUE” or “FALSE”.

ABSNMVI

Meta data group Integer name/value pairs

Associate Integer Name/Value pairs to meta data collectors (ABSTRACT).

Format:

1	2	3	4	5	6	7	8	9	10
ABSNMVI	ABSTID		NAME1	VAL1	NAME2	VAL2	-etc.-		

Example:

1	2	3	4	5	6	7	8	9	10
ABSNMVI	50		SPANS	12					

Descriptor	Meaning
ABSTID	Unique identification number for meta data group (ABSTRACT). (Integer>0).
NAMEi	Up to eight characters defining the name of this Name/Value pair. (Character).
VALi	Value of this Name/Value pair (Integer).

Remarks:

1. This entry has no impact on the solution.

ABSNMVR

Meta data group Real Name/Value pairs

Associate Real Name/Value pairs to meta data collectors (ABSTRACT).

Format:

1	2	3	4	5	6	7	8	9	10
ABSNMVR	ABSTID		NAME1	VAL1	NAME2	VAL2	-etc.-		

Example:

1	2	3	4	5	6	7	8	9	10
ABSNMVR	50		ANGLEA	15.0	ANGLEB	45.0			

Descriptor	Meaning
ABSTID	Unique identification number for meta data group (ABSTRACT). (Integer>0)
NAMEi	Up to eight characters defining the name of this Name/Value pair. (Character)
VALi	Value of this Name/Value pair (Real)

Remarks:

1. This entry has no impact on the solution.

ABSNMVS

Meta data group String name/value pairs

Associate String Name/Value pairs with meta data collectors (ABSTRACT).

Format:

1	2	3	4	5	6	7	8	9	10
ABSNMVS	ABSTID								
	NAME1				VAL1				
	NAME2				VAL2				
	-etc.-								

Example:

1	2	3	4	5	6	7	8	9	10
ABSNMVS	50								
	ABSNAME	B pillar model year 2020							
	ABSDESC	B pillar including locking bushing							

Descriptor	Meaning
ABSTID	Unique identification number for meta data group (ABSTRACT). (Integer>0)
NAMEi	Up to eight characters defining the name of this Name/Value pair. (Character).
VALi	Value of this Name/Value pair. May consist of any Character A-Z, or numbers, and underscore or dash. Limited to 56 characters. (Character).

Remarks:

1. This entry has no impact on the solution.
2. The name/value pair entries may be specified in any order.

3. In the HDF5 viewer

The screenshot shows the HDF5 viewer interface with two main sections:

- INPUT** section:
 - ABSTRACT** dataset:
 - ABSNMVS** group:
 - IDENTITY: ABSTID: ID**
 - NAMEVAL: Position value: "ABSNAME", "ABSDESC"**

ABSTRACT
IDENTITY: ABSTNM: ID, NASTNM, NASTIDs
NASTIDs: NASTNM IDs

HDF5 viewer

The ABSNAME dataset structure is shown in three parts:

- NAME** and **SVAL** table (part 0):

NAME	SVAL
0 ABSNAME	RIGHT SPAN
1 ABSNAME	BEAM SHAPE 1
2 ABSNAME	LEFT SPAN
3 ABSNAME	POINT MASS IN HOLE
4 ABSDESC	THIS IS A REMOTE POINT MASS IN HO.
5 ABSNAME	REGIONS APPLICATION/REGION PON.
6 ABSNAME	FIXED BASE CONSTRAINT
7 ABSDESEL	GOLD
8 ABSNAME	FIXED
- ABSTNAME**, **ABSTID**, **NASTNAME**, **NASTIDS_POS**, **NASTIDS** table (part 0):

ABSTNAME	ABSTID	NASTNAME	NASTIDS_POS	NASTIDS
3 BEAMSPAN 3	CBEAM	50	48	
4 CONSTRAIN 7	SPC	98	1	
5 LEVENT 8	PLOAD	99	1	
6 LEVENT 10	FORCE	100	1	
7 LOAD 9	PLOAD	101	1	
8 LOAD 11	FORCE	102	1	
9 MASS 4	CONN2	103	1	
10 MASS 5	GRID	104	1	
11 MASS 6	RBE3	105	1	
- NASTID** table (part 0):

NASTID
97 2550
98 5
99 12
100 13
101 12
102 13
103 2857
104 13828
105 7589
106 13828

4. In the above Example, the following SET3 entries might include:

```
$ CBEAM elements for B pillar
SET3, 10, ELEM, 7701, THRU, 7799
$ CBUSH elements for locking
SET3, 30, ELEM, 66, 67, 68, 69
$ RIGID ELEMENTS
SET3, 90, RBEIN, 9991, THRU, 9996
```

ABSTRACT**Meta Data Collector for HDF5 output and post processing**

Allows the user to define an arbitrary group of Nastran entities (ABSTRACTion), using any Nastran keyword type including elements, grids, materials, etc. and associate a name and description for the group.

Format:

1	2	3	4	5	6	7	8	9	10
ABSTRACT	ABSTNM	ABSTID	NASTNM	NASTID1	NASTID2	NASTID3	-etc.-		

Example:

1	2	3	4	5	6	7	8	9	10
ABSTRACT	PILLAR	50	CBEAM	10					
ABSTRACT	PILLAR	50	CBUSH	30					
ABSTRACT	PILLAR	50	RBE3	90					

Descriptor	Meaning
ABSTNM	Up to eight characters for the name of the meta data group. (Character).
ABSTID	Unique identification number for the meta data group. (Integer>0).
NASTNM	Up to eight characters for the name of the Nastran entity being added to the meta data group. (Character).
NASTIDi	ID referencing a Nastran entity of the specified type or a set of such entities.

Remarks:

1. The group definition is transmitted from the input file to the .op2 and/or .h5 results file with no impact on the solution.
2. The group definition can subsequently be recovered from either the .op2 or .h5 file by a post-processor.
3. Associated with this entry are the entries: [ABSNMVS](#), [ABSNMVB](#), [ABSNMVI](#), [ABSNMVR](#).

ACCEL**Acceleration Load**

Defines static acceleration loads, which may vary over a region of the structural model. The load variation is based upon the tabular input defined on this Bulk Data entry.

Format:

1	2	3	4	5	6	7	8	9	10	
ACCEL	SID	CID	N1	N2	N3	DIR				
	LOC1	VAL1	LOC2	VAL2	Continues in Groups of 2					

Example(s):

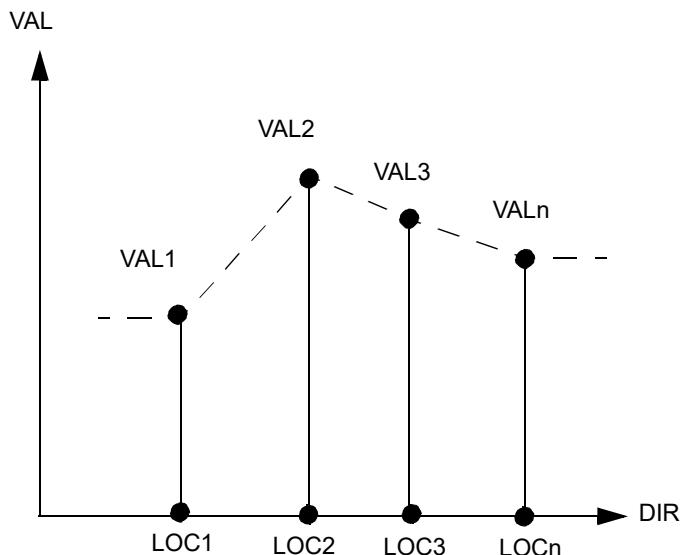
ACCEL	100	2	0.0	1.0	2.0	X			
	0.0	1.0	1000.0	3.0					

Descriptor	Meaning
SID	Load set identification number (Integer > 0)
CID	Coordinate system identification number. (Integer > 0; Default = 0)
Ni	Components of the acceleration vector measured in coordinate system CID. (Real; at least one Ni ≠ 0.0)
DIR	Component direction of acceleration variation. (Character; one of X, Y, and Z)
LOCi	Location along direction DIR in coordinate system CID for specification of a load scale factor. (Real)
VALi	The load scale factor associated with location LOCi. (Real)

Remarks:

- For all grids of the model, the acceleration vector is defined by $\ddot{\mathbf{a}} = VAL \cdot \vec{N}$, where \vec{N} is the vector defined by (N1, N2, N3). The magnitude of $\ddot{\mathbf{a}}$ is equal to VAL times the magnitude of \vec{N} . The scale factor VAL for each grid is found linearly interpolating the DIR coordinate of the grid between table values LOCi/VALi. If the GRID point coordinate in coordinate system CID is outside the range of the table, VAL is determined either from VAL1 or VALn (the last value, see the following figure).
- This type of acceleration load may be combined with other loads, such as FORCE, MOMENT, GRAV, and ACCEL1 loads, by specification on a LOAD entry. The SID on an ACCEL entry may not be the same as that of any other load entry.
- This acceleration load does not include effects due to mass on scalar points.
- A CID of zero references the basic coordinate system.
- The DIR field must contain one of the characters X, Y, or Z. The DIR direction defines the direction of acceleration load variation along direction 1, 2, or 3 respectively of coordinate system CID.

6. A minimum of two pairs of {LOC_i, VAL_i} data must be specified.



Definition of Load Scale Factor vs Location

7. If Modules are present then this entry may only be specified in the main Bulk Data section.
8. In the static solution sequences, SID must be selected by the LOAD Case Control command. In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.

ACCEL1**Acceleration Load**

Defines static acceleration loads at individual GRID points.

Format:

1	2	3	4	5	6	7	8	9	10
ACCEL1	SID	CID	A	N1	N2	N3			
	GRIDID1	GRIDID2	-etc.-						

Example(s):

ACCEL1	100	2	10.0	1.0	2.0	0.0			
	1	2	3	4	THRU	10	BY	2	
	20	21	THRU	30	40	52	69	70	
	82	90	100						

Descriptor	Meaning
SID	Load set identification number (Integer > 0)
CID	Coordinate system identification number. (Integer > 0; Default = 0)
A	Acceleration vector scale factor. (Real)
Ni	Components of the acceleration vector measured in coordinate system CID. (Real; at least one Ni ≠ 0.0)
GRIDID _i LIST	List of one or more GRID point identification numbers. Key words “THRU” and “BY” can be used to assist the listing. (Integer > 0)

Remarks:

1. The acceleration vector is defined by $\ddot{a} = A \cdot \vec{N}$, where \vec{N} is the vector defined by (N1, N2, N3). The magnitude of \ddot{a} is equal to A times the magnitude of \vec{N} .
2. This type of acceleration load may be combined with other loads, such as FORCE, MOMENT, GRAV, and ACCEL loads, by specification on a LOAD entry. The SID on an ACCEL1 entry may not be the same as that of any other load entry.
3. This acceleration load does not include effects due to mass on scalar points.
4. A CID of zero references the basic coordinate system.
5. ACCEL1 loads are not supported with super elements.
6. In the static solution sequences, SID must be selected by the LOAD Case Control command. In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.

ACCSSPT

Access Point Definition

Defines a grid interior to an external superelement whose motion will be retained in the residual structure of the assembled superelement model. This entry is used during the external superelement creation run and is valid only in external superelements.

Format:

1	2	3	4	5	6	7	8	9	10
ACCSSPT	GID1	GID2	GIDi	etc					

Example:

ACCSSPT	117	119							
---------	-----	-----	--	--	--	--	--	--	--

Descriptor	Meaning
GIDi	Identification number of a grid point designated as an access point (Integer>0)

Remarks:

1. An access point may only be defined in an external superelement.
2. Access point IDs must be unique across all external superelements.
3. Access points must be interior points in the external superelement.
4. Access points will be automatically placed in the m-set of the residual structure of the assembly and will have the same displacements in the residual structure as it does in the external superelement.
5. If the user needs the access points to be in the a-set, or if user needs to connect access points from different superelements via MPCs, the following two parameters are required:
PARAM, SEP1XOVR, 128
PARAM, AUTOMSET, YES
6. If Modules are present then this entry may only be specified in the main Bulk Data section.

ACLOAD

ACTRAN Acoustic Pressure Load Matrices for SOL 108/111

Defines ACTRAN acoustic pressure load matrices.

Format:

1	2	3	4	5	6	7	8	9	10
ACLOAD	SID	UNIT1	UNIT2	SCLR	SCLI	LSQID			

Example(s):

ACLOAD	101	41	42	1.5					
--------	-----	----	----	-----	--	--	--	--	--

Field	Contests
SID	Set identification number. See Remark 1. (Integer > 0)
UNIT1	Fortran unit number of mapped data from ACTRAN. See Remark 2. (Integer > 0)
UNIT2	Fortran unit number of property matrices from ACTRAN. See Remark 2. (Integer > 0)
SCLR	The real part of complex scale factor to be multiplied to ACTRAN matrices. (Real; Default = 1.0)
SCLI	The imaginary part of complex scale factor to be multiplied to ACTRAN matrices. (Real; Default = 0.0)
LSQID	Load SeQuence IDentification number on an ACTRAN file with multiple load cases. (Integer > 0, default=1).

Remarks:

1. Dynamic excitation sets must be selected with the Case Control command DLOAD = SID for frequency response analysis.
2. The following type of ASSIGN should be specified in the FMS section with the vacant unit number (see ASSIGN statements). The unit number cannot be selected doubly.

```
ASSIGN INPUTT2='ACTRAN_pressure.f70' UNIT=41
ASSIGN INPUTT4='ACTRAN_pressure.op4' UNIT=42
```
3. SID must be unique for all RLOAD1, RLOAD2, ACSRCE and ACLOAD entries.
4. Refer the ACTRAN manual for the details of exportation of the acoustic pressure load matrix data for Nastran.
5. The residual vectors for ACLOAD will not be computed.
6. Value of LSQID that is greater than the actual load cases in the ACTRAN file is a fatal error.

ACMODL**Fluid-Structure Interface Parameters**

Optional entry to define modeling parameters for Fluid-Structure Interface.

Format:

(If METHOD=“BW”)

1	2	3	4	5	6	7	8	9	10
ACMODL	INTER	INFOR	FSET	SSET	NORMAL	METHOD	SKNEPS	DSKNEPS	
	INTOL	ALLSSET	SRCHUNIT						

(If METHOD=“CP”)

ACMODL	INTER	INFOR	FSET	SSET	NORMAL	METHOD			
--------	-------	-------	------	------	--------	--------	--	--	--

Example(s):

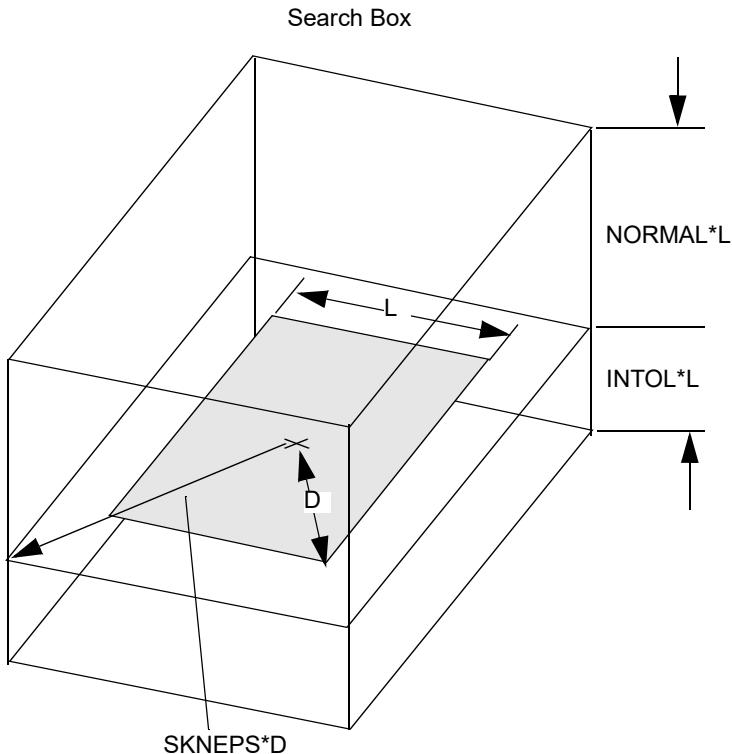
ACMODL	IDENT				NORMAL				
--------	-------	--	--	--	--------	--	--	--	--

Descriptor	Meaning
INTER	Type of structure-fluid interface. (Character = “IDENT” or “DIFF”; Default = “DIFF”)
INFOR	For METHOD = “BW” and INTER = “DIFF”, indicates if FSET and SSET are used to define the fluid-structure interface, “NONE” if not used, and whether they contain grids or elements. (Character = “GRIDS”, “ELEMENTS”, “ALL”, or “NONE”, Default = “NONE”)
FSET	For METHOD = “CP” and INTER = “DIFF”, indicates if FSET and SSET are used to define the fluid-structure interface, “NONE” if not used. See Remark 10. (Character = “ALL”, or “NONE”, Default = “NONE”)
SSET	Optional identification of a SET1 entry that contains a list of fluid elements or grids on the fluid “skin”. See Remark 2. (Integer > 0 or blank)
NORMAL	Optional identification of a SET1 entry that contains a list of structural elements or grids on the structure-fluid interface. See Remark 2. (Integer > 0 or blank)
METHOD	Fluid normal tolerance. See Remark 5. (Real; Default = 1.0 (Real; .001 for IDENT)) Default = “BW” “BW” = Body in White method “CP” = Closed Pressure Vessel Method See Remarks 10. and 11.
SKNEPS	Fluid skin growth tolerance. (Real; Default 0.5)
DSKNEPS	Secondary fluid skin growth tolerance (Real; Default .75)
INTOL	Tolerance of inward normal. (Real; Default .5)

Descriptor	Meaning
ALLSSET	If "NO" then SSET structure is searched and coupled if found. If 'YES' then all the structure given by SSET is coupled. (Character = 'YES', or 'NO'; Default = 'NO')
SRCHUNIT	Search units. (Character='ABS' for absolute model units or 'REL' for relative model units based on element size; Default = 'REL')

Remarks:

1. Only one ACMODL entry is allowed. In general, for large irregular models, it is recommended that, initially, this entry not be used, so the defaults will be applied.
2. For METHOD = "BW" and INTER = "DIFF" (Default), FSET and SSET refer to either grids or elements as selected below. For INTER = "IDENT", FSET and SSET refer to grids.
 - a. For INTER = "DIFF", INFOR = "ELEMENTS", for FSET, the search algorithm is restricted to elements referenced by FSET. For SSET, the search algorithm is restricted to elements referenced by SSET. This allows the user to de-select specific structural faces of a solid structural element. Both these sets are optional and the user can have one without the other.
 - b. For INTER = "DIFF", INFOR = "GRIDS", for FSET, the search algorithm is restricted to grids referenced by FSET. This allows the user to deselect fluid grids. Fluid grid selection is the only way to deselect specific fluid faces. For SSET, the search algorithm is restricted to grids referenced by SSET. This allows the user to de-select structural grids. Both these sets are optional and the user can have one without the other.
 - c. For INTER = "IDENT", INFOR = "ALL", the points referenced by FSET and SSET must lie exactly on the fluid-structure interface. These sets are optional, but if used, both must be present or no fluid interface is calculated.
3. For INTER = "DIFF", a .PCH file is created with a SET1 representing the fluid "skin" and a SET1 representing the structure interface. This file is useful for graphic post-processing for viewing the interface. It also produces the sets that can be used as FSET and SSET.
4. For ALLSSET = 'NO' (default) the elements and grids determined by the couplings algorithm are written to the .PCH file. The user can then deselect elements or grids as defined by the .PCH file by editing them out of the SET1 entries defined in the file and referencing the edited SET1 with the SSET. To add structural elements that the coupling algorithm did not include in the .PCH file, it is not sufficient to just include them on the SET1 entry referenced by SSET. In addition, ALLSSET = 'YES' must be specified.
5. NORMAL determines the height of the fluid box in the outward normal direction to the fluid surface. The fluid box is used to locate the structural elements used in defining the fluid-structure coupling matrix. If L is the smallest edge of the fluid element surface, then the height of the box is L x NORMAL.
For INTER = "IDENT", NORMAL = .001 is the default and represents a tolerance, in units of length, used in determining the fluid-structure interface.
6. SKNEPS represents the enlargement of the plane of the fluid surface used to define the search box. The diagonal distance from the center of the fluid surface to each surface grid is pushed out (diagonal x (1. + SKNEPS)).



7. DSKNEPS represents a secondary enlargement of the plane of the fluid surface used to define the search box if SKNEPS fails to find ANY structural elements. The diagonal distance from the center of the fluid surface to each surface grid is pushed out (diagonal x (1. + DSKNEPS)).
8. INTOL represents a normal direction into the fluid for the case when the fluid protrudes past the structural interface. It is defined as $L \times \text{INTOL}$ where L is the smallest edge of the fluid element surface.
9. The BW method is summarized in the following table:

Table of Fluid/Structure Search Control and Override METHOD=BW

INFOR	Indicates the types of FSET and SSET entered	
	ELEMENTS	Element sets are used to select element fluid and structure faces.
	GRIDS	Grid sets are used to select element fluid and structure faces.
FSET	Fluid Set ID to delete or add fluid faces to fluid/structure interface	
	ALLSET=NO	Delete fluid element faces
	ALLSET=YES	Add fluid element faces
SSET	Structure Set ID to delete or add structure faces to fluid/structure interface	

Table of Fluid/Structure Search Control and Override METHOD=BW

	ALLSET=NO	Delete structural element faces
	ALLSET=YES	Add structural element faces
		A SET specification will for the specified set given override any search parameters. Caution: When deleting items from a set, especially structural element sets an element ID may occur more than once and ALL must be removed.
SRCHUNIT	REL	Units are in the units of smallest fluid edge of current fluid face being searched. L = smallest fluid surface edge of element being searched. D = Diagonal from center of fluid surface to each grid NORMAL = Value_input * L, Height of box +normal to plane of fluid. SKNEPS = D * (1.0 + Value_input), Search plane diagonals. DSKNEPS = D * (1.0 + Value_input) Secondary search plane diagonals DSKNEPS is only used when SKNEPS < DSKNEPS INTOL = Value_input * L, Height of box -normal to plane of fluid.
	ABS	Normal units are in the UNITS of the STRUCTURE. Use when distance between the structure interface and the fluid interface is same constant everywhere. NORMAL = Value_input SKNEPS = As defined above but projected using ABS normal units DSKNEPS = As defined above but projected using ABS normal units INTOL = Value_input
ALLSSET	NO	Designed to take a modified PARAM_SKINOUT PUNCH set and DELETE elements from either the FLUID element set or the STRUCTURE element set or BOTH. Note if the FLUID set has some deleted elements and the STRUCTURE set has added elements this option may result in some change of structure elements in the set and not keep the structural set UNMODIFIED or vice versa. This is because not all of the UNMODIFIED structure elements of the set project with the reduced set of Fluid elements or vice versa. Recommended for DELETE only.
	YES	Designed to take a modified PARAM_SKINOUT PUNCH set and ADD elements to either the FLUID element set or the STRUCTURE element set or BOTH. If some previously found FLUID and STRUCTURE are also deleted the algorithm keeps all the added set elements but will of course recomputed the interface for the deleted boundary so that a sensible set intersection is computed. Recommended for DELETE ADD.

Table of Fluid/Structure Search Control and Override METHOD=BW

	<p>Do NOT use ALLSET =YES when no SETs are specified or one SET is specified and the search algorithm is being used for the other unspecified SET as ALLSET means USE ALL and all of the elements of the unspecified SET may be used.</p>
Coupling Algorithm	<ol style="list-style-type: none"> 1. Use the search box algorithm to locate the fluid free faces and the corresponding structural element faces. 2. For a fluid free face and its list of structural element faces (that were determined by boxing normal to the fluid element) do as follows: <ol style="list-style-type: none"> a. For each fluid free face establish a face coordinate system. b. Determine the resultant pressure force for each grid on the fluid element. c. Resolve this resultant pressure force for a unit grid pressure to the grids of the fluid element. (Determined by principal virtual work). d. Using the origin of the free fluid face, determine the center of pressure. e. Using rigid body motion consider only a unit motion normal to the fluid face with the appropriate moment relationships, determine the resulting load distribution at the grids of each of the structural elements. The area of each structural element projected normal to the fluid element is used as a weighting function. f. Loop over each grid of the fluid element and accumulate the forces at the structural grids. g. The algorithm always maintains of rigid body equilibrium. 3. Repeat for the next fluid element and its associated group of structural elements. Accumulate the forces at the structural grids.
Guidelines	<ol style="list-style-type: none"> 1. Run default ACMODL with PARAM , SKINOUT, PUNCH 2. View SETS both FLUID and STRUCTURE. 3. Adjust ACMODL search parameters until close to desired interface is obtained. 4. Modify the FSET, or SSET, or both to include and exclude Girds or Elements as seems appropriate to get final coupling elements.

Table of Fluid/Structure Search Control and Override METHOD=BW

	<p>5. Run acoustic analysis with ACMODL using final FSET, SSET sets with ALLSET=YES if sets contain both deleted and added elements. Or ALLSET=NO (default) if sets only delete.</p> <p>6. The ABS option IS NOT RECOMMENDED for complex acoustic models such as body in white models with multi contouring surfaces.</p>
Parallel Structural Element Meshes in the Search Box	When there are "parallel" over lapping structural meshes, the search algorithm will pick an element of the closest mesh to the fluid face. All elements connected to this first element are then checked to see if they are in the search box. Once an element, such as the elements of the outer parallel mesh, are found not to connect to the inner patch it is eliminated from the search. Thus outer patch elements tend to be excluded from the search unless some connectivity to the inner patch is detected. The ALLSET=YES is to force an interface between the listed structural elements and the nearest fluid faces.

10. The default METHOD is the new "BW" searching algorithm that requires a special license.

The pre-Version 2004 method is selected with METHOD = "CP" in which:

- The search box is not used so the SKNEPS, DSKNEPS, INTOL, ALLSET, SRCHUNIT fields are ignored.
- If INFOR = 'ALL' (METH = 'CP' only), then both FSET and SSET must be specified and matching is checked at only those grid points referenced by FSET and SSET.
- FSET and SSET refer to grids.
- NORMAL = blank is the default (recommended), $1 \leq \text{NORMAL} \leq 10$. gets acceptable results. In this case, NORMAL represents a maximum cutoff value measured in physical units. When NORMAL = 'blank', Nastran will compute the cutoff value.

This field replaces the pre-Version 2004 FSTOL field. Different units are also used so pre-Version 2004 FSTOL values may need to be changed to obtain the same results.

11. METH = 'CP' is not recommended when complex acoustic cavities (such as body in white automotive models) exist in the model.

12. If Modules are present then this entry may only be specified in the main Bulk Data section.

ACPEMCP

Trim Component Interface Coupling and Constraints Definition

Defines the interface coupling conditions and constraints of a trim component.

Format:

1	2	3	4	5	6	7	8	9	10
ACPEMCP	TID	SGLUED	SSLIDE	SOPEN	SIMPER	OOC	SPM	SAIRGAP	
	SCUX	SCUY	SCUZ	SCRX	SCRY	SCRZ	SCFP		

Example:

ACPEMCP	1		1002		1004				
---------	---	--	------	--	------	--	--	--	--

Descriptor	Meaning
TID	Identification number of trim component. (Integer>0; Required)
SGLUED	Identification number of SET1/SET3 entry of grids belonging to the solid-phase and/or structure volume elements in the trim component, which are glued to a structure; see Remark 2. (Integer ≥ 0 or Blank)
SSLIDE	Identification number of SET1/SET3 entry with grids belonging to the solid-phase and/or structure volume elements in the trim component, which are in the sliding-contact to a structure; see Remark 3. (Integer0 or Blank)
SOPEN	Identification number of SET1/SET3 entry of grids belonging to the fluid-phase elements in the trim component, which has an open interface with the cavities; see Remark 4. (Integer ≥ 0 or Blank)
SIMPER	Identification number of SET1/SET3 entry of grids of the solid-phase and/or structure volume elements in the trim component, which has an impervious interface with the cavities; see Remark 5. (Integer ≥ 0 or Blank)
OOC	Number of blocks for Out-Of-Core solver, see remark 8. (Integer > 1 , Default=1)'
SPM	Flag for selecting single precision MUMPS for computing reduced impedance matrix, see Remark 9. (Integer $>= 0$, default=0)
SAIRGAP	Identification number of SET1/SET3 entry with grids belonging to the solid-phase and/or structure volume elements in the trim component, which are in contact with structure via air gap. See Remark 10. (Integer = 0 or Blank)
SCUX	ID of SET1/SET3 with which grid IDs in the trim component TID is zero-constrained in translation X direction of output coordinate system.
SCUY	ID of SET1/SET3 with which grid IDs in the trim component TID is zero-constrained in translation Y direction of output coordinate system.
SCUZ	ID of SET1/SET3 with which grid IDs in the trim component TID is zero-constrained in translation Z direction of output coordinate system.

Descriptor	Meaning
SCRX	ID of SET1/SET3 with which grid IDs in the trim component TID is zero-constrained in rotation X direction of output coordinate system.
SCRY	ID of SET1/SET3 with which grid IDs in the trim component TID is zero-constrained in rotation Y direction of output coordinate system.
SCRZ	ID of SET1/SET3 with which grid IDs in the trim component TID is zero-constrained in rotation Z direction of output coordinate system.
SCFP	ID of SET1/SET3 with which grid IDs in the trim component TID is zero-constrained in fluid pressure.

Remarks:

1. ACPEMCP is a required entry for each trim component that is referenced by TRIMGRP. It must be put in the main Bulk Data section or BEGIN BULK. It specifies the boundary coupling conditions as well as the coupling degrees of freedom for computing the reduced boundary impedance matrices of the trim component.
2. A glued interface means that the trim component is completely constrained on to the structure, such as a car body. There is no relative movement between the trim component and the structure at the interface.
3. A sliding interface means that the trim component is laid on the surface of structure, such as a car body. There is no relative movement in the normal direction of the interface between the trim component and the structure. Relative sliding movement is allowed in the contact surface.
4. An open interface means that the free flow of fluid between the fluid-phase and cavities occurs at the interface.
5. An impervious or closed interface means that the fluid flow between the trim component and cavities is prohibited at the interface. This can happen when the porous material is covered by a thin film which stops the fluid flow or the cavity is in touch with a structure part of the trim component.
6. The SET1/SET3 entries must reside in the Bulk Data section introduced by Case Control, BEGIN BULK TRMC.
7. Under current PEM implementation, only 3D elements are allowed for trim components. Hence, SCRX, SCRY and SCRZ fields of ACPEMCP are not processed.
8. OOC is intended for large trim component. The number of OOC field defines the number of blocks to be used for the Schur complement evaluation which reduces the memory requirement. A side effect of OOC>1 is full RIM is generated, instead of lower triangular RIM, which is similar to PARAM,PEMFIRIM,1.
9. SPM is a flag for selecting single precision MUMPS in ACTRAN. With default value of 0, double precision MUMPS is utilized. Single precision MUMPS will be used with SPM field having value greater 0. Single precision MUMPS has the benefits of reduced memory requirement and better performance. However, single precision MUMPS may suffer minor degradation on accuracy.

10. An air gap interface means the trim component is assumed to be coupled to the structure through a very thin air layer. The thin air layer is taken into the analysis without being modeled (not being defined in the trim model file).

ACSRCE

Acoustic Source Specification

Defines acoustic source as a function of power vs. frequency.

$$\text{Source Strength} = \{A\} \cdot \left[\frac{1}{2\pi f} \sqrt{\frac{8\pi CP(f)}{\rho}} \right] e^{i(\theta + 2\pi f\tau)}$$

$$C = \sqrt{B/\rho}$$

Format:

1	2	3	4	5	6	7	8	9	10
ACSRCE	SID	EXCITEID	DELAYI/ DELAYR	DPHASEI/ DPHASER	TP/RP	RHO	B		

Example:

ACSRCE	103	11	20	5.0	12	1.0	15.0		
--------	-----	----	----	-----	----	-----	------	--	--

Descriptor	Meaning
SID	Load set identification number. See Remarks 1. and 3. (Integer > 0)
EXCITEID	Identification number of DAREA, FBALOAD (in FRF Based Assembly or FBA process) or SLOAD entry set the defines $\{A\}$. See Remark 6. (Integer > 0)
DELAYI	Identification number of DELAY or FBADLAY (in FRF Based Assembly or FBA process) Bulk Data entry that defines time delay τ . See Remarks 4. and 5. (Integer > 0 or blank)
DELAYR	Value of time delay τ that will be used for all fluid degrees-of-freedom that are excited by this dynamic load entry. See Remark 5. (Real or blank)
DPHASEI	Identification number of DPHASE or FBAPHAS (in FRF Based Assembly or FBA process) Bulk Data entry that defines phase angle θ . (See Remarks 4. and 5. (Integer > 0 or blank))
DPHASER	Value of phase angle θ (in degrees) that will be used for all fluid degrees-of-freedom that are excited by this dynamic load entry. See Remark 5. (Real or blank)
TP	Identification number of a TABLEDi entry that defines power versus frequency, $P(f)$. (Integer > 0)
RP	Value of power P to be used for all frequencies (Real, non-zero)
RHO	Density of the fluid. (Real > 0.0)
B	Bulk modulus of the fluid. (Real > 0.0)

Remarks:

1. Acoustic sources must be selected in the Case Control Section with DLOAD = SID.
2. For additional remarks, see the RLOAD1 entry description.

3. SID need not be unique for all ACSRCE, RLOAD1, RLOAD2, TLOAD1 and TLOAD2 dynamic load entries. The DLOAD = SID Case Control command will select all dynamic load entries with the set identification of SID.
4. The referenced EXCITEID, DELAY, and DPHASE entries must specify fluid points only.
5. If any of the DELAYI/DELAYR or DPHASEI/DPHASER fields are blank or zero, the corresponding τ or θ will be zero.
6. If there is no LOADSET Case Control command, then EXCITEID may reference DAREA and SLOAD entries. If there is a LOADSET Case Control command, then EXCITEID may reference DAREA entries as well as SLOAD entries specified by the LID field in the selected LSEQ entry corresponding to EXCITEID.
7. If Modules are present then this entry may only be specified in the main Bulk Data section.

ACTIVAT Defines Elements That Were Previously Deactivated and Should Be Reactivated for a Particular Subcase in SOL 600

This entry allows the user to re-activate certain elements that were previously deactivated in a previous subcase in SOL 600.

Format:

1	2	3	4	5	6	7	8	9	10
ACTIVAT	ID	IGEOM	ISET						

Example:

ACTIVAT	3	0	300						
---------	---	---	-----	--	--	--	--	--	--

Descriptor	Meaning
ID	Identification number of a matching Case Control ACTIVAT command defining the subcase to which these elements should be activated. (Integer; no Default)
IGEOM	Flag to control whether to update the geometry on the post file. (Integer; Default = 0) 0 Update the geometry. 1 Do not update the geometry.
ISET	ID of a list of elements described by SET3 is ID=ISET to be activated. (Integer; no Default)

Remark:

1. This entry maps to Marc's ACTIVATE History definition option.

ACTRIM**ACTRAN Trimmed Material Matrices for SOL 108/111**

Defines ACTRAN trimmed material matrices.

Format:

1	2	3	4	5	6	7	8	9	10
ACTRIM	NAME	UNIT1	UNIT2	SCLR	SCLI				

Example(s):

ACTRIM	FLOOR	31	32	2.0	0.5				
--------	-------	----	----	-----	-----	--	--	--	--

Field	Context
NAME	Name of the ACTRAN trimmed material matrices. See Remark 1. (One to eight alphanumeric characters, the first of which is alphabetic)
UNIT1	Fortran unit number of mapped data from ACTRAN. See Remark 2. (Integer > 0)
UNIT2	Fortran unit number of property matrices from ACTRAN. See Remark 2. (Integer > 0)
SCLR	The real part of complex scale factor to be multiplied to ACTRAN matrices. (Real; Default = 1.0)
SCLI	The imaginary part of complex scale factor to be multiplied to ACTRAN matrices. (Real; Default = 0.0)

Remarks:

1. ACTRAN trimmed material matrices defined by this entry will be used for frequency response analysis if it is selected via the Case Control ACTRIM = NAME.
2. The following type of ASSIGN should be specified in the FMS section with the vacant unit number (see ASSIGN statements). The unit number cannot be selected doubly.

```
ASSIGN INPUTT2='ACTRAN_trimmed.f70' UNIT=31
ASSIGN INPUTT4='ACTRAN_trimmed.op4' UNIT=32
```
3. Refer the ACTRAN manual for the details of exportation of the trimmed material matrix data for Nastran.

ADAPT**Version Adaptivity Control**

Defines controls for p-version adaptive analysis.

Format:

1	2	3	4	5	6	7	8	9	10
ADAPT	SID	ADGEN	MAXITER	PSTRRID	PMINID	PMAXID			
	PART=name1, option1=value1, option2=value2, etc., PART=name2								
	option1=value1, option2=value2, etc., PART=name3, etc.								

Example:

ADAPT	127		3	23		45			
	PART=LOWSTR, ELSET=11, TYPE=UNIP, SIGTOL=22								
	PART=HISTR, ELSET=111, ERREST=2, EPSTOL=.002								

Descriptor	Meaning	Type	Default
SID	Adapt entry ID selected in Case Control by ADAPT command.	Integer > 0	None
ADGEN	ID of the first PVAL entry generated in the adaptive process. See Remark 14.	Integer > PSTRRID, PMINID, PMAXID	1000
MAXITER	Number of analyses performed before adaptive process is stopped.	Integer > 0	3
PSTRRID	ID of PVAL entry describing the starting p-order distribution.	Integer > 0	None
PMINID	ID of PVAL entry describing the minimum p-order distribution. See Remark 10.	Integer > 0	PSTRRID
PMAXID	ID of PVAL entry describing the maximum p-order distribution. See Remark 10.	Integer > 0	PSTRRID
optioni = valuei	Assigns a value to an option described later. See Remark 16.		
PART	Part name of the elements defined in ELSET and controlled by TYPE, ERREST, ERRTOL, SIGTOL, and EPSTOL.	Character	PART = MODEL
ELSET	ID of the SET command under the SETS DEFINITION command. See Remark 7.	Integer > 0	ELSET = 999999
TYPE	p-order adjustment. See Remark 3.	Character or Integer > 0	TYPE = EBEP

Descriptor	Meaning	Type	Default
ERREST	Error estimator activation flag. See Remark 2.	Integer ≥ 0	ERREST = 1
ERRTOL	Error tolerance. Required if MAXITER is not specified.	0.0 < Real < 1.0	ERRTOL=0.01
SIGTOL	Stress tolerance. If the von Mises stress at the center of the element is below this value, the element will not participate in the error analysis.	Real ≥ 0.0	SIGTOL = 0.0
EPSTOL	Strain tolerance. If the von Mises strain at the center of the element is below this value, the element will not participate in the error analysis.	Real ≥ 0.0	EPSTOL = 1.0E-8

Remarks:

- Only one ADAPT entry may be specified. On the continuation entries, no commas can appear in columns 1 through 8 and the data in fields 2 through 9 must be specified in columns 9 through 72.
- The error estimator is activated by ERREST = 1 and is based on strain energy density sensitivity and stress discontinuity in neighboring elements. ERREST = 0 means no error estimation will be performed on the PART.
- The types of p-order adjustment are:

Type	Description
EBEP	The p-order will increase only in the elements that are required by the error analysis.
UNIP	If any element in the group has an error larger than the tolerance, all elements will be increased by one order in each direction.
NOCH	The p-order of the group does not change during the iterations.
LIST	The PVAL distribution specified as PSTRUPTID is used for the first iteration. The user is required to provide PVAL entries with IDs starting with ADGEN, and these p-distributions will be used in the following iterations

- If a PVAL ID is not specified for PSTRUPTID, PMINID, or PMAXID, then this is equivalent to no change at the last PVAL ID found for the element.
- The elements specified in the SET could overlap. In this case, the highest p_1 , highest p_2 , highest p_3 (the polynomial order of the elements in three directions) determined by the error estimator will be used.
- n restart, PMINID and PMAXID must not refer to any PVAL identification number that was generated in the previous run(s). Also, PARAM,PVALINIT must specify the desired PVAL identification number from which to restart.
- If an element in the SET does not have a PVAL for PSTRUPTID or PMINID or PMAXID, it will be excluded from the adaptivity process.

8. SET = 999999 is a reserved set that includes all elements.
9. The user can specify as many PARTs as needed.
10. Each finite element has to have a unique PVAL for PSTRUPTID, PMINID, PMAXID. Any overlap of the PVAL specification will result in a warning message and the use of the PVAL with the highest pi field (highest p_2 if same p_1 and highest p_3 if same p_1 and p_2) and the lowest CID value.
11. The p-distribution for an element specified by the PVAL entry referenced by PMAXID must be larger than the distribution specified by the PSTRUPTID, which must be larger than the distribution specified by the PMINID. A warning message will be issued if these conditions are not met, and the data is reset.
12. The solution vector of all the elements listed in the SET entries for all loads and boundary conditions will be used in the error estimation. New p values are generated for all the elements.
13. When ERREST = 0, no error analysis is performed. The p-value of the elements in the set are increased uniformly starting from p-values specified on the PVAL entry referenced by PSTRUPTID up to values specified on the PVAL entry referenced by PMAXID.
14. The intermediate PVAL entries generated will have an ID starting with ADGEN; thus, ADGEN must be larger than PSTRUPTID, PMINID, and PMAXID.
15. The displacement and stress output can be requested by a DATAREC Case Control command.
16. Each optioni = valuei must be specified on the same entry. In other words, optioni and valuei may not be specified on two separate continuation entries.

ADUMi**Dummy Element Attributes**

Defines attributes of the dummy elements ($3 \leq i \leq 9$).

Format:

1	2	3	4	5	6	7	8	9	10
ADUMi	NG	NC	NP	ND	ELNM				

Example:

ADUM8	18	0	5	0	CRAC2D				
-------	----	---	---	---	--------	--	--	--	--

Descriptor	Meaning
NG	Maximum number of grid points that may be connected by DUMi dummy element (Integer > 0.)
NC	Number of additional fields (Ai) on the CDUMi connection entry. (Integer ≥ 0)
NP	Number of additional fields (Ai) on the PDUMi property entry. ($24 \geq$ Integer ≥ 0)
ND	Number of displacement components at each grid point used in generation of the differential stiffness matrix. Zero implies no differential stiffness. (Integer 3 or 6)
ELNM	The name of the element connection and property entry. In the example above, the connection entry is named "CRAC2D" and the property entry is named "PRAC2D".

Remarks:

ADUM8 and ADUM9 are used exclusively for the CRAC2D and CRAC3D entries respectively.

AECOMP**Component for an Integrated Load Monitor Point**

Defines a component for use in monitor point definition or external splines.

Format:

1	2	3	4	5	6	7	8	9	10
AECOMP	NAME	LISTTYPE	LISTID1	LISTID2	LISTID3	LISTID4	LISTID5	LISTID6	
	LISTID7	-etc.-							

Example:

AECOMP	WING	AELIST	1001	1002					
--------	------	--------	------	------	--	--	--	--	--

Descriptor	Meaning
NAME	A character string of up to eight characters identifying the component. (Character)
LISTTYPE	One of CAERO, AELIST or CMPID for aerodynamic components and SET1 for structural components. Aerodynamic components are defined on the aerodynamic ks-set mesh while the structural components are defined on the g-set mesh. See Remarks 2. and 4.
LISTIDi	The identification number of either SET1, AELIST or CAEROi entries that define the set of grid points that comprise the component. See Remarks 2. and 4.

Remarks:

1. The Identification name must be unique among all AECOMP and AECOMPL entries.
2. If the component is defined on the structure, LISTIDs must refer to SET1 entry(ies) that define the list of associated GRID points. For the AELIST or CAERO option, the LISTIDs must refer to AELIST or CAERO i entries, respectively. Note that, for DLM models (CAERO1/2), the set of points defined by the AELIST are the box identification numbers. For example, if the control surface's grids are desired, the same AELIST used for the AESURF can be referred to here. An AECOMP component must be defined as either an aerodynamic mesh component or a structural component. The two mesh classes cannot be combined into a single component.
3. The AECOMPL entry can be used to combine AECOMP entries into new components. When combining components, the structural and aerodynamic classes must be kept separate.
4. If LISTTYPE = CMPID, LISTID1 identifies the associated set of AEQUAD4, AETRIA3 elements that define the aero component.

AECOMPL**Component for an Integrated Load Monitor Point**

Defines a component for use in aeroelastic monitor point definition or external splines as a union of other components.

Format:

1	2	3	4	5	6	7	8	9	10
AECOMPL	NAME	LABEL1	LABEL2	LABEL3	LABEL4	LABEL5	LABEL6	LABEL7	
	LABEL8	-etc.-							

Example:

AECOMPL	HORIZ	STAB	ELEV	BALANCE					
---------	-------	------	------	---------	--	--	--	--	--

Descriptor	Meaning
NAME	A character string of up to eight characters Identifying the component. (Character)
LABELi	A string of 8 characters referring to the names of other components defined by either AECOMP or other AECOMPL entries.

Remarks:

1. The Identification name must be unique among all AECOMP and AECOMPL entries.
2. The AECOMPL entry can be used to combine AECOMP entries into new components. When combining components, the structural and aerodynamic classes must be kept separate.

AEDW**Parametric Normal Wash Loading for Aerodynamics**

Defines a downwash vector associated with a particular control vector of the associated aerodynamic configuration (AECONFIG). From this downwash vector, a force vector on the aerodynamic grids will be defined for use in nonlinear static aeroelastic trim.

Format:

1	2	3	4	5	6	7	8	9	10
AEDW	MACH	SYMXZ	SYMXY	UXID	DMIJ	DMIJI			

Example:

AEDW	0.90	SYMM	ASYMM	101	ALP1				
------	------	------	-------	-----	------	--	--	--	--

Descriptor	Meaning
MACH	The Mach number for this force, see Remark 2. (Real $\geq 0.0, \neq 1.0$)
SYMXZ,SYMXY	The symmetry of this force vector. One of SYMM, ASYMM or ANTI (Character).
UXID	The identification number of a UXVEC entry that defines the control parameter vector associated with this downwash vector.
DMIJ	The name of a DMI or DMIJ entry that defines the downwash.
DMIJI	The name of a DMIJI entry that defines the CAERO2 interference element “downwashes”.

Remarks:

1. The AEDW, AEFORCE and AEPRESS are associated with the current AECOMFIG using either Case Control (if in the main Bulk Data Section) or using the BEGIN AECOMFIG=<config> if in a partition of the Bulk Data.
2. The DMIJ field refers to either a DMI or a DMIJ entry. The DMIJI is only applicable to CAERO2 and is only required if nonzero “downwash” (j-set) input is needed on the interference body elements.
3. Mach numbers > 1.0 require that the supersonic aerodynamic option be available.

AEFACT**Aerodynamic Lists**

Defines real numbers for aeroelastic analysis.

Format:

1	2	3	4	5	6	7	8	9	10
AEFACT	SID	D1	D2	D3	D4	D5	D6	D7	
	D8	D9	-etc.-						

Example:

AEFACT	97	.3	.7	1.0					
--------	----	----	----	-----	--	--	--	--	--

Descriptor	Meaning
SID	Set identification number. (Unique Integer > 0)
Di	Number. (Real)

Remarks:

1. AEFACt entries must be selected by a CAEROi, PAEROi or SPLINEX entry.
2. Embedded blank fields are not allowed.
3. To specify division points, there must be one more division point than the number of divisions.
4. When referenced by the CAERO3 entry, AEFACt defines the aerodynamic grid points. The ID number of the first point defined by each AEFACt entry is the value of the CAERO3 ID that selected the AEFACt entry. The ID of each following point defined on the AEFACt is incremented by 1.

AEFORCE

Parametric Force for Aerodynamics

Defines a vector of absolute or “per unit dynamic pressure” forces associated with a particular control vector. This force vector may be defined on either the aerodynamic mesh (ks-set) or the structural mesh (g-set). The force vector will be used in static aeroelastic trim.

Format:

1	2	3	4	5	6	7	8	9	10
AEFORCE	MACH	SYMXZ	SYMXY	UXID	MESH	LSET	DMIK	PERQ	

Example:

AEFORCE	0.90	SYMM	ASYMM	101	AERO		BETA		
---------	------	------	-------	-----	------	--	------	--	--

Descriptor	Meaning
MACH	The Mach number for this force. (Real ≥ 0.0 , $\neq 1.0$)
SYMXZ,SYMXY	The symmetry conditions for this force vector. One of SYMM, ASYMM or ANTI. (Character)
UXID	The identification number of a UXVEC entry that defines the control parameter vector associated with this force vector.
MESH	One of AERO or STRUCT that declares whether the force vector is defined on the aerodynamic ks-set mesh or the structural g-set mesh.
LSET	SID of a load set that defines the vector. See Remark 2. (Integer > 0 if MESH=STRUCT)
DMIK	The name of a DMIK entry that defines the aerodynamic force vector. See Remark 3. (Character; Required if MESH=AERO)
PERQ	The string PERQ or blank. If PERQ and MESH=STRUCT, the input FORCE set is multiplied by the dynamic pressure. (Default = blank; see Remark 4.)

Remarks:

1. The AEFORCE is associated with the current AECONFIG and must be entered for the appropriate Mach numbers and aerodynamic symmetries.
2. For the STRUCT mesh option, the LSET can refer to any existing load type (e.g., FORCE1, PLOAD4 or LOAD) that is available to define static loads.
3. For the AERO mesh option, the DMIK Bulk Data are used. Any forces associated with the aerodynamic model’s permanently SPC’d degrees-of-freedom (which are dependent on the type of aerodynamic model being used) will be ignored.
4. For the AERO mesh option, the PERQ field is not used. The FORCE data input on the DMIK entry is always multiplied by dynamic pressure.

AEGRID**Aerodynamic Grid Point**

Defines the location of an aerodynamic grid point.

Format:

1	2	3	4	5	6	7	8	9	10
AEGRID	GID	CP	X1	X2	X3	CD	ASID		

Example:

AEGRID	1	1	2.1	3.2	2.0	3	4		
--------	---	---	-----	-----	-----	---	---	--	--

Descriptor	Meaning
GID	Grid Point Identification Number ($0 < \text{Integer} < 100,000,000$)
CP	Identification number of a coordinate system in which the location of the grid point is defined. ($\text{Integer} \geq 0$ or blank)
X1, X2, X3	Location of the grid point in the coordinate system CP.
CD	Identification number of coordinate system in which the degrees-of-freedom of the grid point are defined. ($\text{Integer} \geq 0$ or blank)
ASID	Identification number of an AESCALE Bulk Data entry. ($\text{Integer} > 0$ or blank)

Remarks:

1. All grid point identification numbers must be unique with respect to all other aerodynamic grid point identification numbers.
2. If both CP and ASID are defined, coordinates are first scaled and subsequently transformed to the basic coordinate system.
3. The meaning of X1, X2 and X3 depends on the type of coordinate system CP. (See Remark 2 of the GRID entry).
4. A zero or blank in the CP and CD fields refers to the basic coordinate system.
5. The AEGRID, AETRIA3, AEQUAD4, and AESCALE entries provide an aerodynamic mesh in a readable, portable format. There are no internal aerodynamics created on this mesh.

AELINK**Links Aeroelastic Variables**

Defines relationships between or among AESTAT and AESURF entries, such that:

$$u^D + \sum_{i=1}^n C_i u_i^I = 0.0$$

Where:

u^D = dependent variable

u_i^I = independent variable

Format:

1	2	3	4	5	6	7	8	9	10
AELINK	ID	LABLD	LABL1	C1	LABL2	C2	LABL3	C3	
	LABL4	C4	-etc.-						

Example:

AELINK	10	INBDA	OTBDA	-2.0					
--------	----	-------	-------	------	--	--	--	--	--

Descriptor	Meaning
ID	If an integer > 0 is specified, this is the TRIM set ID selected in Case Control and the AELINK only applies to that subcase. If an integer value of 0 or the character string "ALWAYS" is specified, this AELINK is applicable to all subcases. (Integer ≥ 0 or the "ALWAYS" character string.)
LABLD	Character string to identify the dependent aerodynamic variable. (Character)
LABLi	Character string to identify the i-th independent aerodynamic variable. (Character)
Ci	Linking coefficient for the i-th variable. (Real)

Remarks:

1. If the ID is a positive integer, the AELINK entry (or entries) is selected by the TRIM-ID in Case Control.
2. If the ID is 0 or the character string ALWAYS, the linking relationship applies to all subcases.
3. The entry constrains the dependent variable to be a linear combination of the independent variables.
4. LABLD data must be unique for a given ID or if ID=0 or AWAYS is being used (i.e., the variable cannot be constrained more than once).
5. LABLD and LABLi refer to AEPARM, AESTAT or AESURF Bulk Data entries.

AELIST**Aerodynamic List**

Defines a list of aerodynamic elements or grid ID's.

Format:

1	2	3	4	5	6	7	8	9	10
AELIST	SID	E1	E2	E3	E4	E5	E6	E7	
	E8	-etc.-							

Example:

AELIST	75	1001	THRU	1075	1101	THRU	1109	1201	
	1202								

Descriptor	Meaning
SID	Set identification number. (Integer > 0)
Ei	List of aerodynamic box ID's or aerodynamic grid ID's (Integer>0 or "THRU")

Remarks:

1. The AELIST entry can be referenced by AESURF, AECOMP, SPLINEi, SPLINRB, SPBLND2 and SPRELAX entries. Refer to these entries for the meaning of the data provided.
2. When the "THRU" option is used, all intermediate grid points must exist. The word "THRU" may not appear in field 3 or 9 (2 or 9 for continuations).
3. Intervening blank fields are not allowed.

AELISTC

Character Item List

Defines a list of 8-character strings.

Format:

1	2	3	4	5	6	7	8	9	10
AELISTC	SID	C1	C2	C3	C4	C5	C6	C7	
	C8	-etc.-							

Example:

AELISTC	101	FBS	STRING12						
---------	-----	-----	----------	--	--	--	--	--	--

Descriptor Meaning

SID Set identification number. (Integer > 0)

Ci List of 8-character strings.

Remark:

1. Intervening blank fields are not allowed.

AEPARM**General Controller for Use in Trim**

Defines a general aerodynamic trim variable degree-of-freedom (aerodynamic extra point). The forces associated with this controller will be derived from AEDW, AEFORCE and AEPRESS input data.

Format:

1	2	3	4	5	6	7	8	9	10
AEPARM	ID	LABEL	UNITS						

Example:

AEPARM	5	THRUST	LBS						
--------	---	--------	-----	--	--	--	--	--	--

Descriptor	Meaning
ID	Controller identification number. (Integer > 0)
LABEL	Controller name. See Remark 1. (Character)
UNITS	Label used to describe the units of the controller values. (Character)

Remarks:

1. Controller LABELs that comprise the unique set relative to all the AESURF, AESTAT and AEPARM entries will define the set of trim variable degrees-of-freedom for the aeroelastic model.
2. Unit labels are optional and are only used to label outputs. No units will be associated with the controller if left blank.

AEPRESS

Parametric Pressure Loading for Aerodynamics

Defines a vector of pressure/unit dynamic pressure associated with a particular control vector. From this pressure vector, a force vector on the aerodynamic grids will be defined for use in nonlinear static aeroelastic trim.

Format:

1	2	3	4	5	6	7	8	9	10
AEPRESS	MACH	SYMXZ	SYMXY	UXID	DMIJ	DMIJI			

Example:

AEPRESS	0.90	SYMM	ASYMM	101	ALP1				
---------	------	------	-------	-----	------	--	--	--	--

Descriptor	Meaning
MACH	The Mach number for this force, see Remark 2. (Real ≥ 0.0 , $\neq 1.0$)
SYMXZ,SYMXY	The symmetry of this force vector. One of SYMM, ASYMM or ANTI. (Character)
UXID	The identification number of a UXVEC entry that defines the control parameter vector associated with this pressure vector.
DMIJ	The name of a DMI or DMIJ entry that defines the pressure per unit dynamic pressure.
DMIJI	The name of a DMIJI entry that defines the CAERO2 interference element “downwashes”.

Remarks:

1. The AEDW, AEFORCE, and AEPRESS are associated with the current AECONFIG using Case Control.
2. Mach numbers > 1.0 require that the supersonic aerodynamic option be available.
3. The DMIJ field refers to either a DMI or a DMIJ entry. The DMIJI is only applicable to CAERO2 and is only required if nonzero “downwash” (j-set) input is needed on the interference body elements.

AEQUAD4**Quadrilateral Aerodynamic Element Connection**

Defines the connectivity of a quadrilateral aerodynamic element.

Format:

1	2	3	4	5	6	7	8	9	10
AEQUAD4	EID	CMPID	G1	G2	G3	G4			

Example:

AEQUAD4	100	2	12	32	41	50			
---------	-----	---	----	----	----	----	--	--	--

Descriptor	Meaning
EID	Element Identification Number. (0 < Integer < 100,000,000)
CMPID	Aerodynamic Component Identification Number. (Integer > 0)
Gi	Grid Point Identification Numbers of Connection Points. (Integer > 0)

Remarks:

1. The geometry of a quadrilateral aerodynamic element may collapse to a triangle, i.e., two connection points may have the same geometric location. However, all four grid point identification numbers must be different.
2. The AEGRID, AETRIA3, AEQUAD4, and AESCALE entries provide an aerodynamic mesh in a readable, portable format. There are no internal aerodynamics created on this mesh.

AERO**Aerodynamic Physical Data**

Gives basic aerodynamic parameters for unsteady aerodynamics.

Format:

1	2	3	4	5	6	7	8	9	10
AERO	ACSID	VELOCITY	REFC	RHOREF	SYMXZ	SYMXY			

Example:

AERO	3	1.3+4	100.	1.-5	1	-1			
------	---	-------	------	------	---	----	--	--	--

Descriptor	Meaning
ACSID	Aerodynamic coordinate system identification. See Remark 2. (Integer ≥ 0 ; Default is the basic coordinate system)
VELOCITY	Velocity for aerodynamic force data recovery and to calculate the BOV parameter. See Remark 5. (Real > 0.0 or blank)
REFC	Reference length for reduced frequency. (Real > 0.0)
RHOREF	Reference density. (Real > 0.0)
SYMXZ	Symmetry key for the aero coordinate x-z plane. See Remark 6. (Integer = +1 for symmetry, 0 for no symmetry, and -1 for antisymmetry; Default = 0)
SYMXY	The symmetry key for the aero coordinate x-y plane can be used to simulate ground effect. (Integer = -1 for symmetry, 0 for no symmetry, and +1 for antisymmetry; Default = 0)

Remarks:

1. This entry is required for aerodynamic problems. Only one AERO entry is allowed.
2. The ACSID must be a rectangular coordinate system. Flow is in the positive x-direction.
3. Set SYMXY = -1 to simulate ground effect.
4. PARAM,WTMASS does not affect aerodynamic matrices. RHOREF must be input in mass units.
5. VELOCITY is used only in aeroelastic response analysis, and it must be equal to V on the GUST Bulk Data entry.
6. The symmetry fields on this entry are only used if neither of the Case Control commands (AESYMXY, AESYMXZ) are supplied. If either Case Control command is supplied, even the defaults from Case will override these Bulk Data entries. The Case Control symmetry is the preferred means of declaring the flow and geometric symmetry for aeroelastic analysis.

AEROS**Static Aeroelasticity Physical Data**

Defines basic parameters for static aeroelasticity.

Format:

1	2	3	4	5	6	7	8	9	10
AEROS	ACSID	RCSID	REFC	REFB	REFS	SYMXZ	SYMXY		

Example:

AEROS	10	20	10.	100.	1000.	1			
-------	----	----	-----	------	-------	---	--	--	--

Descriptor	Meaning
ACSID	Aerodynamic coordinate system identification. See Remark 2. (Integer ≥ 0 ; Default is the basic coordinate system)
RCSID	Reference coordinate system identification for rigid body motions. (Integer ≥ 0 ; Default is the basic coordinate system)
REFC	Reference chord length. (Real > 0.0)
REFB	Reference span. (Real > 0.0)
REFS	Reference wing area. (Real > 0.0)
SYMXZ	Symmetry key for the aero coordinate x-z plane. See Remark 6. (Integer = +1 for symmetry, 0 for no symmetry, and -1 for antisymmetry; Default = 0)
SYMXY	The symmetry key for the aero coordinate x-y plane can be used to simulate ground effects. (Integer = +1 for antisymmetry, 0 for no symmetry, and -1 for symmetry; Default = 0)

Remarks:

1. This entry is required for static aeroelasticity problems. Only one AEROS entry is allowed.
2. The ACSID must be a rectangular coordinate system. Flow is in the positive x-direction (T1).
3. The RCSID must be a rectangular coordinate system. All AESTAT degrees-of-freedom defining trim variables will be defined in this coordinate system.
4. REFB should be full span, even on half-span models.
5. REFS should be half area on half-span models.
6. The symmetry fields on this entry are only used if neither of the Case Control commands (AESYMXY, AESYMXZ) are supplied. If either Case Control command is supplied, even the defaults from Case will override these Bulk Data entries. The Case Control symmetry is the preferred means of declaring the flow and geometric symmetry for aeroelastic analysis.

AESCALE**Aerodynamic Grid Point Scaling**

Defines reference lengths to scale aerodynamic grid points.

Format:

1	2	3	4	5	6	7	8	9	10
AESCALE	ASID	X1REF	X2REF	X3REF					

Example:

AESCALE	100	1	1	1					
---------	-----	---	---	---	--	--	--	--	--

Descriptor	Meaning
ASID	Identification Number called out on an AEGRID entry. (0 < Integer < 100,000,000)
X1REF	Reference length to scale X1. (Real or blank; Default = 1.0)
X2REF	Reference length to scale X2. (Real or blank; Default = 1.0)
X3REF	Reference length to scale X3. (Real or blank; Default = 1.0)

Remark:

1. The scaled coordinates are computed from $XiS = Xi \cdot XiREF$.
2. The AEGRID, AETRIA3, AEQUAD4, and AESCALE entries provide an aerodynamic mesh in a readable, portable format. There are no internal aerodynamics created on this mesh.

AESTAT**Static Aeroelasticity Trim Variables**

Specifies rigid body motions to be used as trim variables in static aeroelasticity.

Format:

1	2	3	4	5	6	7	8	9	10
AESTAT	ID	LABEL							

Example:

AESTAT	5001	ANGLEA							
--------	------	--------	--	--	--	--	--	--	--

Descriptor	Meaning
ID	Identification number of an aerodynamic trim variable degree-of-freedom. See Remark 1. (Integer > 0)
LABEL	An alphanumeric string of up to eight characters used to identify the degree-of-freedom. See Remark 1. (Character)

Remarks:

1. The degrees-of-freedom defined with this entry represent rigid body motion in the reference coordinate system defined on the AEROS entry. The standard labels that define the various rigid body motions are as follows:

Table 1 Standard Labels Defining Rigid Body Motions

LABEL	Degree-of-Freedom Motion	Description
ANGLEA	u_r (R2)	Angle of Attack
SIDES	u_r (R3)	Angle of Sideslip
ROLL	\dot{u}_r (R1)	Roll Rate = $p_b/2V$
PITCH	\dot{u}_r (R2)	Pitch Rate = $q_c/2V$
YAW	\dot{u}_r (R3)	Yaw Rate = $r_b/2V$
URDD1	\ddot{u}_r (T1)	Longitudinal (See Remark 3.)
URDD2	\ddot{u}_r (T2)	Lateral
URDD3	\ddot{u}_r (T3)	Vertical
URDD4	\ddot{u}_r (R1)	Roll
URDD5	\ddot{u}_r (R2)	Pitch
URDD6	\ddot{u}_r (R3)	Yaw

These reserved names may be defined on the AEPARM entry instead, in which case the incremental load due to the unit perturbation of the rigid body degree-of-freedom (as it will with AESTAT). See the AEPARM, AEPRESS, and AEFORCE entries.

2. The degrees-of-freedom defined with this entry are variables in the static aeroelastic trim solution, unless they are constrained by referencing them with a TRIM Bulk Data entry.
3. If a label other than those above is specified, then the user must either generate the corresponding forces with an AELINK or via a DMI Bulk Data entry along with a DMAP alter that includes the DMIIN module and additional statements to merge into the appropriate matrices. Or, using AEPARM and the AEDW, AEPRESS, and/or AEFORCE, you can accomplish this purpose without the need for any alters.

AESURF**Aerodynamic Control Surface**

Specifies an aerodynamic control surface as a member of the set of aerodynamic extra points. The forces associated with this controller will be derived from rigid rotation of the aerodynamic model about the hinge line(s) and from AEDW, AEFORCE and AEPRESS input data. The mass properties of the control surface can be specified using an AESURFS entry.

Format:

1	2	3	4	5	6	7	8	9	10
AESURF	ID	LABEL	CID1	ALID1	CID2	ALID2	EFF	LDW	
	CREFC	CREFS	PLLIM	PULIM	HMLLIM	HMULIM	TQLLIM	TQULIM	

Example:

AESURF	6001	ELEV	100	100	200	200			
	10.0	180.0			-1.4E4	1.2E4	20	30	

Descriptor	Meaning
ID	Controller identification number. (Integer > 0)
LABEL	Controller name. (Character)
CIDi	Identification number of a rectangular coordinate system with a y-axis that defines the hinge line of the control surface component. (Integer > 0)
ALIDi	Identification of an AELIST Bulk Data entry that identifies all aerodynamic elements that make up the control surface component. (Integer > 0)
EFF	Control surface effectiveness. See Remark 4. (Real ≠ 0.0; Default = 1.0)
LDW	Linear downwash flag. See Remark 2. (Character, one of LDW or NOLDW; Default = LDW).
CREFC	Reference chord length for the control surface. (Real > 0.0; Default = 1.0)
CREFS	Reference surface area for the control surface. (Real > 0.0; Default = 1.0)
PLLLIM,PULIM	Lower and upper deflection limits for the control surface in radians. (Real, Default = ± π/2)
HMULLIM, HMULI M	Lower and upper hinge moment limits for the control surface in force-length units. (Real; Default = no limit)
TQLLIM,TQULIM	Set identification numbers of TABLEDi entries that provide the lower and upper deflection limits for the control surface as a function of the dynamic pressure. (Integer > 0; Default = no limit)

Remarks:

1. The ID on AESURF, AESTAT, and AEPARM entries are ignored. AESURFS can be used to define mass properties of the control surface.
2. The degrees-of-freedom defined on this entry represent a rigid body rotation of the control surface components about their hinge lines. In the default LDW (Linear DownWash) case, the downwash due to a unit perturbation of the control surface will be computed as part of the database. In the NOLDW case, the user must prescribe the controller's effects by direct definition of the induced forces using the AEPRESS, AEDW and/or AEFORCE entries.
3. Either one or two control surface components may be defined.
4. If EFF is specified, then the forces produced by this surface are modified by EFF (e.g., to achieve a 40% reduction, specify EFF=0.60).
5. The continuation is not required.
6. The CREFC and CREFS values are only used in computing the nondimensional hinge moment coefficients.
7. Position limits may be specified using either PiLIM or TQiLIM, but not both.
8. Position and hinge moment limits are not required.

AESURFS

Structural Grids on an Aerodynamic Control Surface

Optional specification of the structural nodes associated with an aerodynamic control surface that has been defined on an AESURF entry. The mass associated with these structural nodes define the control surface moment(s) of inertia about the hinge line(s).

Format:

1	2	3	4	5	6	7	8	9	10
AESURFS	ID	LABEL		LIST1		LIST2			

Example:

AESURFS	6001	ELEV		6002		6003			
---------	------	------	--	------	--	------	--	--	--

Descriptor	Meaning
ID	Controller identification number, see Remark 1. (Integer > 0)
LABEL	Controller name, see Remark 1. (Character)
LISTi	Identification number of a SET1 entry that lists the structural grid points that are associated with this component of this control surface. (Integer > 0)

Remarks:

1. The LABEL on the AESURFS entry must match one on an AESURF entry. The ID is ignored.
2. The mass of the GRID points listed on the SETi entries is used to compute the mass moment of inertia of the control surface about its i-th hinge line. The presence of these data will allow the hinge moments to include the inertial forces in the computations. These data are optional, and, if omitted, result in hinge moments which include only the applied, aeroelastically corrected, forces.
3. These data will be associated to a structural superelement by grid list or partitioned SUPER=<seid> if the AESURFS is defined in the main bulk data section.

AETRIA3**Triangular Aerodynamic Element Connection**

Defines the connectivity of a triangular aerodynamic element.

Format:

1	2	3	4	5	6	7	8	9	10
AETRIA3	EID	CMPID	G1	G2	G3				

Example:

AETRIA3	3768	8	368	872	999				
---------	------	---	-----	-----	-----	--	--	--	--

Descriptor	Meaning
EID	Element Identification Number. (0 < Integer < 100,000,000)
CMPID	Aerodynamic Component Identification Number. (Integer > 0)
Gi	Grid Point Identification Numbers of Connection Points. (Integer > 0)

Remarks:

1. The AEGRID, AETRIA3, AEQUAD4, and AESCALE entries provide an aerodynamic mesh in a readable, portable format. There are no internal aerodynamics created on this mesh.

ALIASM

Alias Element Formulation in SOL 600

Allows selected elements which normally use a default formulation to be aliased to a different formulation in SOL 600.

Format:

1	2	3	4	5	6	7	8	9	10
ALIASM	TYPE	ID1	THRU	ID2	BY	ID			

Example:

ALIASM	139	101	THRU	300	BY	2			
	138	501	THRU	600					

Descriptor	Meaning
ITYPE	Desired element formulation type. (Integer; no Default.) See Marc Vol B for a list of element types in the example, type 139 is a bilinear 3-node thin shell. ITYPE = -1 means to use the companion reduced integration element formulation (if it exists) default full integration formulation. (See Note 4)
ID1	Starting element number to be aliased. (Integer; no Default)
THRU	Enter "THRU" if a range of elements is to be specified. (Character; no Default)
ID2	Ending element number to be aliased. (Integer; no Default)
BY	Enter "BY" if the range of elements is not to be incremented by one.
ID3	Element "increment by" value. (Integer; Default = 1; must be positive)

Remarks:

1. This entry should only be used if the Marc GEOMETRY entries are identical for the original and new element types.
2. This entry may be repeated as often as desired to identify all elements requiring aliases.
3. ITYPE and ID1 are required fields. All other data fields may be blank.
4. This entry may not be combined with parameters MRALIAS, MALIAS02, MALIAS03, etc.
5. If all elements of a particular type should have alias values, it may be easier to use parameters MRALIAS, MALIAS02, MALIAS03, etc. than this entry.
6. For ITYPE = -1, the following reduced integration element types will be used:

Original Type	Reduced Integration Type	Description
26	53	Plane stress 8-node quad
27	54	Plane strain 8-node quad
28	55	Axisymmetric 8-node quad
29	56	Generalized plane strain 8+2 node quad
21	57	20-node brick
32	58	Plane strain Herrmann quad
33	59	Axisymmetric Herrmann 8-node quad
34	60	Plane strain Herrmann 8+2 node quad
35	61	Herrmann 20-node brick
62	73	Axisymmetric 8-node quad, arbitrary
63	74	Axisymmetric Herrman 8-node quad
3	114	Plane stress quad
11	115	Plane strain quad
10, 20	116	Axisymmetric ring (quad)
7	117	8-node brick
80	118	Plane strain incompressible quad
82, 83	119	Axisymmetric incompressible ring
84	120	8-node incompressible brick
75	140	4-node quad

ASET**Degrees-of-Freedom for the a-set**

Defines degrees-of-freedom in the analysis set (a-set).

Format:

1	2	3	4	5	6	7	8	9	10
ASET	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Example:

ASET	16	2	23	3516	1	4			
------	----	---	----	------	---	---	--	--	--

Descriptor	Meaning
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component numbers. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.)

Remarks:

1. Degrees-of-freedom specified on this entry form members of the mutually exclusive a-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
2. When ASET, ASET1, QSET, and/or QSET1 entries are present, all degrees-of-freedom not otherwise constrained (e.g., SPCi or MPC entries) will be placed in the omitted set (o-set).
3. In nonlinear analysis, all degrees-of-freedom attached to nonlinear elements must be placed in the a-set. In other words, if the ASET or ASET1 entry is specified then all nonlinear degrees-of-freedom must be specified on the ASET or ASET1 entry.

ASET1

Degrees-of-Freedom For the a-set, Alternate Form of ASET Entry

Defines degrees-of-freedom in the analysis set (a-set).

Format:

1	2	3	4	5	6	7	8	9	10
ASET1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	ID10	-etc.-					

Example:

ASET1	345	2	1	3	10	9	6	5	
	7	8							

Alternate Format and Example:

ASET1	C	ID1	"THRU"	ID2					
ASET1	123456	7	THRU	109					

Descriptor	Meaning
C	Component number. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.)
IDi	Grid or scalar point identification numbers. (Integer > 0; for THRU option, ID1 < ID2)

Remarks:

1. Degrees-of-freedom specified on this entry form members of the a-set that is exclusive from other sets defined by Bulk Data entries. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
2. When ASET, ASET1, QSET, and/or QSET1 entries are present, all degrees-of-freedom not otherwise constrained (e.g., SPCi or MPC entries) will be placed in the omitted set (o-set).
3. If the alternate format is used, all points in the sequence ID1 through ID2 are not required to exist, but there must be at least one degree-of-freedom in the a-set for the model, or a fatal error will result. Any points implied in the THRU that do not exist will collectively produce a warning message but will otherwise be ignored.
4. In nonlinear analysis, all degrees-of-freedom attached to nonlinear elements must be placed in the a-set. In other words, if the ASET or ASET1 entry is specified then all nonlinear degrees-of-freedom must be specified on the ASET or ASET1 entry.

ATBACC**Acceleration Field applied to ATB Segments- SOL 700**

Defines an acceleration field that will be applied to ATB segments. Used in SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
ATBACC	LID		SCALE	NX	NY	NZ			
+	NAME1	NAME2	NAME3	NAME4	NAME5	NAME6	NAME7	NAMEi	

Example:

ATBACC	32		386.088	1.0	0.0	0.0	0	0	
+	LT	MT	UT	N	H	RUL	RLL		

Field	Content
LID	Number of a set of loads. (Integer > 0; required)
SCALE	ATBACC scale factor. (Real ≥ 0.0 ; default=1.0)
NX, NY, NZ	Components of gravity vector. At least one component must be nonzero. (Real; default=0.0)
NAMEi	Name of an ATB segment as given in the first field of a B.2 entry in the ATB input file. (Character; required)

Remarks:

1. The acceleration is defined as:

$$\alpha(t) = T(t).SCALE. N$$

where SCALE is the acceleration scale factor; is the vector defined by NX, NY, and NZ; is the value interpolated at time from the table referenced by the TLOADn entry.

2. LID must be referenced by a TLOADn entry.
3. The type field on the TLOADn entry must be set to zero.
4. More than one ATBACC acceleration field can be defined per problem.
5. This acceleration field is intended to apply a crash pulse to ATB segments that define a crash dummy. The acceleration is multiplied by the mass of the segment and the resulting force is added as an external force.
6. To compare the accelerations of the ATB segments to experiments, the crash pulse needs to be subtracted from the total acceleration. The acceleration of the segments as defined on the H1 entries in the ATB input file are automatically corrected.

ATBJNT**Interface to ATB Joints- SOL 700**

This entry can only be used together with the ATBSEG entries that this joint connects. The ATBSEG entries overwrite the position and orientation of the ATB segments as specified in the ATB input file. The ATBJNT entry can be used to create a Bulk Data file containing elements and grid points to visualize the ATB segment together with its joints. This visualization of the joints makes it possible to position the ATB model in any available preprocessor. Used in SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
ATBJNT	ID	NAME							
+	G0	G1	G2	G3	EID1	EID2	EID3		
+	G4	G5	G6	G7	EID4	EID5	EID6		

Example:

ATBJNT	1	HN							
+	1010	1011	1012	1013	1004	1005	1006		
+	2010	2011	2012	2013	2004	2005	2006		

Field	Content	
LID	Unique ATBJNT number. (Integer > 0; required)	
NAME	Name of an ATB joint as given in the first field of a B.3 entry in the ATB input file. (Character; required)	
G0-G7	An ATB joint connects two segments. A local joint coordinate system is attached to each of these two segments. The position and orientation of these two coordinate systems relative to the segment coordinate systems is given on entry B.3 in the ATB input file. For each joint (J = 1, NJNT) a B.3 entry is defined in the ATB input file. The joint J connects the segment JNT(J) as given on the B.3 entry and the segment J + 1. SOL700 finds the two segments that are connected by the joint with name = NAME. The grid points G0-G3 and G4-G7 define the joint coordinate systems for the segments JNT(J) and J + 1, respectively. (Integer > 0; required)	
G0	located at the origin of the joint coordinate system for the ATB segment JNT (J)	
G1	located on the local x-axis.	
G2	located on the local y-axis.	
G3	located on the local z-axis.	
G4	located at the origin of the joint coordinate system for the ATB segment J + 1.	

Field	Content
G5	located on the local x-axis.
G6	located on the local y-axis.
G7	located on the local z-axis.

Remarks:

All elements related to an ATB segment refer to the same material number. This material number is defined on the ATBSEG entry. If the material is defined to be rigid by means of a MATRIG entry, all elements can be easily connected to the contact ellipsoid of the ATB segment by means of METHOD=RELLIPS in BCTABLE or BCONPRG entry referencing the MATRIG entry. In this way, all elements related to an ATB segment move together with the ATB segment during the analyses and can be postprocessed.

ATBSEG**Interface to ATB Segments- SOL 700**

Defines the position and orientation of the ATB segments. The position and orientation as specified on the G.2 and G.3 entries in the ATB input file will be overruled by the definitions given here.

This entry can be used to create a Bulk Data file containing elements and grid points to visualize the ATB segment, together with the contact ellipsoid and the joints it is connected by. See also ATBJNT and DYPARAM, ATBSEG. Used in SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
ATBSEG	ID	NAME	COVER	NUMELM	GSTART	ESTART	MID	PIDCOV	
+	G0	G1	G2	G3	EID1	EID2	EID3	PIDCG	

Example:

ATBSEG	1	HEAD	YES	100	1000	1000	1000		
+	1010	1011	1012	1013	1001	1002	1003	1001	

Field	Content	
ID	Unique ATBSEG number. (Integer > 0; required)	
NAME	Name of an ATB segment as given in the first field of a B.2 entry in the ATB input file. (Character; required)	
G0~G3	The grid points span the local coordinate system of the ATB segment. G0~G3 are used by SOL700 to overwrite the initial position and orientation of the segments as specified in the ATB input file. EID1-EID3 are about how to generate the grid points for an existing ATB input file. (Integer > 0; required)	
G0	located at the origin of the ATB segment.	
G1	located on the local x-axis.	
G2	located on the local y-axis.	
G3	located on the local z-axis.	

AXIC**Conical Shell Problem Flag**

Defines the existence of an axisymmetric conical shell problem.

Format:

1	2	3	4	5	6	7	8	9	10
AXIC	H								

Example:

AXIC	15								
------	----	--	--	--	--	--	--	--	--

Descriptor	Meaning
H	Highest harmonic defined for the problem. (0 ≤ Integer < 998)

Remarks:

1. Only one AXIC entry is allowed. When the AXIC entry is present, most other entries are not allowed.
The types that are allowed with the AXIC entry are listed below:

CCONEAX	MATT1	SPCADD
DAREA	MOMAX	SPCAX
DELAY	MOMENT	SUPAX
DLOAD	MPCADD	TABDMP1
DMI	MPCAX	TABLED1
DMIG	NOLIN1	TABLED2
DPHASE	NOLIN2	TABLED3
EIGB	NOLIN3	TABLED4
EIGC	NOLIN4	TABLE11
EIGP	NSM	TABLEM2
EIGR	NSM1	TABLEM3
EIGRL	NSMADD	TABLEM4
EPOINT	OMITAX	TEMPAX
FORCE	PARAM	TF
FORCEAX	PCONEAX	TIC
FREQ	POINTAX	TLOAD1
FREQ1	PRESAX	TLOAD2
FREQ2	RINGAX	TSTEP

GRAV	RFORCE
LOAD	RLOAD1
MAT1	RLOAD2
MAT2	SECTAX

2. For a discussion of the conical shell element, see the [Conical Shell Element \(RINGAX\)](#) in the *MSC Nastran Reference Guide*.

AXIF**Fluid Related Axisymmetric Parameters**

Defines basic parameters and the existence of an axisymmetric fluid analysis.

Format:

1	2	3	4	5	6	7	8	9	10
AXIF	CID	G	DRHO	DB	NOSYM	F			
	N1	N2	N3	N4	N5	-etc.-			

Example:

AXIF	2	32.2	0.12	2.4+5	YES				
	1	2	3		4		7	10	

Alternate Formats and Examples of Continuation Data:

	N1	"THRU"	Ni						
	0	THRU	10						

	N1	"THRU"	Ni	"STEP"	NS				
	0	THRU	9	STEP	3				

AXIF	100	-386.0		0.0	NO				
	0	THRU	50	STEP	5				
	52								
	54	THRU	57						
	61	THRU	65						
	68		71		72	75			
	81	92							

Descriptor	Meaning
CID	Fluid coordinate system identification number. (Integer > 0)
G	Value of gravity for fluid elements in the axial direction. (Real)
DRHO	Default mass density for fluid elements. (Real > 0.0 or blank)
DB	Default bulk modulus for fluid elements. (Real)
NOSYM	Request for nonsymmetric (sine) terms of series. (Character: "YES" or "NO")
F	Flag specifying harmonics. (Blank if harmonic is specified, or Character: "NONE")

Descriptor	Meaning
Ni	Harmonic numbers for the solution, represented by an increasing sequence of integers. On continuation entries, without the “THRU” option, blank fields are ignored. “THRU” implies all numbers including upper and lower harmonics. ($0 \leq \text{Integer} < 100$, or Character: “THRU”, “STEP” or blank)
NS	Every NSth step of the harmonic numbers specified in the “THRU” range is used for solution. If field 5 is “STEP”, $\text{Ni} = i^*NS+N1$ where i is the number of harmonics. (Integer)

Remarks:

1. Only one AXIF entry is allowed.
2. CID must reference a cylindrical or spherical coordinate system.
3. Positive gravity (+G) implies that the direction of free fall is in the -Z direction of the fluid coordinate system.
4. The DRHO value replaces blank values of RHO on the FSLIST, BDYLIST and CFLUIDi entries.
5. The DB value replaces blank values of B on the CFLUIDi entries. If the CFLUIDi entry is blank and DB is zero or blank, the fluid is incompressible.
6. If NOSYM = “YES”, both sine and cosine terms are specified. If NOSYM = “NO”, only cosine terms are specified.
7. If F = “NONE”, no harmonics are specified, no fluid elements are necessary, and no continuations may be present. In this case, AXIS = “FLUID” should not be specified in the Case Control Section.
8. Superelements cannot be used.

AXSLOT**Axisymmetric Slot Analysis Parameters**

Defines the harmonic index and the default values for acoustic analysis entries.

Format:

1	2	3	4	5	6	7	8	9	10
AXSLOT	RHOD	BD	N	WD	MD				

Example:

AXSLOT	0.003	1.5+2	3	0.75	6				
--------	-------	-------	---	------	---	--	--	--	--

Descriptor	Meaning
RHOD	Default density of fluid in units of mass/volume. (Real > 0.0 or blank)
BD	Default bulk modulus of fluid in units of force/volume ratio change. (Real \geq 0.0 or blank)
N	Harmonic index number. (Integer \geq 0 or blank)
WD	Default slot width. (Real $>$ 0.0 or blank)
MD	Default number of slots. (Integer \geq 0 or blank)

Remarks:

1. Only one AXSLOT entry is allowed.
2. If any of the RHO, B, W, and M fields on the GRID, SLBDY, CAXIFI, and CSLOTi entries are blank, then values must be specified for the RHOD, BD and MD fields.
3. If the number of slots (M) is different in different regions of the cavity, this fact may be indicated on the CSLOTi and SLBDY entries. If the number of slots is zero, no matrices for CSLOTi elements are generated.
4. BD= 0.0 implies the fluid is incompressible.

BAROR**CBAR Entry Default Values**

Defines default values for field 3 and fields 6 through 8 of the CBAR entry.

Format:

1	2	3	4	5	6	7	8	9	10
BAROR		PID			X1	X2	X3	OFFT	

Example:

BAROR		39			0.6	2.9	-5.87	GOG	
-------	--	----	--	--	-----	-----	-------	-----	--

Alternate Format and Example:

BAROR		PID			G0			OFFT	
BAROR		39			18			GOG	

Descriptor	Meaning
PID	Property identification number of the PBAR entry. (Integer > 0 or blank)
X1, X2, X3	Components of orientation vector \vec{v} , from GA, in the displacement coordinate system at GA (default), or in the basic coordinate system. See Remark 5. (Real)
G0	Alternate method to supply the orientation vector \vec{v} , using grid point G0. The direction of \vec{v} is from GA to G0. \vec{v} is then translated to End A. (Integer > 0; G0 ≠ GA or GB on CBAR entry)
OFFT	Offset vector interpretation flag. See Remark 5. (Character or blank)

Remarks:

1. The contents of fields on this entry will be assumed for any CBAR entry whose corresponding fields are blank.
2. Only one BAROR entry is allowed.
3. For an explanation of bar element geometry, see the [Three-Node Beam Element \(CBEAM3\)](#) in the *MSC Nastran Reference Guide*.
4. If field 6 is an integer, then G0 is used to define the orientation vector and X2 and X3 must be blank. If field 6 is real or blank, then X1, X2, and X3 are used.
5. OFFT is a character string code that describes how the offset and orientation vector components are to be interpreted. By default (string input is GGG or blank), the offset vectors are measured in the displacement coordinate systems at grid points A and B and the orientation vector is measured in the displacement coordinate system of grid point A. At user option, the offset vectors can be measured in an offset coordinate system relative to grid points A and B, and the orientation vector can be measured in the basic system as indicated in the following table:

String	Orientation Vector	End A Offset	End B Offset
GGG	Global	Global	Global
BGG	Basic	Global	Global
GGO	Global	Global	Offset
BGO	Basic	Global	Offset
GOG	Global	Offset	Global
BOG	Basic	Offset	Global
GOO	Global	Offset	Offset
BOO	Basic	Offset	Offset

Any attempt to specify invalid combinations results in a bulk data entry input error message. For example, a value of OOO (indicating offset and orientation vectors are specified in an offset reference system) results in a fatal error since the orientation vector cannot be specified in an offset system. The offset system x-axis is defined from GA to GB. The orientation vector \hat{v} and the offset system x-axis are then used to define the z and y axes of the offset system. (Note: The character "O" in the table replaces the obsolete character "E".)

BARRIER

Barrier for Eulerian Transport

Defines a barrier for transport in a Eulerian mesh. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
BARRIER	BID	BCID	MESH	DIR	SKFRIC				
	XMIN	XMAX	YMIN	YMAX	ZMIN	ZMAX			

Example:

BARRIER	100	20							

Descriptor	Meaning
BID	Unique barrier number. (Integer > 0 ; Required)
BCID	Number of a set of BCSEG entries that define the element faces that are barriers to Eulerian transport. See Remark 4. (Integer ≥ 0)
MESH	Denotes the ID of the Euler mesh to which the boundary condition has to be applied. See Remark 5. (Integer ≥ 0)
DIR	Allowed values are NEGX, POSX, NEGY, POSY, NEGZ and POSZ. See Remark 6. (Character)
SKFRIC	Skin friction value. See Remark 8. (Real $\geq 0.0, 0.0$)
XMIN-ZMAX	Defines a square by specifying the ranges of the x,y,z coordinates. For a square in for example the x-plane it is required that either XMIN = XMAX or that XMAX is left blank. See Remark 7. (Real)

Remarks:

1. Material cannot pass through any of the faces referenced.
2. Barriers can be modeled on the edge as well as the inside of an Eulerian mesh.
3. BARRIER overrules FLOW and FLOWT definition.
4. BCID is optional. If used, all other inputs are ignored. If not used, the barrier can be defined by either using DIR or by using XMIN, XMAX, YMIN, etc.
5. The MESH-ID is only used when multiple Euler domains have been defined and when BCID is blank. If multiple Euler domains have been defined but if the MESH-ID is blank all Euler domains will be considered in assigning the boundary condition.
6. DIR is optional. It will only be used when BCID is blank. When DIR is used XMIN, XMAX, YMIN etc. are ignored.

7. XMIN, XMAX, YMIN, etc are only used when both BCID and DIR are blank. The XMIN,YMIN option defines an area on the MESH BOX boundary as shown in [Figure 9-1](#). If neither the MIN nor MAX value has been set the default value is respectively -1E+20 and 1e+20 for the MIN and MAX value. If the MIN value has been set the default value of the MAX value is the MIN value.

8. The skin friction is defined as:

$$C_f = \frac{\tau_w}{0.5 \cdot \rho u^2}$$

Here τ_w denotes the shear friction in an Euler element adjacent to a couple surface segment where ρ is the density and u is the tangential relative velocity in the Euler element that is adjacent to a couple surface segment. SKFRIC will only be used when VISC has been set on either an EOSGAM or an EOSPOL entry. If VISC has been set and if SKFRIC has not been set then a no slip condition will be prescribed at the interface between fluid and structure.

9. Internal barriers can only be defined on existing Euler element faces using the BCID option. They cannot be defined by using the XMIN,YMIN option.

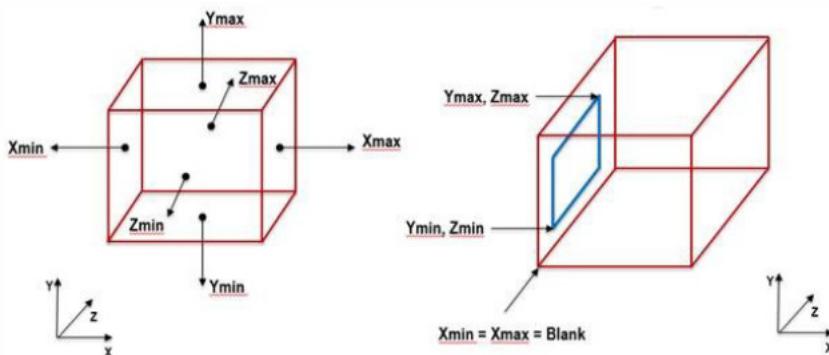


Figure 9-1 MESH BOX boundary

BCAUTOP

Automatic contact generation property.

Specifies parameters for automatic contact generation (ACG). With ACG, the code automatically generates the contact bodies based on the grids and elements given, then establishes the contact pairs in which two contact bodies are in contact or may come into contact potentially based on the DISTANCE tolerance. Please refer to the section of [Chapter 9: Contact](#), SOL 400 User's Guide for details.

Format:

1	2	3	4	5	6	7	8	9	10
BCAUTOP	PID		Param1	Value1	Param2	Value2	Param3	Value3	
	Param4	Value4	etc						

Example:

BCAUTOP			CTYPE	PGLUE	DISTANCE	0.1			
---------	--	--	-------	-------	----------	-----	--	--	--

Descriptor	Meaning
PID	Reserved for future use (integer ≥ 0 , default is 0)
Param(i)	Name of a parameter. Allowable names are given below (Character).
Value(i)	Value of the parameter. See below (Real, Integer or character). See Remark 3.

Remarks:

1. Only one entry of BCAUTOP is allowed.
2. This entry does not have effect if BCONTACT=AUTO is not present in case control section.
3. The parameters in this entry may be divided into two types – primary parameters and secondary parameters. The primary parameters are the most important parameters for build of contact bodies, match of contact pairs. The secondary parameters are helpful for user to provide the more detail and further requirement for contact model establishment and contact analysis with automatic contact generation. The primary and secondary parameters are listed in the tables as follows.

Table 9-1 Primary Parameters in BCAUTOP

Descriptor	Meaning
CTYPE	Characters, one of TOUCH(default), PGLUE, SGLUE and GGLUE. If CTYPE is specified in both BCAUTOP entry and Case Control Command BCONTACT=AUTO, the specification in BCAUTOP entry will be used.
DISTANCE	Distance tolerance of contact pair. If the distance between any two points which are belonging to two different contact bodies is less than this value, these two contact bodies are recognized to be a contact pair. Default value of DISTANCE is 100 times of contact ERROR tolerance. ERROR may be defined in either BCAUTOP or left blank. With default of ERROR, the code calculates ERROR as the smallest one of the following values:
	<ol style="list-style-type: none"> 1. 1/20 of the smallest nonzero element dimension (plates or solids) in the contact body; 2. 1/4 of the thinnest shell thickness in the contact body.
	Please see BCPARA for ERROR definition in detail.
ESET	ID of SET1 to specify element ids consisting contact bodies. If it is not specified, all elements in the model will be used. Only one of ESET, SeedESet, SeedGSet and PropSet can be specified in the same entry.
SeedESet	ID of SET1 to specify element ids as seeds of contact bodies. The program searches elements as contact bodies from the seed elements. Only one of ESET,SeedESet, SeedGSet and PropSet can be specified.
SeedGSet	ID of SET1 to specify grids ids as seeds of contact bodies. The program searches elements as contact bodies from the elements connecting to the seed grid ids. Only one of ESET,SeedESet, SeedGSet and PropSet can be specified.
PropSet	ID of SET1 to specify property ids to construct contact bodies. Only one of ESET,SeedESet, SeedGSet and PropSet can be specified.
RigidSet	ID of SET1 to specify IDs of rigid bodies to be in contact

Table 9-2 Secondary Parameters in BCAUTOP

Descriptor	Meaning
BEAMCNT	Characters, YES or NO (default). Determining if beam is included in contact definition. YES: BEAMs are included in the contact description. NO: No BEAM is included in the contact description.
BEAMCAP	Characters, FREEEND(default) or NO. If set to FREEEND(default), BCSCAP entries will be added to free end of beam elements, if NO, no BCSCAP will be added. It is mainly used for contact pair detection where the contact occurs at free beam end.
DIMENS	Characters, 2D or 3D (default), dimension of contact bodies.
EDGECNT	Characters, YES or NO (default). Determining if shell edge is included in contact description. YES: The free and hard shell edges are included in the contact definition NO: No shell edges are included in the contact definition.
FTYPE	Characters, BLCOUL/BLSHEAR; or Integer, 6 / 7. BLCOUL or 6 is Bilinear Coulomb friction. BLSHEAR or 7 is Bilinear Shear friction. Default is No Friction.
FRIC	Friction coefficient. If the value is an integer, it represents the ID of a TABLEM1, TABLEM2 or TABL3D, i.e., a temperature-dependent or multi-dimensional table. (Real ≥ 0.0 or Integer > 0 ; Default = 0.0)
IGNTHK	Character, YES or NO (default), Ignore thickness of shell for contact YES: ignore the thickness of the shell NO: include the thickness of the shell.
INISTF	Character, YES or NO (default). Set the option of initial stress free. Yes: initial stress free contact No: general contact without implementation of initial stress free.
SelfCont	Characters, YES or NO (default). Option of self-contact.

Note:

Note that the parameters defined in BCAUTOP are applied only for BONATCT=AUTO. Their default values are defined only when BCONTACT=AUTO is applied. All the parameters in BCPARA, BCONPRP, BCONPRG and BCBDPRP entries may be used in BCAUTOP directly. For the detailed list of these parameters, please refer to BCPARA, BCONPRP, BCONPRG and BCBDPRP.

BCBDPRP**Contact Body Parameters in SOLs 101 and 400**

Defines contact body parameters used in SOLs 101 and 400 only. The parameters defined here are referenced by the BCBODY1 entry.

Format:

1	2	3	4	5	6	7	8	9	10
BCBDPRP	PID		PARAM1	VAL1	PARAM2	VAL2	PARAM3	VAL3	
	PARAM4	VAL4	PARAM5	VAL5	-etc.-				

Example:

BCBDPRP	90		FRIC	0.05	ISTYP	0			
---------	----	--	------	------	-------	---	--	--	--

Descriptor	Meaning
------------	---------

PID Parameter identification number (Integer > 0).

PARAMi Name of a parameter. Allowable names are given in [Table 9-3](#) (Character).

VALi Value of a parameter. See [Table 9-3](#) (Real or Integer).

Table 9-3

Contact Body Parameters in SOLs 101 and 400

Name	Description, Type and Value (Default is 0 for integer, 0.0 for Real Unless Otherwise Indicated)
BNC	Exponent associated with the natural convective heat flow for near field behavior. (Real or Integer; Default = 1.0). If Real, the value entered is the exponent associated with the near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 OR TABLEM2 entry specifying the exponent associated with the near field natural convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the near field natural convection coefficient vs temperature and possibly other variables.
BNCE	Exponent associated with natural convection heat flow to the environment. (Real or Integer; Default = 1.0). If Real, the value entered is the exponent associated with the environment natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the exponent associated with the environment natural convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the environment natural convection coefficient vs temperature and possibly other variables.
BNL	Exponent associated with the nonlinear convective heat flow for near field behavior. (Real or Integer; Default = 1.0) If Real, the value entered is the exponent associated with the near field nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 OR TABLEM2 entry specifying the exponent associated with the near field nonlinear convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the near field nonlinear convection coefficient vs temperature and possibly other variables.
BNLE	Exponent associated with the nonlinear convective heat flow to the environment. (Real or Integer; Default = 1.0) If Real, the value entered is the exponent associated with the environment nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the exponent associated with the environment nonlinear convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the environment nonlinear convection coefficient vs temperature and possibly other variables.
CFILM	Heat transfer coefficient (film) to environment. (Real or Integer, Default = 0.0) If Real, the value entered is the film coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the heat transfer coefficient vs temperature or a TABL3D entry specifying the film coefficient vs temperature and possibly other variables.
CMB	Heat capacity of the rigid body, when entered as a geometric entity with an associated scalar point. (Real ≥ 0.0 ; Default = 0.0) For nonzero values, a grounded CDAMP4 element with the scalar point on its first side is generated internally which obtains this capacity.
CMS	Heat capacity of the environment, when associated with a scalar point. (Real ≥ 0.0 ; Default = 0.0) For nonzero values a grounded CDAMP4 element with the scalar point on its first side is generated internally which obtains this capacity.
COPTB	Flag to indicate how body surfaces may contact. See Remark 2. on the BCONPRG entry. (Integer; Default = 0)

EMISS	Emissivity for radiation to the environment or near thermal radiation. (Real or Integer; Default = 0.0) If real, the value entered is the emissivity. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the emissivity vs temperature or a TABL3D entry specifying the emissivity vs temperature and possibly other variables.
FRIC	<p>Friction coefficient. If the value is an integer, it represents the ID of a TABLEM1, TABLEM2 or TABL3D, i.e., a temperature-dependent or multi-dimensional table. (Real ≥ 0.0 or Integer > 0; Default = 0.0)</p> <p>When a grid point contacts a rigid body, the coefficient of friction associated with the rigid body is used. When a grid point contacts a deformable body, the average of the coefficients for the two bodies are used.</p> <p>In general, entering the friction coefficient for a contact body pair via a BCTABLE and BCONECT/BCONPRP is strongly recommended.</p>
HBL	Separation distance dependent thermal convection coefficient. (Real or Integer; Default = 0.0) If Real, the value entered is the separation distance dependent thermal convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the convection coefficient vs temperature or a TABL3D entry specifying the convection coefficient vs temperature and possibly other variables.
HCT	Contact heat transfer coefficient. (Real or Integer; Default = 0.0) If Real, the value entered is the contact heat transfer coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the contact heat transfer coefficient vs temperature or a TABL3D entry specifying the contact heat transfer coefficient vs temperature and possibly other variables. See Remark 3.
HCV	Convection coefficient for near field behavior. (Real or Integer; Default = 0.0) If Real, the value entered is the near field convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field convection coefficient vs temperature or a TABL3D entry specifying the near field convection coefficient vs temperature and possibly other variables.
HNC	Natural convection coefficient for near field behavior. (Real or Integer; Default = 0.0) If Real, the value entered is the near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field natural convection coefficient vs temperature or a TABL3D entry specifying the near field natural convection coefficient vs temperature and possibly other variables.
HNCE	Natural convection coefficient for heat flow to the environment. (Real or Integer; Default = 0.0) If Real, the value entered is the environment natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the environment natural convection coefficient vs temperature or a TABL3D entry specifying the environment natural convection coefficient vs temperature and possibly other variables.
HNL	Heat transfer coefficient for nonlinear convective heat flow for near field behavior. (Real or Integer; Default = 0.0) If Real, the value entered is the near field nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field nonlinear convection coefficient vs temperature or a TABL3D entry specifying the near field nonlinear convection coefficient vs temperature and possibly other variables.

HNLE	Heat transfer coefficient for nonlinear convective heat flow to the environment. (Real ≥ 0.0 or Integer; Default = 0.0) If Real, the value entered is the environment nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the environment nonlinear convection coefficient vs temperature or a TABL3D entry specifying the environment nonlinear convection coefficient vs temperature and possibly other variables.
IDSPL	Controls geometric smoothing of boundary of deformable body option. (Integer; Default = 0) =0 or blank, discrete geometric representation. >0 the surface of the body is smoothed out with splines (2D) or Coons surfaces (3D) and discontinuity edges/corners are being defined by using abs(IDSPL) as the ID of the BLSEG entries. If BLSEG with ID=abs(IDSPL) does not exist, the whole body is smoothed and there are no user-defined discontinuity corners(2D) or edges(3D). (See Remark 1.) <0 Same as IDSPL>0. Furthermore, additional discontinuity edges are being generated automatically if the difference in patch normals exceeds the value of SANGLE.
ISTYP	Check of contact conditions. (Integer ≥ 0 ; Default = 0 for all solutions) ISTYP is not necessary in segment-to-segment contact. For a deformable body: =0 check each body, versus the other. =2 double-sided contact with automatic optimization of contact constraint equations (this option is known as "optimized contact"). Note that ISTYP is supported with ISEARCH=0 in BCTABLE or BCONPRG only.
ITYPE	An option entry for heat transfer only (Integer; no Default) 1 - Heat sink (rigid body) 2 - Deformable body (with heat conduction) 4 - Heat conduction body (heat-rigid body) (Not supported in SOL 400 coupled thermal-mechanical analysis, i.e. the same contact body with ITYPE=4 cannot be used in mechanical analysis of SOL 400.)

MIDNOD	Mid-side node projection flag. (Integer ≥ 0 ; Default = 0) When MIDNOD > 0 and IDSPL $\neq 0$, the mid-side grid of quadratic elements are projected onto the selected spline surfaces. This operation is performed before the contact process starts and it may change the location of grids in contact bodies. It may operate in combination with the initial stress-free contact. Only used if IDSPL is not zero.
SANGLE	Threshold for automatic discontinuity detection in degrees. (Real; Default = 60.0) Used for geometric smoothing option in SOL 400 only. SANGLE is not used and is always set to 0.0 when IDSPL ≥ 0 .
TBODY	Body temperature. (Real or Integer; Default = 0.0) If Real, the value entered is the body temperature. If Integer, the value entered is the ID of a TABLED1 or TABLED2 entry specifying the body temperature vs time or a TABL3D entry specifying the body temperature vs time and possibly other variables. When entered as a negative integer, its absolute value is a scalar point identification number. If a scalar point is specified on this entry, it need not be defined on an SPOINT entry. See Remark 8.
TSINK	Environment sink temperature. (Real or Integer, Default = 0.0) If Real, the value entered is the sink temperature. If Integer, the value entered is the ID of a TABLED1 or TABLED2 entry specifying temperature vs time or a TABL3D entry specifying the sink temperature vs time and possibly other variables. When entered as a negative integer its absolute value is a scalar point identification number. If a scalar point is specified on this entry, it need not be defined on an SPOINT entry. See Remark 8.

Remarks:

1. When IDSPL is greater than 1, these nodes are entered in pairs. For a quad surface (for example, CQUAD4 or edge of a CHEXA) usually 4 sets of nodal pairs are needed to describe the surface. For example, a CQUAD4 with grid numbering 1,2,4,3 would need pairs of nodes, 1,2 2,4 4,3 3,1. The nodal pairs may be entered in any order. See *MSC Nastran Nonlinear User's Guide*, Chapter 9, Contact for more details.
2. For hard contact, with HGLUE=1 (see BCONPRP for the meaning of HGLUE):
 - a. The temperature of the contacting grid is tied to the temperatures of the contacted element face or the temperature of the rigid geometry when it has a scalar point associated with it.
 - b. The temperature of the contacting grid is set to the rigid geometry temperature when it has no scalar point associated with it.

Note: “Glued” thermal contact can result in overshoot of the temperatures at the interface, in particular, if two bodies that have non-uniform initial temperatures are placed in contact. The overshoot effect may be damped somewhat if one uses a near contact distance with some convective heat transfer.

3. For hard contact, with HGLUE=0:

The convective heat flow per unit area over the two interfaces is given by:

$$q = HCT \cdot (T_A - T_B)$$

where T_A is the contacting grid temperature and T_B is the face temperature in the contact point in case of a meshed body or the T_{BODY} temperature in case of a rigid geometry.

It is recommended to enter the HCT value on the BCONPRP Bulk Data entry, since it generally applies to body pairs and is not a property of a single body.

4. For near contact:

$$q = HCV \cdot (T_A - T_B) +$$

$$HNC \cdot (T_A - T_B)^{BNC} +$$

$$HNL \cdot (T_A^{BNL} - T_B^{BNL}) +$$

$$\sigma \cdot EMISS \cdot (T_A^4 - T_B^4) +$$

$$\left[HCT \cdot \left(1 - \frac{dist}{DQNEAR} \right) + HBL \cdot \frac{dist}{DQNEAR} \right] (T_A - T_B)$$

where the last term is only activated when $HBL \neq 0$, T_A is the contacting grid temperature and T_B is the face temperature in the contact point in case of a meshed body or the T_{BODY} temperature in case of a rigid geometry.

It is recommended to enter the near contact heat transfer coefficients and the corresponding exponents on the BCONPRP Bulk Data entry, since they generally apply to body pairs and are not properties of a single body.

5. For no contact:

$$q = CFILM \cdot (T_A - T_{SINK}) +$$

$$HNCE \cdot (T_A - T_{SINK})^{BNCE} +$$

$$HNLE \cdot (T_A^{BNLE} - T_{SINK}^{BNLE}) +$$

$$\sigma \cdot EMISS \cdot (T_A^4 - T_{SINK}^4)$$

6. The heat transfer coefficients and associated exponents can all be temperature dependent, when they are entered as an integer value. This integer value is the table ID of a TABLEM1, TABLEM2 or TABL3D entry (formulas are not supported on TABL3D).
7. The T_{SINK} and T_{BODY} temperatures can be time dependent when they are entered as a positive integer value. This integer value is the table ID of a TABLED1, TABLED2 or TABL3D entry (formulas are not supported on TABL3D).
8. TBODY entries only apply to rigid bodies (i.e., RIGID as the BEHAV value in BCBODY1 entry). TSINK entries only apply to deformable bodies (meshed regions with DEFORM or HEAT as the BEHAV value in BCBODY1 entry).

BCBMRAD

Modify Equivalent Radius for Beam-to-Beam Contact

Allows the equivalent radius in beam-to-beam contact to be different for each beam cross section in SOL 101, SOL 400 and SOL 600. The BCBMRAD entry is only used for node-to-segment beam-to-beam contact. For segment-to-segment beam contact, the beam cross section geometry is defined via PBARL or PBEAML

Format:

1	2	3	4	5	6	7	8	9	10
BCBMRAD	RADIUS	TYPE	ID1	ID2	THRU	ID3	BY	N	
	ID4	THRU	ID5	ID6	ID7	ID8	ID9		

Example 1:

BCBMRAD	2.5	EID	100	20	THRU	300	BY	2	
	200	3457	8456	4712	1000	THRU	2000		
	3.0		4112	THRU	4700				
	2.8	BODY	502	517	3459				

Example 2:

BCBMRAD	2.5	ALL							
	2.8	EID	2567	1240	THRU	1760			

Descriptor	Meaning
RADIUS	Equivalent radius to be used for beam-beam contact problems. See Remark 6. (Real; no Default)
TYPE	The attribute of all following ID's. (Character; Default = "EID")
EID	Defines all the following entries are the IDs of beam-type elements.
BODY	Defines all the following entries are the IDs of BCBODYs.
ALL	Defines the default RADIUS for all beam-type elements.
IDi	ID of a beam-type element, CROD, CBAR, CBEAM and CBEAM3, or a BCBODY with the specified radius. (Integer; no Default)

Remarks:

1. Multiple BCBMRAD Bulk Data entries, which are open-ended are allowed in one file.
2. In each entry of BCBMRAD, there is only one RADIUS input allows, on the field 2. From the field 4 to the rest fields, including all continuation entries, user can input all ID's in any combination of the following 3 basic formats
 - ID1 ID2 ID3 ID4 ...
 - ID1 THRU ID2

- ID1 THRU ID2 BY N

Note that blank fields are allowed for readability.

3. When all beam contact radii are the same, user can put “ALL” on the field 3 following a RADIUS value on the BCBMRAD Bulk Data entry. This value of RADIUS will be applied to all beam-type elements (See the previous Example 2). User can also use this way to give default radius of all beam-type elements.
4. The RADIUS value of the other BCBMRAD's (TYPE=BODY or EID) can override the default value (TYPE=ALL) in the following order:
 - a. When any BCBODY is selected on BCBMRAD with TYPE=BODY, the specified RADIUS will be applied to all beam-type elements in this BCBODY. This value always overrides the default (TYPE=ALL).
 - b. The RADIUS with TYPE=EID always overrides the value from TYPE=BODY and ALL.
5. When TYPE=ALL, then all IDi must be blank.
6. For tubes or round bars, enter the outer radius. For beam, enter an equivalent radius calculated as follows:

$$I = 0.5 \cdot (I_x + I_y)$$

$$R = \sqrt{\frac{A}{p_i^2} + 2 \cdot \frac{I}{A}}$$

7. SOL 600 does not support the TYPE=ALL option.
8. This is only used for node-to-segment contact.

BCBODY

Flexible or Rigid Contact Body in 2D and 3D

Defines a flexible or rigid contact body in 2D or 3D used in SOLs 101, 400, and 700 only. The BCBODY1 option is the preferred method to defining contact bodies for SOL 400.

Use only as many forms (i.e. HEAT, PATCH3D, BEZIER, POLY, CYLIND, SPHERE, NURBS2, or NURBS) as necessary to describe the body (if rigid). Deformable bodies are described using as many standard elements as necessary and are specified by the BSID field with BEHAV=DEFORM (only the first line should be entered for deformable bodies). Unless shrink fit its being analyzed, deformable bodies should not be inside other deformable bodies when the thickness of each body is taken into account.

The “RIGID” header may be used with any of the other rigid entries but only once per body. Also, only one of the character entries after RIGID (HEAT, PATCH3D, NURBS, etc.) should be entered for any particular body. See Remark 4. for an important note regarding how to define the outward direction of rigid bodies (which must point towards a flexible body for contact to occur).

Format: (SOLs 101 and 400 only)

1	2	3	4	5	6	7	8	9	10
BCBODY	BID	DIM	BEHAV	BSID	ISTYP	FRIC	IDSPL	CONTROL	
	NLOAD	ANGVEL	DCOS1	DCOS2	DCOS3	VELRB1	VELRB2	VELRB3	
	“ADVANCE”	SANGLE	COPTB		MIDNO D				
	“RIGID”	CGID	NENT	--- Rigid Body Name ---					
	“APPROV”	A	N1	N2	N3	V1	V2	V3	
	“GROW”	GF1	GF2	GF3	TAB-GF1	TAB-GF2	TAB-GF3		
	“HEAT”	CFILM	TSINK	CHEAT	TBODY	HCV	HNC	ITYPE	
		BNC	EMISS	HBL	HNL	BNL	HNLE	BNLE	
		HNCE	BNCE	CMB	CMS				
	“PATCH3D”	NPATCH							
		IDP	G1	G2	G3	G4			
		IDP	G1	G2	G3	G4			
		etc.	(npatch entries)						
	“BEZIER”	NP1	NP2	NSUB1	NSUB2				
		G1	G2	G3	G4	etc		(np1*np2 values)	
	“NURBS2D”	NPTU	NORU	NSUB				(2D Contact)	
		G1 or X1	G2 or Y1	G3	G4 or X2	G5 or Y2	G6	[abs(nptu) grids or x, y,z, values]	See Remark 8

		Homo1	Homo2	Homo3	Homo4	etc.		(nptu values)	
		Knot1	Knot2	Knot3	Knot4	Knot5	etc.	(nptu+nor u values)	
	"NURBS"	NPTU	NPTV	NORU	NORV	NSUBU	NSUBV	NTRIM	
		G1 or X1	G2 or Y1	G3 or Z1	G4 or X2	G5 or Y2	G6 or Z2	G7	See Remark 8
		G8 or X3	G9 or Y3	G10 or Z3	etc.		[abs(nptu) *nptv values]	See Remark 8	
		Homo1	Homo2	Homo3	Homo4	Homo5	Homo6	Homo7	
		Homo8	Homo9	etc.			(nptu*npt v ales)		
		Knot1	Knot2	Knot3	Knot4	Knot5	Knot6	Knot7	'
		Knot8	Knot9	etc.			(nptu+nor u+nptv+n orv values)		
		IDtrim	NPTUtrim	NORUtrim	NSUBtrim		(repeat this and all following lines NTRIM times)		
			Xisoparam	Yisoparam			(NPTUtri m entries)		
			Homo1	Homo2	Homo3	etc	(NPTUtri m entries)		
			Knot1	Knot2	Knot3	etc.	(NPTUtri m + NORUtri m entries)		

Format: (SOL 700 only)

BCBODY	BID	DIM	BEHAV	BSID					
--------	-----	-----	-------	------	--	--	--	--	--

Examples (of Deformable and Rigid Contact):

Example 1 -- Typical deformable body

BCBODY	1		DEFORM	101	0	.05			
--------	---	--	--------	-----	---	-----	--	--	--

Example 2 -- Simple 4-node rigid patch (see Remark 4. for rigid bodies)

BCBODY	2		RIGID	102	0	.08			
	PATCH3D	1							
		1	101	102	103	104			

Descriptor	Meaning
BID	Contact body identification number referenced by BCTABLE, BCHANGE, or BCMOVE. (Integer > 0; Required)
DIM	Dimension of body. (Character; Default= 3D) (Ignored by SOL 700) DIM=2D planar body in x-y plane of the basic coordinate system, composed of 2D elements or curves. DIM=3D any 3D body composed of rigid surfaces, shell elements or solid elements.
BEHAV	Behavior of curve or surface (Character; Default = DEFORM) (Ignored by SOL 700) DEFORM body is deformable, RIGID body is rigid, SYMM body is a symmetry body, HEAT indicates body is a heat-rigid body. <i>See Remark 4. for Rigid Bodies.</i>
BSID	Identification number of a BSURF or BCPROP entry if BEHAV=DEFORM. For SOL 700 the BSID may also be the identification number of a BCBOX, BCMATL, BCSEG, BCGRID or BCELIPS entry. (Integer > 0)
ISTYP	Check of contact conditions. (Integer ≥ 0 ; Default = 0 for all solutions) ISTYP is not necessary in segment-to-segment contact. For a deformable body: =0 check each body, versus the other . =2 double-sided contact with automatic optimization of contact constraint equations (this option is known as “optimized contact”).
	Note that ISTYP is supported with ISEARCH=0 in BCTABLE or BCONPRG only.
FRIC	Friction coefficient. If the value is an integer, it represents the ID of a TABLEM1, TABLEM2 or TABL3D, i.e., a temperature-dependent or multi-dimensional table. (Real ≥ 0.0 or Integer > 0; Default = 0.0) When a grid point contacts a rigid body, the coefficient of friction associated with the rigid body is used. When a grid point contacts a deformable body, the average of the coefficients for the two bodies are used.
	In general, entering the friction coefficient for a contact body pair via a BCTABLE and BCNECT/BCONPRP is strongly recommended.
IDSPL	Controls geometric smoothing of boundary of deformable body. (Integer; Default = 0) =0 or blank, discrete geometric representation >0 the surface of the body is smoothed out with splines (2D) or Coons surfaces (3D) and discontinuity edges/corners are being defined by using abs (IDSPL) as the ID of the BLSEG entries. If BLSEG with ID=abs(IDSPL) does not exist, the whole body is smoothed and there are no user-defined discontinuity corners(2D) or edges(3D). (See Remark 7.) <0 Same as IDSPL>0. Furthermore, additional discontinuity edges are being generated automatically if the difference in patch normals exceeds the value of SANGLE.

Descriptor	Meaning
CONTROL	= -1 for position control. The coordinates of the final position of GRID Point defined in CGID is given in VELRBi in the following line. = 0 for velocity control (default). = positive number for "load control". The positive number entered is the grid number defined in CGID at which translational forces or SPCD are applied. (Note: The rotation in this case is defined by NLOAD in the following line.)
NLOAD	Enter a positive number if "load controlled" and rotations are allowed (Integer). The positive number is the grid number where the moments or rotations are applied. The rotations are specified using SPCD at grid ID NLOAD and can be specified using dof's 1-3 (for rotation about x, y, z respectively), or by dof's 4-6 (for rotation about x, y, z respectively). Note: This rotation takes the position of the grid point defined in CGID field as the center of rotation.
ANGVEL	Angular velocity or angular position about local axis through center of rotation. If the value is an integer it represents the ID of a TABLED1, TABLED2 or TABL3D, i.e., a time-dependent or multi-dimensional table; however, no log scales, only linear scales. (Real or Integer; Default = 0.0)
DCOSi	Components of direction cosine of local axis if ANGVEL is nonzero. If the value is an integer, it represents the ID of a TABLED1, TABLED2 or TABL3D, i.e., a time-dependent or multi-dimensional table; however, no log scales, only linear scales. (Real or Integer; Default=0.0) In 2D contact only DCOS3 is used and the Default is 1.0.
VELRBi	Translation velocity or final position (depending on the value of CONTROL) of rigid body at the grid point defined in CGID filed. For velocity control only, if the value is an integer, it represents the ID of TABLED1, TABLED2 or TABL3D, i.e., a time-dependent or multi-dimensional table; however, no log scales, only linear scales. Only VELRB1 and VELRB2 are used in 2D contact. (Real or Integer; Default = 0.0)
"ADVANCE"	The entries for this continuation line are for advanced options starting with MD Nastran R2.
SANGLE	Threshold for automatic discontinuity detection in degrees. (Real; Default = 60.0) Used for SPLINE option in SOL 400 only. SANGLE is not used when IDSPL \geq 0.
COPTB	Flag to indicate how body surfaces may contact. See Remark 9. on the BCTABLE entry. (Integer; Default = 0)
MIDNOD	Mid-side node projection flag. (Integer \geq 0; Default = 0) When MIDNOD > 0 and IDSPL \neq 0, the mid-side grid of quadratic elements are projected onto the selected spline surfaces. This operation is performed before the contact process starts and it may change the location of grids in contact bodies. It may operate in combination with the initial stress-free contact.
"RIGID"	The entries of this continuation line are for the rigid body description. See Remark 4.

Descriptor	Meaning
CGID (5,i) i=1,2,3	Grid point identification number defining the initial position of the reference point of the rigid body or the point where a concentrated force or moment is applied
NENT	Number of geometric entities to describe this rigid surface. A rigid surface can be described by multiple sets of patches, nurbs, etc. For example, if it takes 3 sets of PATCH3D entries to describe a rigid surface, then set NENT=3. (Integer > 0; Default = 1)
Rigid Body Name	Name associated with the rigid body. (Default is blank, 24-characters maximum)
“APPROV”	The entries of this continuation line are for approaching velocity to establish initial contact.
A	Angular velocity about local axis through center of rotation. (Real, Default = 0.0)
Ni	Components of direction cosines of local axis of the angular velocity. The N1, N2, N3 define the axis through the point defined in the “RIGID”, CGID entry. Only N1 and N2 are used in 2D contact. (Real, Default = 0.0)
Vi	V1, V2 and V3 define the three components of the approaching velocity. Only V1 and V2 are used in 2D contact. (Real; Default = 0.0)
“GROW”	The entries of this continuation line are for rigid body growth. If tables are used for growth, they should either be TABLED1, TABLED2(growth vs time) or TABL3D (growth vs one or more variables).
GFi	Components of Growth factor of rigid body in the coordinate system of the “RIGID”, CGID entry. (Real, Default = 1.0)
TAB-GFi	Tabled IDs for growth factor of rigid body in the coordinate system of the “RIGID”, CGID entry. (Integer > 0 or blank, Default is blank)
“HEAT”	The entries of this continuation line(s) are for contact in heat transfer in a pure thermal analysis or in a coupled thermal/structural analysis. In a pure structural analysis they are ignored.
CFILM (9,1)/(10,1)	Heat transfer coefficient (film) to environment. (Real or Integer, Default = 0.0) If Real, the value entered is the film coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the heat transfer coefficient vs temperature or a TABL3D entry specifying the film coefficient vs temperature and possibly other variables.
TSINK (9,2)/(10,2)	Environment sink temperature. (Real or Integer, Default = 0.0). If Real, the value entered is the sink temperature. If Integer, the value entered is the ID of a TABLED1 or TABLED2 entry specifying temperature vs time or a TABL3D entry specifying the sink temperature vs time and possibly other variables. When entered as a negative integer its absolute value is a scalar point identification number. If a scalar point is specified on this entry it need not be defined on an SPOINT entry.

Descriptor	Meaning
CHEAT (9,3)/(10,3)	Contact heat transfer coefficient. (Real or Integer; Default = 0.0). If Real, the value entered is the contact heat transfer coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the contact heat transfer coefficient vs temperature or a TABL3D entry specifying the contact heat transfer coefficient vs temperature and possibly other variables.
TBODY (9,4)/(10,4)	Body temperature. (Real or Integer; Default = 0.0). If Real, the value entered is the body temperature. If Integer, the value entered is the ID of a TABLED1 or TABLED2 entry specifying the body temperature vs time or a TABL3D entry specifying the body temperature vs time and possibly other variables. When entered as a negative integer its absolute value is a scalar point identification number. If a scalar point is specified on this entry it need not be defined on an SPOINT entry.
HCV (9,5)/(10,5)	Convection coefficient for near field behavior (Real or Integer; Default = 0.0). If Real the value entered is the near field convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field convection coefficient vs temperature or a TABL3D entry specifying the near field convection coefficient vs temperature and possibly other variables.
HNC (9,6)/(10,6)	Natural convection coefficient for near field behavior (Real or Integer; Default = 0.0). If Real, the value entered is the near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field natural convection coefficient vs temperature or a TABL3D entry specifying the near field natural convection coefficient vs temperature and possibly other variables.
ITYPE [4,8]	An option entry for heat transfer only (Integer; no Default) 1 - Heat sink (rigid body) 2 - Deformable body (with heat conduction) 4 - Heat conduction body (heat-rigid body) (Not supported in SOL 400 coupled thermal-mechanical analysis)
BNC (9,7)/(10,7)	Exponent associated with the natural convection coefficient for near field behavior (Real or Integer; Default = 1.0). If Real, the value entered is the exponent associated with the near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the exponent associated with the near field natural convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the near field natural convection coefficient vs temperature and possibly other variables.
EMISS (9,8)/(10,8)	Emissivity for radiation to the environment or near thermal radiation (Real or Integer; Default = 0.0). If real, the value entered is the emissivity. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the emissivity vs temperature or a TABL3D entry specifying the emissivity vs temperature and possibly other variables.

Descriptor	Meaning
HBL (7,6)/(8,6)	Separation distance dependent thermal convection coefficient (Real or Integer; Default = 0.0). If Real, the value entered is the separation distance dependent thermal convection coefficient. If Integer, the value entered is the ID of a TABLEM1 entry specifying the convection coefficient vs temperature or a TABL3D entry specifying the convection coefficient vs temperature and possibly other variables.
HNL	Heat transfer coefficient for nonlinear convective heat flow for near field behavior. (Real or Integer; Default = 0.0). If Real, the value entered is the near field nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field nonlinear convection coefficient vs temperature or a TABL3D entry specifying the near field nonlinear convection coefficient vs temperature and possibly other variables.
BNL	Exponent associated with the nonlinear convective heat flow for near field behavior. (Real or Integer; Default = 1.0). If Real, the value entered is the exponent associated with the near field nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 OR TABLEM2 entry specifying the exponent associated with the near field nonlinear convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the near field nonlinear convection coefficient vs temperature and possibly other variables.
HNLE	Heat transfer coefficient for nonlinear convective heat flow to the environment. (Real ≥ 0.0 or Integer; Default = 0.0). If Real, the value entered is the environment nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the environment nonlinear convection coefficient vs temperature or a TABL3D entry specifying the environment nonlinear convection coefficient vs temperature and possibly other variables.
BNLE	Exponent associated with the nonlinear convective heat flow to the environment (Real or Integer; Default = 1.0). If Real, the value entered is the exponent associated with the environment nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the exponent associated with the environment nonlinear convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the environment nonlinear convection coefficient vs temperature and possibly other variables.
HNCE	Natural convection coefficient for heat flow to the environment. (Real or Integer, Default = 0.0). If Real the value entered is the environment natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the environment natural convection coefficient vs temperature or a TABL3D entry specifying the environment natural convection coefficient vs temperature and possibly other variables.

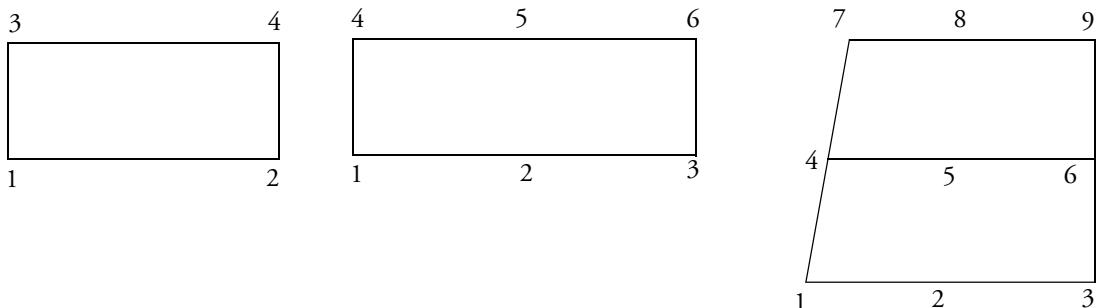
Descriptor	Meaning
BNCE	Exponent associated with natural convection heat flow to the environment. (Real or Integer; Default = 1.0). If Real, the value entered is the exponent associated with the environment natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the exponent associated with the environment natural convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the environment natural convection coefficient vs temperature and possibly other variables.
CMB	Heat capacity of the rigid body, when entered as a geometric entity with an associated scalar point. (Real ≥ 0.0 ; Default = 0.0). For nonzero values a grounded CDAMP4 element with the scalar point on its first side is generated internally which obtains this capacity.
CMS	Heat capacity of the environment, when associated with a scalar point. (Real ≥ 0.0 ; Default = 0.0). For nonzero values a grounded CDAMP4 element with the scalar point on its first side is generated internally which obtains this capacity.
“PATCH3D”	Entries for this continuation line describe a rigid body made up of as many 4-node patches as desired. (Triangular patches are not available.)
IDP	ID of the patch (Integer number 1 through highest value).
G1, G2, G3, G4	Grid numbers for each of the 4 nodes of the patch (see Note 5).
“BEZIER”	Entries for this continuation line describe a rigid body made up of Bezier Surfaces.
NP1	Number of points in 1st direction. (Integer > 0)
NP2	Number of points in 2nd direction. (Integer > 0)
NSUB1	Number of subdivisions in 1st direction. (Integer > 0)
NSUB2	Number of subdivisions in 2nd direction. (Integer > 0)
G1, G2, G3, etc	Grid numbers of each point (must be in order). There must be NP1*NP2 grid points defined. Enter NP1 points for NP2=1, then NP2 points for NP2=2, etc. (Integer)
“NURBS2D”	Entries for this continuation line describe a 2D rigid body made up of nurmbs.
NPTU	Number of control points. If the control points are entered as coordinates rather than grid IDs NPTU may be set to a negative value whose absolute value is the number of xyz coordinates, but that is not required. (Integer, no Default)
NORU	Order
NSUB	Number of subdivisions
G1, G2, G3, G4	Grid numbers for each of the NPTU control points
X1, Y1, X2, Y2, etc.	Alternate method to define control points without using GRID points. There must be abs(NPTU)*NPTV (x,y,z) entries.
Homo1, Homo2, Homo3, etc.	Homogeneous coordinates (0.0 to 1.0) (Real). There must be NPTU entries.

Descriptor	Meaning
Knot1, Knot2, Knot3, etc.	Knot vectors (0.0 to 1.0) (Real). There must be (NPTU+NORU) entries.
"NURBS"	Entries for this continuation line describe a rigid body made up of nurbs.
NPTU	Number of control points in U direction. If the control points are entered as coordinates rather than grid IDs NPTU may be set to a negative value whose absolute value is the number of xyz coordinates, but that is not required. (Integer > 0; Required)
NPTV	Number of control points in V direction. (Integer > 0; Required)
NORU	Order along U direction. (Integer > 0; Required)
NORV	Order along V direction (Integer > 0; Required)
NSUBU	Number of subdivisions in U direction (Integer > 0; Required)
NSUBV	Number of subdivisions in V direction (Integer > 0; Required)
NTRIM	Number of trimming curves (Integer ≥ 0 or blank)
G1, G2, G3, etc.	Grid point IDs defining control points (Integer > 0). There must be NPTU*NPTV entries.
X1, Y1, Z1, X2, Y2, Z2, etc.	Alternate method to define control points without using GRID points. There must be abs(NPTU)*NPTV (x,y,z) entries.
Homo1, Homo2, Homo3, etc	Homogeneous coordinates (0.0 to 1.0). There must be NPTU*NPTV entries. (Real)
Knot1, Knot2, Knot3, etc	Knot vectors (0.0 to 1.0). There must be (NPTU+NORU)+(NPTV+NORV) entries. (Real)
IDtrim	ID of trimming vector. There must be NTRIM of these entries and those entries that follow. (Integer > 0)
NPUTtrim	Number of control points for this trimming vector. (Integer > 0)
NORUtrim	Order for this trimming vector. (Integer > 0)
NSUBtrim	Number of subdivisions for this trimming vector. (Integer > 0)
Xisoparam	First coordinate of point in isoparametric space. (Real)
Ysoparam	Second coordinate of point in isoparametric space. (Real)
Homo1, Homo2, Homo3, etc	Homogeneous coordinates (0.0 to 1.0) of this trimming vector. There must be NPTUtrim entries. (Real)
Knot1, Knot2, Knot3, etc	Knot vectors (0.0 to 1.0) of this trimming vector. There must be NPTUtrim+NORUtrim entries. (Real)

Remarks:

1. Named continuation entries are ignored for a deformable curve or surface (BEHAV=DEFO), except for "HEAT".
2. The grid CGID is the reference grid for the rigid body motion. Loads and enforced motion must be defined in the global coordinate system of CGID.
3. All continuation lines may be omitted if not required.
4. WARNING: For rigid contact, the right hand rule determines the interior side of the rigid surface. A deformable surface which contacts a rigid surface must be on the exterior side of the rigid surface (i.e., in the direction opposite to the right hand rule). If a rigid surface is described backwards, contact will not occur because the deformable body is already inside the rigid body at the start of the analysis. For 3D patches, if all need to be reversed, the parameter PARAM,MARCREVR,1 may be entered to automatically reverse all 3D patches.
5. For BEZIER surfaces, enter np1*np2 points in the order shown below:

Mesh	Normal Order	Reversed Order
2x2	1,2,3,4	2,1,4,3
3x2	1,2,3,4,5,6	3,2,1,6,5,4
3x3	1,2,3,4,5,6,7,8,9	3,2,1,6,5,4,9,8,7



6. For NURBS, enter NPTU grid points G1, G2, G3, etc. (set NPTU to a positive value equal to the number of grid points or enter X1, Y1, Z1, X2, Y2, Z2, etc. coordinates for abs(NPTU) points and set NPTU to a negative value).
7. When IDSPL is greater than 1, these nodes are entered in pairs. For a quad surface (for example, CQUAD4 or edge of a CHEXA) usually 4 sets of nodal pairs are needed to describe the surface. For example, a CQUAD4 with grid numbering 1,2,4,3 would need pairs of nodes, 1,2 2,4 4,3 3,1. The nodal pairs may be entered in any order. See Marc Volume C SPLINE (SOL 400 chapter 9) option documentation for more details.
8. For hard contact, with HGLUE=1 (see BCTABLE for the meaning of HGLUE):
 - a. The temperature of the contacting grid is tied to the temperatures of the contacted element face or the temperature of the rigid geometry when it has a scalar point associated with it.

- b. The temperature of the contacting grid is set to the rigid geometry temperature when it has no scalar point associated with it.

Note: “Glued” thermal contact can result in overshoot of the temperatures at the interface, in particular, if two bodies that have non-uniform initial temperatures are placed in contact. The overshoot effect may be damped somewhat if one uses a near contact distance with some convective heat transfer.

9. For hard contact, with HGLUE=0:

The convective heat flow per unit area over the two interfaces is given by:

$$q = \text{CHEAT} \cdot (T_A - T_B)$$

where T_A is the contacting grid temperature and T_B is the face temperature in the contact point in case of a meshed body or the T_{BODY} temperature in case of a rigid geometry. It is recommended to enter the CHEAT value on the BCTABLE Bulk Data input, since it generally applies to body pairs and is not a property of a single body.

10. For near contact:

$$\begin{aligned} q = & HCV \cdot (T_A - T_B) + \\ & HNC \cdot (T_A - T_B)^{BNC} + \\ & HNL \cdot (T_A^{BNL} - T_B^{BNL}) + \\ & \sigma \cdot EMISS \cdot (T_A^A - T_B^A) + \\ & \left[\text{CHEAT} \cdot \left(1 - \frac{\text{dist}}{DQNEAR} \right) + HBL \cdot \frac{\text{dist}}{DQNEAR} \right] \cdot (T_A \cdot T_B) \end{aligned}$$

where the last term is only activated when $HBL \neq 0$, T_A is the contacting grid temperature and T_B is the face temperature in the contact point in case of a meshed body or the T_{BODY} temperature in case of a rigid geometry. It is recommended to enter the near contact heat transfer coefficients and the corresponding exponents on the BCTABLE Bulk Data input, since they generally apply to body pairs and are not properties of a single body.

11. For no contact:

$$\begin{aligned} q = & CFILM \cdot (T_A - T_{SINK}) + \\ & HNCE \cdot (T_A - T_{SINK})^{BNCE} + \\ & HNLE \cdot (T_A^{BNLE} - T_{SINK}^{BNLE}) + \\ & \sigma \cdot EMISS \cdot (T_A^A - T_{SINK}^A) \end{aligned}$$

12. The heat transfer coefficients and associated exponents can all be temperature dependent, when they are entered as an integer value. This integer value is the table ID of a TABLEM1, TABLEM2 or TABL3D entry (formulas are not supported on TABL3D).

13. The T_{SINK} and T_{BODY} temperatures can be time dependent when they are entered as a positive integer value. This integer value is the table ID of a TABLED1, TABLED2 or TABL3D entry (formulas are not supported on TABL3D).
14. Table IDs of tables used on the BCBODY and the BCTABLE entry must be unique.
15. TBODY entries only apply to rigid bodies (i.e., RIGID as the BEHAV value). TSINK entries only apply to deformable bodies (meshed regions with DEFORM or HEAT as the BEHAV value).

BCBODY**Flexible or Rigid Contact Body in 2D and 3D in SOL 600**

Defines a flexible or rigid contact body in 2D or 3D used in SOL 600 only.

Important Notes for SOL 600:

1. The 2nd line is required if any of the subsequent lines are to be entered (if this line is blank, enter a "+" in column 1 or Nastran will ignore the entire line).
2. PARAM,MRCONVER,11 is required if CTYPE, APPROV, or GROW headers are entered.
3. PARAM,MRCONTAB,11 is required if any of the variables are described by tables.

Use only as many forms (i.e., HEAT, PATCH3D, BEZIER, POLY, CYLIND, SPHERE, NURBS2, or NURBS) as necessary to describe the body (if rigid). Deformable bodies are described using as many standard elements as necessary and are specified by the BSID field with BEHAV=DEFORM (only the first line should be entered for deformable bodies). Unless shrink fit its being analyzed, deformable bodies should not be inside other deformable bodies when the thickness of each body is taken into account.

The "RIGID" header may be used with any of the other rigid entries but only once per body. Also, only one of the character entries after RIGID (HEAT, PATCH3D, NURBS, etc.) should be entered for any particular body. See Remark 4. for an important note regarding how to define the outward direction of rigid bodies (which must point towards a flexible body for contact to occur).

Format:

1	2	3	4	5	6	7	8	9	10
BCBODY	BID	DIM	BEHAV	BSID	ISTYP	FRIC	IDSPL	CONTROL	
	NLOAD	ANGVEL	DCOS1	DCOS2	DCOS3	VELRB1	VELRB2	VELRB3	
"ADVANCE"	SANGLE	COPTB	USER						
"CTYPE"	ISMALL	ITYPE	IAUG	PENALT	AUGDIST				
"RIGID"	CGID	NENT	--- Rigid Body Name ---						
"APPROV"	A	N1	N2	N3	V1	V2	V3		
"RTEMP"	G(temp)	Tempr	T(Tempr)						
"SINK"	G(sink)	Tsink	T(Tsink)						
"GROW"	GF1	GF2	GF3	TAB-GF1	TAB-GF2	TAB-GF3			
"HEAT"	CFILM	TSINK	CHEAT	TBODY	HCV	HNC	ITYPE		
	BNC	EMISS	HBL						
"PATCH3D"	NPATCH								
	IDP	G1	G2	G3	G4				
	IDP	G1	G2	G3	G4				
	etc.	(npatch entries)							
"BEZIER"	NP1	NP2	NSUB1	NSUB2					
	G1	G2	G3	G4	etc			(np1*np2 values)	
"POLY"	NP1	NP2							
	G1	G2	G3	G4	etc			(np1*np2 values)	

	"CYLIND"	NSUB								
		Gtop	Rtop	Gbottom	Rbottom					
	"SPHERE"	NSUB								
		Gcenter	Radius							
	"LINE"	NPTS	Ix	Iy	Rx	Ry		(2D Contact)		
		G1	G2	G3	G4	etc.		(npts values)		
	"ARC"	NPTS	MethArc	Ix	Iy	Rx	Ry	(2D Contact)		
		G1	G2	G3	G4	etc.		(npts values)		
	"SPLINE"	NPTS	Ix	Iy	Rx	Ry		(2D Contact)		
		G1	G2	G3	G4	etc.		(npts values)		
	"NURBS2D"	NPTU	NORU	NSUB	Rx	Ry		(2D Contact)		
		G1 or X1	G2 or Y1	G3	G4 or X2	G5 or Y2	G6	[abs(nptu) grids or x,y,z, values]	See Remark 8	
		Homo1	Homo2	Homo3	Homo4	etc.		(nptu values)		
		Knot1	Knot2	Knot3	Knot4	Knot5	etc.	(nptu+noru values)		
	"NURBS2"	IDN								
	"NURBS"	NPTU	NPTV	NORU	NORV	NSUBU	NSUBV	NTRIM		
		G1 or X1	G2 or Y1	G3 or Z1	G4 or X2	G5 or Y2	G6 or Z2	G7	See Remark 8	
		G8 or X3	G9 or Y3	G10 or Z3	etc.			[abs(nptu)*n ptv values]	See Remark 8	
		Homo1	Homo2	Homo3	Homo4	Homo5	Homo6	Homo7		
		Homo8	Homo9	etc.				(nptu*nptv values)		
		Knot1	Knot2	Knot3	Knot4	Knot5	Knot6	Knot7	'	
		Knot8	Knot9	etc.				(nptu+noru+ ntpv+nrv values)		
		IDtrim	NPTUtrim	NORUtrim	NSUBtrim			(repeat this and all following lines NTRIM times)		
			Xisoparam	Yisoparam				(NPTUtrim entries)		
			Homo1	Homo2	Homo3	etc		(NPTUtrim entries)		
			Knot1	Knot2	Knot3	etc.		(NPTUtrim + NORUtrim entries)		

Examples (of Deformable and Rigid Contact):

Example 1 -- Typical deformable body

BCBODY	1		DEFORM	101	0	.05			
--------	---	--	--------	-----	---	-----	--	--	--

Example 2 -- Simple 4-node rigid patch (see Remark 5 for rigid bodies)

BCBODY	2		RIGID		0			
	PATCH3D	1						
		1	101	102	103	104		

Example 3-- Same as Example 2 except that a user subroutine named motion.f will be employed to specify the rigid body motion.

BCBODY	2		RIGID		0			
	ADVANCE			M				
	PATCH3D	1						
		1	101	102	103	104		

Descriptor	Meaning
BID (4,1)	Contact body identification number referenced by BCTABLE, BCHANGE, or BCMOVE. (Integer > 0; Required)
DIM	Dimension of body. (Character; Default = 3D) DIM=2D planar body in x-y plane of the basic coordinate system, composed of 2D elements or curves. DIM=3D any 3D body composed of rigid surfaces, shell elements or solid elements.
BEHAV (4,8)	Behavior of curve or surface (Character; Default = DEFORM) DEFORM body is deformable, RIGID body is rigid, SYMM body is a symmetry body, ACOUS indicates an acoustic body, WORK indicates body is a workpiece, HEAT indicates body is a heat-rigid body. <i>See Remark 3. for Rigid Bodies..</i>
BSID	Identification number of a BSURF, BCBOX, BCPROP or BCMATL entry if BEHAV=DEFORM. (Integer > 0)

Descriptor	Meaning
ISTYP (4,3)	<p>Check of contact conditions. (Integer ≥ 0; Default = 0)</p> <p>ISTYP is not supported in segment-to-segment contact.</p> <p>For a deformable body:</p> <ul style="list-style-type: none"> =0 symmetric penetration, double sided contact. =1 unsymmetric penetration, single sided contact. (Integer > 0) =2 double-sided contact with automatic optimization of contact constraint equations (this option is known as “optimized contact”). <p>Notes: single-sided contact (ISTYP=1) with the contact bodies arranged properly using the contact table frequently runs much faster than ISTYP=2.</p> <p>For a rigid body:</p> <ul style="list-style-type: none"> =0 no symmetry condition on rigid body. =1 rigid body is a symmetry plane.
FRIC (6,7)	Friction coefficient. (Real ≥ 0 or integer; Default = 0) If the value is an integer it represents the ID of a TABL3Di.
IDSPL (4,5)	<p>Set IDSPL=1 to activate the SPLINE (analytical contact) option for a deformable body and for a rigid contact surface. Set it to zero or leave blank to not have analytical contact. (Integer; Default = 0)</p> <p><u>SOL 600 Options:</u></p> <ul style="list-style-type: none"> =0 or blank, SPLINE option is turned off. =1, The body is smoothed out with splines (2D) or Coons surfaces (3D). If SANGLE is not entered, the default 60.0 degrees is used. IDSPL=1 triggers the Marc SPLINE option with the 3rd field of the 3rd datablock set to 1. If analytical contact changes between increment zero and any subcase, the SANGLE Bulk Data entry is required (see the following SANGLE). SANGLE is placed in the 4th field of the 3rd Mac SPLINE datablock and a value of 1 is placed in the 3rd field of the 3rd datablock. >1, Identification number of a BLSEG entry that lists nodes on edges of the body which are excluded from the SPLINE option. (See Remark 10.) <p>This option may be used for deformable or rigid contact surfaces. See PARAM,MSPLINC0 to enforce C0 continuity.</p>
CONTROL (4,6)	<p>Indicates the type of control for the body.</p> <p>Integer:</p> <ul style="list-style-type: none"> = -1 for position control, = 0 for velocity control. = positive number for “load control” (the positive number is the grid number which has translational forces or SDCP’s are applied. The position of this grid is at the center of rotation given in the CGID field. For velocity controlled surfaces, the velocity of the body must be specified by the VELRBi fields. For displacement controlled surfaces the final displacement can be specified using SPCD at the grid ID specified by CONTROL. CONTROL > 0 is not available for enforced motion using SPCD’s for 2D contact using SOL 600.

Descriptor	Meaning
NLOAD (4,7)	Enter a positive number if “load controlled” and rotations are allowed (Integer). The positive number is the grid number where the moments or rotations are applied. The position of this grid is at the center of rotation given in the CGID field. The rotations are specified using SPCD at grid ID NLOAD and can be specified using dof's 1-3 (for rotation about x, y, z respectively), or starting with the MSC Nastran 2007 release by dof's 4-6 (for rotation about x, y, z respectively). For versions prior to MSC Nastran 2007 only dof's 1-3 could be used.
ANGVEL (6,1)	Angular velocity or angular position about local axis through center of rotation. (Real or Integer; Default = 0.0) If the value is an integer it represents the ID of a TABL3Di.
DCOS1 (6,4)	3D - First component direction cosine of local axis if ANGVEL is nonzero. (Real)
	2D - First coordinate of initial position of rotation of rigid body.
DCOS2 (6,5)	3D - Second component direction cosine of local axis if ANGVEL is nonzero. (Real)
	2D - Second coordinate of initial position of rotation of rigid body
DCOS3 (6,6)	3D - Third component direction cosine of local axis if ANGVEL is nonzero. (Real)
	2D - Not used.
VELRB1 (5,4)	2D & 3D - Velocity or final position (depending on the value of CONTROL) of rigid body in 1st direction. (Real or Integer; Default = 0.0) If the value is an integer it represents the ID of a TABL3Di.
VELRB2 (5,5)	2D & 3D - Velocity or final position (depending on the value of CONTROL) of rigid body in 2nd direction. (Real or Integer; Default = 0.0) If the value is an integer it represents the ID of a TABL3Di.
VELRB3 (5,6)	3D - Velocity or final position (depending on the value of CONTROL) of rigid body in 3rd direction. (Real or Integer; Default = 0.0) If the value is an integer it represents the ID of a TABL3Di.
	2D - Not used.
ADVANCE	The entries for this continuation line are for advanced options starting with MSC Nastran 2007.
SANGLE	Threshold for automatic discontinuity detection in degrees. (Real, Default = 60.0) SANGLE values on the BCBODY entry can be overridden using the SANGLE Bulk Data entry. This allows SANGLE to change for different subcases. It also allows analytical contact for this body to be “on” or “off” for different subcases (see the SANGLE Bulk Data entry). [Marc Spline option (3,4)]
COPTB	Flag to indicate how body surfaces may contact. See Remark 9. on the BCTABLE entry. (Integer; Default = 0)

Descriptor	Meaning
USER	<p>A series of character flags to indicate which (if any) user subroutines are required for this contact body. Enter as many characters as necessary to specify the user subroutines desired, for example to use MOTION and UFIRC enter MF. (Character; Default = blank meaning no user subroutines are required) (SOL 600 Only)</p> <p>M = The MOTION user subroutine is required to describe complex motion of a rigid contact body.</p> <p>F = The UFRIC or UFRICBBC user subroutine is required to describe complex friction behavior of the contact body.</p> <p>C = Film coefficient and sink temperature user subroutine UHTCON is required (coupled structural/heat analysis only)</p> <p>P = 4-Node patch user subroutine DIGEOM is required (the coordinates of the patch are entered in the user subroutine rather than on the BCBODY entry).</p> <p>S = The SEPSTR or SEPFOR user subroutine is required depending on whether stress-based or force-based friction has been specified using BCPARA IBSEP.</p>
“CTYPE”	The entries of this continuation line define whether the original node-to-segment or segment-to-segment contact algorithm is used. Also the selection of general or small sliding contact and associated values when segment-to-segment contact is selected are entered here. These options may be placed on BCPARA instead of BCBODY (which is the recommended option. BCBODY entries override BCPARA entries). For MSC Nastran 2012, these items should be considered beta-test capabilities.
ISMALL [3,1]	Enter 0 for general contact or 1 for small sliding, small displacement contact. See Remarks 15 and 16. (Integer; Default = 0)
ICTYPE [3,2]	Enter 0 for node-to-segment contact or 1 for segment-to-segment contact. See Remarks 15 and 16. (Integer; Default = 0)
IAUG [3,3]	<p>This entry applies only if segment-to-segment contact is selected. See Remarks 15 and 16. (Integer; Default = 0)</p> <p>0 = No augmentation</p> <p>1 = Augmentation is based on a constant Lagrange multiplier field for linear elements and on a bilinear Lagrange multiplier field for quadratic elements.</p> <p>2 = Augmentation is based on a constant Lagrange multiplier field.</p> <p>3 = Augmentation is based on a bilinear Lagrange multiplier field.</p>

Descriptor	Meaning
PENALT [5,1]	Augmented Lagrange penalty factor, used by segment-to-segment contact only. The default is derived from the contact tolerance distance and the stiffness of the deformable contact bodies or average stiffness if two deformable bodies are in contact. See Remark 18. (Real; Default = leave blank)
AUGDIST [5,2]	Penetration distance beyond which an augmentation will be applied, used by segment-to-segment contact only. This default is 1% of the contact tolerance distance. See Remark 18. (Real; Default = leave blank)
"RIGID"	The entries of this continuation line are for the rigid body description. See Remark 3.
CGID (5,i) i=1,2,3 (4,6)	Grid point identification number defining the initial position of the center of rotation for the rigid body or the point where a concentrated force or moment is applied.
NENT (4,2)	Number of geometric entities to describe this rigid surface. A rigid surface can be described by multiple sets of patches, nurbs, etc. For example, if it takes 3 sets of PATCH3D entries to describe a rigid surface, then set NENT=3. (Integer > 0; Default = 1)
Rigid Body Name (4,9)	Name associated with the rigid body. (Default is blank, 24-characters maximum)
"APPROV"	The entries for this continuation line are used for rigid contact body approach velocity and angular velocity. (For SOL 600, APPVEL may be used instead of APPROV if desired.)
A	Angular velocity about local axis through center of rotation. The local axis is defined by DCOS1, DCOS2, DCOS3. (Real; Default = 0.0)
Ni	Not used by SOL 600. The direction cosines for approach velocity in SOL 600 are DCOS1, DCOS2, DCOS3 on the second BCBODY line. (Real; Default = 0.0; Ni values entered on this line are ignored for SOL 600)
Vi	V1, V2 and V3 define the three components of approach velocity. For 2D contact only two should be entered as defined by Rx or Ry (see each 2D rigid contact body description for the meaning of Rx and Ry). (Real; Default = 0.0)
"RTEMP"	The entries of this continuation line describe rigid contact surface temperature variations.
G(Temp)	Grid point identification number defining the location of the temperature. (Integer; no Default)
Tempr	Temperature at the start of the run. (Real; no Default)
T(Tempr)	Identification number of a TABL3Di describing the variation of the rigid body temperature during the analysis. (Integer; Default = 0 which means a constant value Ttempr is used during the entire analysis)

Descriptor	Meaning
“SINK”	The entries of this continuation line describe sink temperatures and their variation for flexible or heat transfer bodies.
G(sink)	Grid point identification number of defining the location of the sink temperature. (Integer; no Default)
Tsink	Sink temperature at the start of the run. (Real; no Default)
T(Tsink)	Identification number of a TABL3Di describing the variation of the sink temperature during the analysis. (Integer; Default = 0 which means a constant value Tsink is used during the entire analysis)
“GROW”	The entries for this continuation line are for rigid body growth - This line is rarely used. If entered PARAM,MRCONVER,11 must be used. If tables are used for growth, they should either be TABLED1 (growth vs time) or TABL3Di (growth vs one or more variables) and PARAM,MRCONTAB,11 must be entered.
GF1 (7,5)	Growth factor of rigid body in first coordinate direction. (Real; Default = 1.0)
GF2 (7,6)	Growth factor of rigid body in second coordinate direction. (Real; Default = 1.0)
GF3 (7,7)	Growth factor of rigid body in third coordinate direction. (Real; Default = 1.0)
TAB-GF1 (8,5)	Table for growth factor of rigid body in first coordinate direction. (Integer or blank; Default is blank which means no table and growth factor varies from 0.0 to 1.0 over the subcase being analyzed.)
TAB-GF2 (8,6)	Table for growth factor of rigid body in second coordinate direction. (Integer or blank; Default is blank which means no table and growth factor varies from 0.0 to 1.0 over the subcase being analyzed.)
TAB-GF3 (8,7)	Table for growth factor of rigid body in third coordinate direction. (Integer or blank; Default is blank which means no table and growth factor varies from 0.0 to 1.0 over the subcase being analyzed.)
“HEAT”	The entries of this continuation line(s) are for contact in heat transfer. Do not enter these line(s) for structural analyses.
CFILM (9,1)/(10,1)	Heat transfer coefficient (film) to environment. (Real or Integer; Default = 0.0 for a heat transfer problem, omit for a structural problem) If Real, the value entered is the film coefficient. If Integer, the value entered is the ID of a TABLEM1 entry specifying the heat transfer coefficient vs temperature. This is usually called HCVE in the Marc documentation.

Descriptor	Meaning
TSINK (9,2)/(10,2)	Environment sink temperature. (Real or Integer; Default = 0.0 for a heat transfer problem, omit for a structural problem). If Real, the value entered is the sink temperature. If Integer, the value entered is the ID of a TABLEM1 entry specifying temperature vs time. At present, this variable should not be a function of temperature.
CHEAT (9,3)/(10,3)	Contact heat transfer coefficient. (Real or Integer; Default = 1.0 for a heat transfer problem, omit for a structural problem). If Real, the value entered is the contact heat transfer coefficient. If Integer, the value entered is the ID of a TABLEM1 entry specifying the contact heat transfer coefficient vs temperature.
TBODY (9,4)/(10,4)	Body temperature. (Real or Integer; Default = 0.0 for a heat transfer problem, omit for a structural problem). If Real, the value entered is the body temperature. If Integer, the value entered is the ID of a TABLEM1 entry specifying the body temperature vs time. At present, this variable should not be a function of temperature.
HCV (9,5)/(10,5)	Convection coefficient for near field behavior (Real or Integer; Default = 1.0 for a heat transfer problem, omit for a structural problem). If Real the value entered is the near field convection coefficient. If Integer, the value entered is the ID of a TABLEM1 entry specifying the near field convection coefficient vs temperature.
HNC (9,6)/(10,6)	Natural convection coefficient for near field behavior (Real or Integer; Default = 1.0 for a heat transfer problem, omit for a structural problem). If Real, the value entered is the near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 entry specifying the near field natural convection coefficient vs temperature.
ITYPE [4,8]	An option entry for heat transfer only (Integer; no Default) 1 - Heat sink 4 - Heat conduction body
BNC (9,7)/(10,7)	Exponent associated with the natural convection coefficient for near field behavior (Real or Integer; Default = 0.0 for a heat transfer problem, omit for a structural problem). If Real, the value entered is the exponent associated with the near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 entry specifying the exponent associated with the near field natural convection coefficient vs temperature. At present, this variable should not be a function of temperature.
EMISS (9,8)/(10,8)	Emissivity for radiation to the environment or near thermal radiation (Real or Integer; Default = 0.0 for a heat transfer problem, omit for a structural problem). If real, the value entered is the emissivity. If Integer, the value entered is the ID of a TABLEM1 entry specifying the emissivity vs temperature.

Descriptor	Meaning
HBL (11,1)[12,1]	Separation distance dependent thermal convection coefficient (Real or Integer; Default = 0.0 for a heat transfer problem, omit for a structural problem). If Real, the value entered is a convection coefficient. If Integer, the value entered is the ID of a TABLEM1 entry specifying the convection coefficient vs temperature.
“PATCH3D”	Entries for this continuation line describe a rigid body made up of as many 4-node patches as desired. (Triangular patches are not available.)
IDP	ID of the patch (Integer number 1 through highest value).
G1, G2, G3, G4	Grid numbers for each of the 4 nodes of the patch (see Note 5).
“BEZIER”	Entries for this continuation line describe a rigid body made up of Bezier Surfaces.
NP1	Number of points in 1st direction. (Integer > 0)
NP2	Number of points in 2nd direction. (Integer > 0)
NSUB1	Number of subdivisions in 1st direction. (Integer > 0)
NSUB2	Number of subdivisions in 2nd direction. (Integer > 0)
G1, G2, G3, etc	Grid numbers of each point (must be in order). There must be NP1*NP2 grid points defined. Enter NP1 points for NP2=1, then NP2 points for NP2=2, etc. (Integer)
“POLY”	Entries for this continuation line describe a rigid body made up of Poly Surfaces.
NP1	Number of points in the 1st direction. (Integer > 0)
NP2	Number of points in the 2nd direction. (Integer > 0)
G1, G2, G3, etc	Grid numbers of each point (must be in order). There must be NP1*NP2 grid points defined. Enter NP1 points for NP2=1, then NP2 points for NP2=2, etc. (Integer)
“CYLIND”	Entries for this continuation line describe a cylindrical rigid body.
NSUB	Number of subdivisions. (Integer > 0)
Gtop	Grid point ID of a grid in the center of the top of the cylinder. (Integer > 0)
Rtop	Radius of the top of the cylinder. (Real > 0.0)
Gbottom	Grid point ID of a grid in the center of the bottom of the cylinder. (Integer > 0)
Rbottom	Radius of the bottom of the cylinder. (Real > 0.0)
“SPHERE”	Entries for this continuation line describe a spherical rigid body.
NSUB	Number of subdivisions. (Integer > 0)
Gcenter	Grid point ID of a grid in the center of the sphere. (Integer > 0)
Radius	Radius of the sphere. (Real > 0.0)
“LINE”	Entries for this continuation line describe a 2D rigid body made up of as many line segments as desired.

Descriptor	Meaning
NPTS	Number of points in the line segment.
Ix	Grid direction (1, 2, or 3) of the G1, G2, ... list to be used as the first coordinate in Marc's 2D line description. See Remark 12. (Integer; Default = 1).
Iy	Grid direction (1, 2, or 3) of the G1, G2, ... list to be used as the second coordinate in Marc's 2D line description. See Remark 12. (Integer; Default = 2).
Rx	Direction (1, 2, or 3) used as the first VELRBi in Marc's 2D contact description. (Integer; Default = 1)
Ry	Direction (1, 2, or 3) used as the second VELRBi in Marc's 2D contact description. (Integer; Default = 2)
G1, G2, G3, G4	Grid numbers for each of the NPTS points on the line segment.
"ARC"	Entries for this continuation line describe a 2D rigid body made up of as many segments as desired describing an arc.
NPTS	Number of points in the arc. NPTS must be 4 for an ARC.
MethArc	Method to generate arc (Integer 0 to 4) (see Marc volume C Contact description (SOL 400 chapter 9) Figures 3-3 and 3-4).
Ix	Grid direction (1, 2, or 3) of the G1, G2, ... list to be used as the first coordinate in Marc's 2D line description. See Remark 12. (Integer; Default = 1).
Iy	Grid direction (1, 2, or 3) of the G1, G2, ... list to be used as the second coordinate in Marc's 2D line description. See Remark 12. (Integer; Default = 2).
Rx	Direction (1, 2, or 3) used as the first VELRBi in Marc's 2D contact description. (Integer; Default = 1)
Ry	Direction (1, 2, or 3) used as the second VELRBi in Marc's 2D contact description. (Integer; Default = 2)
G1, G2, G3, G4	Grid numbers for each of the 4 points as described by method.
"SPLINE"	Entries for this continuation line describe a 2D rigid body made up of as many spline segments as desired.
NPTS	Number of points for the spline.
Ix	Grid direction (1, 2, or 3) of the G1, G2, ... list to be used as the first coordinate in Marc's 2D line description. See Remark 12. (Integer; Default = 1).
Iy	Grid direction (1, 2, or 3) of the G1, G2, ... list to be used as the second coordinate in Marc's 2D line description. See Remark 12. (Integer; Default = 2).
Rx	Direction (1, 2, or 3) used as the first VELRBi in Marc's 2D contact description. (Integer; Default = 1)
Ry	Direction (1, 2, or 3) used as the second VELRBi in Marc's 2D contact description. (Integer; Default = 2)
G1, G2, G3, G4	Grid numbers for each of the NPTS points on the spline.

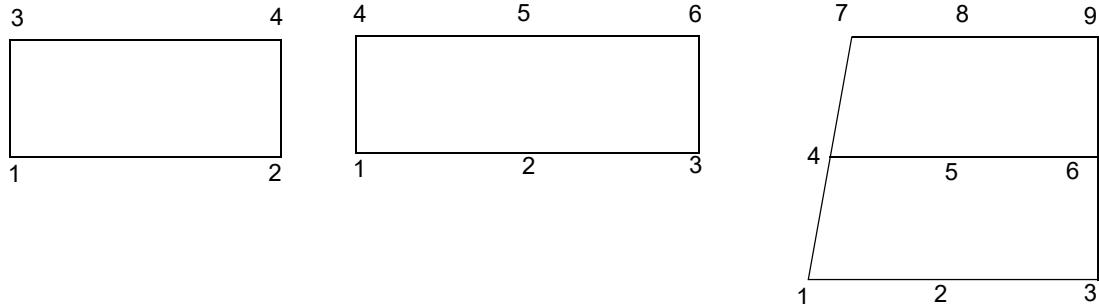
Descriptor	Meaning
"NURBS2D"	Entries for this continuation line describe a 2D rigid body made up of nurmsbs.
NPTU	Number of control points. If the control points are entered as coordinates rather than grid IDs NPTU may be set to a negative value whose absolute value is the number of xyz coordinates, but that is not required. (Integer; no Default)
NORU	Order
NSUB	Number of subdivisions
Rx	Direction (1, 2, or 3) used as the first VELRBi in Marc's 2D contact description. (Integer; Default = 1)
Ry	Direction (1, 2, or 3) used as the second VELRBi in Marc's 2D contact description. (Integer; Default = 2)
G1, G2, G3, G4	Grid numbers for each of the NPTU control points
X1, Y1, X2, Y2, etc.	Alternate method to define control points without using GRID points. There must be $\text{abs}(\text{NPTU}) * \text{NPTV}$ (x,y,z) entries.
Homo1, Homo2, Homo3, etc.	Homogeneous coordinates (0.0 to 1.0) (Real). There must be NPTU entries.
Knot1, Knot2, Knot3, etc.	Knot vectors (0.0 to 1.0) (Real). There must be (NPTU+NORU) entries.
"NURBS2"	Entries for this continuation line describe a rigid body made up of nurbs.
IDN	ID of a matching GMNURB entry. The GMNURB is an entry that contains the same information as at shown for the NURBS option. (Integer > 0)
"NURBS"	Entries for this continuation line describe a rigid body made up of nurbs.
NPTU	Number of control points. If the control points are entered as coordinates rather than grid IDs NPTU may be set to a negative value whose absolute value is the number of xyz coordinates, but that is not required. (Integer; no Default)
NPTV	Number of control points in V direction. (Integer > 0; Required)
NORU	Order along U direction. (Integer > 0; Required)
NORV	Order along V direction (Integer > 0; Required)
NSUBU	Number of subdivisions in U direction (Integer > 0; Required)
NSUBV	Number of subdivisions in V direction (Integer > 0; Required)
NTRIM	Number of trimming curves (Integer ≥ 0 or blank)
G1, G2, G3, etc.	Grid point IDs defining control points (Integer > 0). There must be $\text{NPTU} * \text{NPTV}$ entries.
X1, Y1, Z1, X2, Y2, Z2, etc.	Alternate method to define control points without using GRID points. There must be $\text{abs}(\text{NPTU}) * \text{NPTV}$ (x,y,z) entries.

Descriptor	Meaning
Homo1, Homo2, Homo3, etc	Homogeneous coordinates (0.0 to 1.0). There must be NPTU*NPTV entries. (Real)
Knot1, Knot2, Knot3, etc	Knot vectors (0.0 to 1.0). There must be (NPTU+NORU)+(NPTV+NORV) entries. (Real)
IDtrim	ID of trimming vector. There must be NTRIM of these entries and those entries that follow. (Integer > 0)
NPUTtrim	Number of control points for this trimming vector. (Integer > 0)
NORUtrim	Order for this trimming vector. (Integer > 0)
NSUBtrim	Number of subdivisions for this trimming vector. (Integer > 0)
Xisoparam	First coordinate of point in isoparametric space. (Real)
Ysoparam	Second coordinate of point in isoparametric space. (Real)
Homo1, Homo2, Homo3, etc	Homogeneous coordinates (0.0 to 1.0) of this trimming vector. There must be NPTUtrim entries. (Real)
Knot1, Knot2, Knot3, etc	Knot vectors (0.0 to 1.0) of this trimming vector. There must be NPTUtrim+NORUtrim entries. (Real)

Remarks:

- Named continuation entries are ignored for a deformable curve or surface (BEHAV=DEFO), except for "HEAT".
- The grid CGID is the reference grid for the rigid body motion. Loads and enforced motion must be defined in the global coordinate system of CGID.
- WARNING: For rigid contact, the right hand rule determines the interior side of the rigid surface. A deformable surface which contacts a rigid surface must be on the exterior side of the rigid surface (i.e., in the direction opposite to the right hand rule). If a rigid surface is described backwards, contact will not occur because the deformable body is already inside the rigid body at the start of the analysis. For 3D patches, if all need to be reversed, the parameter PARAM,MARCREVR,1 may be entered to automatically reverse all 3D patches.
- (i,j) refers to data block i and field j of Marc's CONTACT model definition entry. IDSPL covers the SPLINE history definition in Marc. For structural analysis (i,j) refers to contact without tables. For heat transfer (i,j) refers to contact with tables.
- For BEZIER surfaces, enter np1*np2 points in the order shown:

Mesh	Normal Order	Reversed Order
2x2	1,2,3,4	2,1,4,3
3x2	1,2,3,4,5,6	3,2,1,6,5,4
3x3	1,2,3,4,5,6,7,8,9	3,2,1,6,5,4,9,8,7



6. For NURBS, enter NPTU grid points G1, G2, G3, etc. (set NPTU to a positive value equal to the number of grid points or enter X1, Y1, Z1, X2, Y2, Z2, etc. coordinates for abs(NPTU) points and set NPTU to a negative value.
7. The heat transfer options are available for SOL 600 starting with MSC Nastran 2005 r2 and must use Marc 2005 or later.
8. For heat transfer items described using a TABLEM1 ID, the smallest value in the table will be entered into Marc's 9th contact (with tables) datablock. The table ID will be translated directly to Marc's 10th contact (with tables) datablock.
9. All flexible surfaces must have smaller BID values than the rigid surfaces so that in the Marc file all flexible surfaces are defined prior to all rigid surfaces. This is a Marc limitation.
10. When IDSPL is greater than 1, these nodes are entered in pairs. For a quad surface (for example, CQUAD4 or edge of a CHEXA) usually 4 sets of nodal pairs are needed to describe the surface. For example, a CQUAD4 with grid numbering 1,2,4,3 would need pairs of nodes, 1,2 2,4 4,3 3,1. The nodal pairs may be entered in any order. See Marc Volume C SPLINE (SOL 400 chapter 9) option documentation for more details.
11. With NURBS2D, the coordinates x1, x2, y1, y2 etc. or those given by G1, G2, etc. will be reversed (Y becomes X and X becomes Y) if any CQUADX, CTRIA6 or CTRIA6 entries are found in the bulk data. It is up to the user to ensure that the rigid contact surface is orientated in the correct direction after this reversal. If the user does not want the reversal to happen, enter PARAM,MARCREVX,-1 in the bulk data.
12. Ix and Iy for LINE, ARC, SPLINE and NURBS2D must be the same for all entries in the model. An example would be if CTRIA6 is defined in the Z-X plane, set Ix = 3 and Iy = 1 for all 2D rigid surfaces in the model. For CTRIA6 and CQUADX, normally Ix=2 and Iy=1.
13. Rx and Ry for LINE, ARC, SPLINE and NURBS2D must be the same for all entries in the model. An example would be if CTRIA6 is defined in the Z-X plane, set Rx = 3 and Ry = 1 for all 2D rigid surfaces in the model. For CTRIA6 and CQUADX, normally Rx=2 and Ry=1.
14. For SOL 600, tables using integers for ANGVEL, VELRB1, VELRB2 and/or VELRB3 require PARAM,MRCONVER,11 and PARAM,MRACTOTT,1 to be set. the TABLE3D's used must define the entire motion/velocity desired.
15. If any CTYPE entries are used, PARAM,MRCONVER,11 and PARAM,MRCONVER,11 are required.

16. ISMALL, ITYPE, IAUG should be the same for all BCTABLE entries. The first entry “CTYPE” entry will be used for all bodies. PENALT and AUGDIST may vary from body to body.
17. 2D and 3D rigid bodies may not be mixed in the same input file.
18. The entries on the CTYPE header may optionally be specified on the BCPARA entry.

Correlation between BCBODY “CTYPE” entries and equivalent BCPARA entries:

BCPARA	BCPARA	CTYPE value(s)
METHOD	NODESURF	ictype=0, ismall=0
METHOD	NODSMALL*	ictype=0, ismall=1
METHOD	SEGSMALL	ictype=1, ismall=1
METHOD	SEGMENT*	ictype=1, ismall=1
METHOD	SEGLARGE*	ictype=1, ismall=0
AUTMENT	Integer Value	IAUG=Integer Value
PENALT	Real Value	PENALT=Real Value
AUGDIST	Real Value	AUTDIST=Real Value

* Not available in SOL 600.

19. Most continuation lines may be omitted if not required except that the second line is required if the third or subsequent lines are to be entered.

BCBODY1

Flexible or Rigid Contact Body in 2D and 3D (New Form)

Defines a flexible or rigid contact body in 2D or 3D used in SOLs 101, 400, and 700 only.

If the body is rigid, this entry may refer to BCRIGID entry by the BCRGID field to describe the body. See Remark 1. for an important note regarding how to define the outward direction of rigid bodies (which must point towards a flexible body for contact to occur).

Deformable bodies are described using as many standard elements as necessary and are specified by the BSID field with BEHAV=DEFORM. Unless shrink fit or interference fit is being analyzed, deformable bodies should not be inside other deformable bodies when the thickness of each body is taken into account.

Format: (SOLs 101 and 400 only)

1	2	3	4	5	6	7	8	9	10
BCBODY1	BID	BPID	DIM	BEHAV	BSID	BCRGID			

Format: (SOL 700 only)

1	2	3	4	5	6	7	8	9	10
BCBODY1	BID				BSID				

Examples of Deformable and Rigid Contact:

Example 1 -- Typical deformable body

BCBODY1	94	3001		DEFORM	103				
---------	----	------	--	--------	-----	--	--	--	--

Example 2 -- Simple rigid patch (see Remark 1. for rigid bodies)

BCBODY1	6	3001		RIGID	206	115			
---------	---	------	--	-------	-----	-----	--	--	--

Descriptor **Meaning**

BID Unique contact body identification number referenced by BCNECT, BCHANGE, or BCMOVE. (Integer > 0; Required)

BPID Parameter identification number of a BCBDPRP entry. (Integer > 0 or blank) Ignored in SOL 700. See Remark 2.

DIM Dimension of body. (Character; Default= 3D) Ignored in SOL 700.
DIM=2D: planar body in x-y plane of the basic coordinate system, composed of 2D elements or curves.

DIM=3D: any 3D body composed of rigid surfaces, shell elements or solid elements.

BEHAV Behavior of curve or surface (Character; Default = DEFORM) Ignored in SOL 700.
DEFORM body is deformable; RIGID body is rigid (See Remark 1.); SYMM body is a symmetry rigid body; HEAT indicates body is a heat-rigid body (See Remark 3.).

BSID	For SOLs 101 and 400: Identification number of a BSURF or BCPROP entry if BEHAV=DEFORM or HEAT, or identification number of a BCRGSRF, BCPATCH, BCBZIER, BCNURB2, or BCNURBS entry if BEHAV=RIGID or SYMM (See Remark 4.). For SOL 700: Identification number (RBID) of a BSURF, BCBOX, BCPROP, BCMATL, BCSEG, BCGRID or BCELIPS entry. (Integer > 0).
BCRGID	For SOLs 101 and 400: Identification number of a BCRIGID entry if BEHAV=RIGID or SYMM. Ignored in SOL 700. (Integer > 0)

Remarks:

1. WARNING: For rigid contact, the right hand rule determines the interior side of the rigid surface. A deformable surface which contacts a rigid surface must be on the exterior side of the rigid surface (i.e., in the direction opposite to the right hand rule). If a rigid surface is described backwards, contact will not occur because the deformable body is already inside the rigid body at the start of the analysis. For 3D patches, if all need to be reversed, the parameter PARAM,MARCREVR,1 may be entered to automatically reverse all 3D patches
2. If BPID field is blank, then default values are set for the parameters defined in BCBDPRP entry.
3. For pure thermal analysis, BEHAV=DEFORM and BEHAV=HEAT are identical. However, in coupled thermal-mechanical analysis, BEHAV=DEFORM specifies a deformable in mechanical analysis part, while BEHAV=HEAT defines a rigid meshed body in mechanical analysis part.
4. If BCRGSRF entry does not exist, BCBODY1 entry can refer to one of the identification number of BCPATCH, BCBZIER, BCNURB2, or BCNURBS entry directly (RBID).

BCBOX**3D Contact Region**

Defines a 3D contact region - all elements within the region define a contact body in SOL 600 and 700.

Format (Form 1):

1	2	3	4	5	6	7	8	9	10
BCBOX	ID		HOW						
	N1	N2	N3	N4	N5	N6	N7	N8	

Form 2:

BCBOX	ID	COORD	HOW						
			X1	Y1	Z1	X2	Y2	Z2	
			X3	Y3	Z3	X4	Y4	Z4	
			X5	Y5	Z5	X6	Y6	Z6	
			X7	Y7	Z7	X8	Y8	Z8	

Example (for Form 1):

BCBOX	101		0						
	1001	1002	1003	1004	1005	1006	1007	1008	

Descriptor	Meaning
ID	Identification of a deformable surface corresponding to a BSID value on the BCBODY entry if the Case Control BCONTACT=BCBOX is specified. All elements corresponding to the designated box may potentially come into contact. See Remark 1. (Integer > 0)
COORD	Enter COORD in field 3 if x,y,z coordinates of the box are to be specified rather than grid IDs. (Character)
HOW	A flag indicating whether an element is in the defined box or not. (Integer; Default = 0)
	0 If only one grid point of an element is in the box, the entire element is considered to be in the box.
	1 All grid points comprising the element must be within the box, otherwise the element is considered outside of the box.
N1-N8	Enter 8 Grid IDs defining a box (hexa-like region) if the third field is blank. (Integer; Required if COORD is blank)
Xi, Yi, Zi	Enter eight x,y,z values in the basic coordinate system if the third field is COORD. See Remark 6. (Real; Required if "COORD" is entered in field 3 of line 1)

Remarks:

1. ID must be unique with respect to all other BSURF, BCBOX, BCPROP, and BCMATL entries.
2. The deformable surface may alternately be defined using BSURF, BCPROP, or BCMATL entries.
3. Only one kind of entry (BSURF, BCBOX, BCPROP, or BCMATL) may be used to define a particular deformable surface.
4. All elements corresponding to the IDs entered will be used to define the deformable surface.
5. The model is searched to determine whether each element lies within the specified box region as specified by the HOW criteria option.
6. For SOL 700, the BCBOX allows a general box and edges do not need to be aligned with the coordinates axis. The eight grid points define the box identical to the CHEXA grid point numbering.
7. BCBOX is not supported in SOLs 101 and 400.

BCBZIER

Defines a Rigid Contact Body Made up of Bezier Surfaces in SOLs 101 and 400

Defines a rigid contact body made up of Bezier surfaces used in SOLs 101 and 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
BCBZIER	RBID	NP1	NP2	NSUB1	NSUB2				
	G1	G2	G3	G4	-etc.-				

Example:

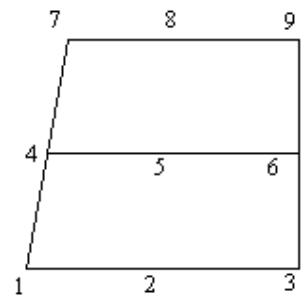
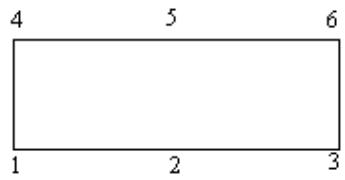
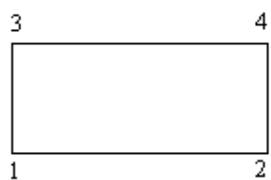
BCBZIER	98	3	3	25	15				
	21	11	1	84	74	64	147	137	
	127								

Descriptor	Meaning
RBID	Unique identification number referenced by a BCRGSRF or BCBODY1 entry. (Integer > 0) See Remark 1.
NP1	Number of points in 1st direction. (Integer > 0)
NP2	Number of points in 2nd direction. (Integer > 0)
NSUB1	Number of subdivisions in 1st direction. (Integer > 0)
NSUB2	Number of subdivisions in 2nd direction. (Integer > 0)
G1, G2, G3, etc.	Grid numbers of each point (must be in order). There must be NP1*NP2 grid points defined. Enter NP1 points for NP2=1, then NP1 points for NP2=2, etc. (Integer > 0)

Remarks:

1. If BCRGSRF entry does not exist, BCBZIER entry will be referenced by the [BCBODY1](#) entry directly.
2. For BEZIER surfaces, enter np1*np2 points in the order shown below

Mesh	Normal Order	Reversed Order
2x2	1,2,3,4	2,1,4,3
3x2	1,2,3,4,5,6	3,2,1,6,5,4
3x3	1,2,3,4,5,6,7,8,9	3,2,1,6,5,4,9,8,7



BCELIPS**Contact Ellipsoid List- SOL 700**

Defines a list of ellipsoid names (character strings) for use of contact analysis. Used in SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
BCELIPS	CID	NAME1	NAME2	NAME3	NAME4	NAMEi	etc		

Example:

BCELIPS	1	HUB	RIM	HEAD	CHEST				
---------	---	-----	-----	------	-------	--	--	--	--

Field	Content
CID	Unique identification number of ellipsoid names which can be used for contact. (Integer > 0; required)
NAME	Name of an ellipsoid. (Character; required)

Remarks:

Use as many continuations as required to define the complete list of names. A blank field terminates the list..

BCGRID (SOL 700 only)**Contact Node Region**

Grids to be included in contact analyses in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
BCGRID	CID	GID1	GID2	GID3	GID4	GID5	GID6	GID7	
	GID8	GID9	GID10	GID11	GID12	GID13	-etc.-		

Example:

BCGRID	100	12	14	17	121	234	235	270	
	309	1001	THRU	2000	BY	2			

Descriptor	Meaning
ID	Unique identification number of a “cloud” of grid points which can be used for contact or RCONN, BJOIN or WALL. (Integer > 0; Required)
GID1, GID2,...	Gridpoint ID. THRU indicates a range to be used. BY is the increment to be used within this range. (Integer > 0; Required)

Remarks:

1. ID must be unique with respect to all other BSURF, BCBOX, BCPROP, BCMATL, or BCSEG entries.
2. BCGRID may only be used for SLAVE body definitions on the BCTABLE entry.
3. As many continuation lines as necessary may be used to define all GRID points used in the definition.

BCGRID

Contact region by a list of grid points.

Define a contact region by a grid point list, for SOLs 101, 103, 105, 107, 108, 109, 110, 111, 112, 200 and 400.

Format:

1	2	3	4	5	6	7	8	9	10
BCGRID	BID	BPID	DIM						
	GID1	GID2	GID3	"THRU"	GID4	"BY"	GID5	GID6	
	GID7								

Example:

BCGRID	2	3	3D						
	12	21	THRU	101	3	6			

Descriptor	Meaning
BID	Unique contact face identification number referenced by BCNECT (Integer > 0; Required). See remark 1.
BPID	Parameter identification number of a BCBDRP entry. (Integer > 0 or blank)
DIM	Dimension of the contact entry. (Character; Default= "3D"), "3D" or "2D".
GIDi	Grid point ids. (Integer > 0)

Remarks:

1. BID must be unique among all BCSURF, BCBODY1 and BCGRID entries.
2. BCGRID entries are able to coexist with BCBODY1 entries, however a BCGRID can only be used to construct a contact pair with a BCSURF in a BCNECT, it is must be a slave and the BCSURF must be a master.
3. This entry only works in node to segment method.
4. The identification of BCGRID cannot be specified in the continuations of a BCNECT entry.

BCHANGE**Changes Definitions of Contact Bodies**

Changes definitions of contact bodies used in SOL 101 and SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
BCHANGE	ID	TYPE			IDBOD1	N1	N2	INC	
	IDBOD2	N1	N2	INC	IDBOD3	etc.			

Example:

BCHANGE	201	NODE			1	1001	1010	1	
	2	2001	2021	2					

Descriptor	Meaning
ID	Identification number referenced by a BCONTACT or BCHANGE Case Control command within a SUBCASE or STEP. See Remark 1. (Integer ≥ 0 ; Required)
Type	Type of modification (Character; Required)
	NODE Defines nodes of a contact body which may come into contact.
	EXCLUDE Excludes 2 node segments in 2D or 4 node patches in 3D.
IDBODi	Identification number of a contact body, BCBODY or BCBODY1 entry. (Integer > 0)
N1	Starting grid ID. More than one N1-N2-INC range may be entered for each body, see Remark 7. (Integer > 0 ; Default = 1)
N2	Ending grid ID.
INC	Grid ID increment.

Remarks:

1. To place an entry in the loadcase 0, set ID=0, which does not need any corresponding Case Control command BCONTACT=0 or BCHANGE=0, and it is always executed automatically. To place an entry in any physical loadcase (SUBCASE or STEP), the ID must be selected by the Case Control command BCONTACT=ID or BCHANGE=ID. Note that if BCHANGE Case Control command exists, it always dominates the selection of BCHANGE Bulk Data entries.
2. The BCHANGE entry does not apply to rigid bodies. Multiple BCHANGE entries are allowed. A body may be entered more than once with different grid IDs.
3. BCHANGE is useful only for saving computer time and is not recommended for general usage.
4. Warning -- For the NODE option, if some nodes in a body are inadvertently omitted, they may penetrate other bodies.
5. BCHANGE with the NODE option is not supported in Segment-to-segment contact.

6. NODE and EXCLUDE may not be used simultaneously in the same BCHANGE entry.
7. If TYPE=NODE, the form of N1-N2-INC range has the following rules:
 - a. The format of INC is either blank or integer (≥ 0).
 - b. N1-N2-0 or N1-N2-blank represents 2 nodes (N1,N2) where $0 < N1$ and $0 < N2$,
 - c. N1-N2-INC, INC>0, represent a range input but $0 < N1 < N2$.
8. If TYPE=EXCLUDE, the form of N1-N2-INC range has the following rules:
 - a. For node segment, N1-N2-INC is used. If $0 < N1 < N2$ and INC>0, the range is applied. If INC-blank or 0, it is 2 nodes (N1,N2) input that can be in any order.
 - b. For 4 nodes patch, (N1,N2,N3,N4), 2 sets of range, which have to input in sequence, are required.
(IDBOD1,-N1,N2,INC1) and (IDBOD2,N3,N4,INC2) where IDBOD1=IDBOD2, Ni>0, INC1 and INC2 are ignored.

The following example is for a 4-node patch (100,110,300,200)

1	2	3	4	5	6	7	8	9	10
BCHANGE	1	EXCLUDE			1	-100	110		
	1	300	200						

- c. For 3 nodes patch, (N1,N2,N3), 2 sets of range, which have to input in sequence, are required
(IDBOD1,-N1,N2,INC1) and (IDBOD2,N3,N3,INC2)
where IDBOD1=IDBOD2,Ni>0, INC1 and INC2 are ignored.

The following example is for a 3-noded patch (132,97,95)

1	2	3	4	5	6	7	8	9	10
BCHANGE	1	EXCLUDE			1	-132	97		
	1	95	95						

BCHANGE**Changes Definitions of Contact Bodies in SOL 600**

Changes definitions of contact bodies used in SOL 600.

Format:

1	2	3	4	5	6	7	8	9	10
BCHANGE	ID	TYPE	NBOD		IDBOD1	N1	N2	INC	
	IDBOD2	N1	N2	INC	IDBOD3	etc.			

Example:

BCHANGE	201	NODE	2		1	1001	1010	1	
	2	2001	2021	2					

Descriptor	Meaning
ID	Identification number referenced by a SUBCASE Case Control command. See Remark 2. (Integer ≥ 0 ; Required)
Type	Type of modification (Character; Required)
	NODE Defines nodes of a contact body which may come into contact.
	EXCLUDE Excludes 2 node segments in 2D or 4 node patches in 3D.
NBOD	Number of bodies to be modified -- must match number of bodies actually entered. More than one N1-N2-INC range may be entered for each body, see Remark 7. See N1 below. (Integer > 0 ; Default = 1)
IDBODi	Identification number of a contact body, BCBODY entry. (Integer > 0)
N1	Starting grid ID.
N2	Ending grid ID.
INC	Grid ID increment.

Remarks:

1. To place an entry in Marc's phase 0, set ID=0. To activate the entry for the first SUBCASE, SET ID=1, for the 2nd, set ID=2.
2. The BCHANGE entry does not apply to rigid bodies. Multiple BCHANGE entries are allowed. A body may be entered more than once with different grid IDs.
3. The BCHANGE entry covers Marc's history definitions CONTACT NODE and EXCLUDE.
4. BCHANGE is useful only for saving computer time and is not recommended for general usage.
5. Warning -- For the NODE option, if some nodes in a body are inadvertently omitted, they may penetrate other bodies.

6. If more than one N1-N2-INC range is required for a body, enter N1 as a negative value for all ranges except for the last range for which N1 is entered as a positive value.
7. If more than one N1-N2-INC range is required, all ranges with IBOD1 must come first, followed by all with IBOD2, etc.
8. The EXCLUDE option is obsolete. The reasons why it was added in the past have been alleviated by better improved contact algorithms. If the EXCLUDE option is entered for 3D shapes two N1-N2-INC ranges are normally required to define all 4 nodes of a patch. If the patch is triangular, the last two nodes must be repeated. The following are examples of how data are entered for one element using the exclude option. The first example is for a 4-node patch (nodes 100, 110, 200, 300) and the second is for a 3-node patch (nodes 132, 97, 95, 95).

BCHANGE	1	EXCLUDE	1		1	-100	110	10	
	1	200	300	100					

BCHANGE	1	EXCLUDE	1		1	-132	132	0	
	1	-97	97	0	1	-95	95	0	
	1	95	95	0					

BCMATL**3D Contact Region by Element Materials**

Defines a 3D contact region by element material. All elements with the specified materials define a contact body used in SOL 600 and SOL 700.

Format:

1	2	3	4	5	6	7	8	9	10
BCMATL	ID	IM1	IM2	IM3	IM4	IM5	IM6	IM7	
	IM8	IM9	etc.						

Example:

BCMATL	1001	101	201	301					
--------	------	-----	-----	-----	--	--	--	--	--

Descriptor	Meaning
ID	Identification of a deformable surface corresponding to a BSID value on the BCBODY entry if the Case Control command, BCONTACT=BCMATL is specified. All elements corresponding to the material IDs specified may potentially come into contact. See Remark 1. (Integer > 0)
IMi	Material ID. A minimum of one entry is required. (Integer)

Remarks:

1. ID must be unique with respect to all other BSURF, BCBOX, BCPROP, and BCMATL entries.
2. The deformable surface may alternately be defined using BSURF, BCBOX, or BCPROP entries.
3. Only one kind of entry (BSURF, BCBOX, BCPROP, or BCMATL) may be used to define a particular deformable surface.
4. All elements corresponding to the IDs entered will be used to define the deformable surface.
5. As many continuation lines as necessary may be used to define all material IDs associated with a particular deformable body.
6. BCMATL may not be used to define contact regions made up of composite elements.

BCMOVE**Movement of Bodies in Contact**

Defines movement of bodies in contact used in SOL 101 and SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
BCMOVE	ID	MTYPE							
	IDRBOD1	IDRBOD2	IDRBOD3	etc.					

Examples:

BCMOVE	33	RELEASE							
	1	3	5	7					
BCMOVE	1	approach							

Descriptor	Meaning
------------	---------

ID	Identification number referenced by a BCONTACT or BCMOVE Case Control command within a SUBCASE or STEP. See Remark 1. (Integer ≥ 0 ; Required)
----	---

MTYPE	Movement type. (Character; Default = APPROACH)
-------	--

APPROACH	All rigid bodies are moved so that they all make contact with deformable bodies.
----------	--

RELEASE	The contact condition is released for selected bodies.
---------	--

SYNCHRON	All rigid bodies are moved until the first rigid body makes contact with a deformable body.
----------	---

IDRBODi	Identification numbers of contact bodies to be released, for option RELEASE only. Points to BCBODY or BCBODY1 Bulk Data entries.
---------	--

Remarks:

1. To place an entry in the loadcase 0, set ID=0, which does not need any corresponding Case Control command BCONTACT=0 or BCMOVE=0, and it is always executed automatically. To place an entry in any physical loadcase (SUBCASE or STEP), the ID must be selected by the Case Control command BCONTACT=ID or BCMOVE=ID. Note that if BCMOVE Case Control command exists, it always dominates the selection of BCMOVE Bulk Data entries. ID must be unique (only one BCMOVE per SUBCASE).
2. For MTYPE=APPROACH and MTYPE=SYNCHRON leave all following fields blank.
3. The APPROACH and SYNCHRON options apply to rigid contact surfaces only.
4. You may release a deformable body from contact with either a deformable or rigid body.

BCMOVE**Movement of Bodies in Contact in SOL 600**

Defines movement of bodies in contact to be used in SOL 600.

Format:

1	2	3	4	5	6	7	8	9	10
BCMOVE	ID	MTYPE	IREL						
	IDRBOD1	IDRBOD2	IDRBOD3	etc.					

Examples:

BCMOVE	33	RELEASE	20						
	1	3	5	7					
BCMOVE	1	approach							

Descriptor	Meaning
ID	Identification number referenced by a SUBCASE or STEP Case Control command. See Remark 1. (Integer ≥ 0 ; Required)
MTYPE	Movement type. (Character; Default = APPROACH)
APPROACH	All rigid bodies are moved so that they all make contact with deformable bodies.
RELEASE	The contact condition is released for selected bodies.
SYNCHRON	All rigid bodies are moved until the first rigid body makes contact with a deformable body.
IREL	Flag to indicate how contact forces are removed, for option RELEASE only. (Integer)
= 0	Contact forces are immediately removed. (Default)
> 0	Contact forces are reduced to zero over the number of increments specified in this load period. See NLPARM and TSTEP1 for the number of increments.
IDRBODi	Identification numbers of rigid bodies to be released, for option RELEASE only. Points to BCBODY Bulk Data entries.

Remarks:

1. To place an entry in Marc's phase 0, set ID=0. To activate the entry for the 1st SUBCASE, SET ID=1, for the 2nd, set ID=2. ID must be unique (only one BMOVE per SUBCASE).
2. This entry matches Marc's history definitions RELEASE, APPROACH, and SYNCHRONIZED. Note that Marc's history definition MOTION CHANGE is done in MSC Nastran by describing the enforced motion for the grid which is defined to be the center of rotation of the rigid body, see CGID of the BCBODY entry.

3. For MTYPE=APPROACH and MTYPE=SYNCHRON leave all following fields blank.
4. The APPROACH and SYNCHRON options apply to rigid contact surfaces only.
5. You may release a deformable body from contact with either a deformable or rigid body.

BCNURB2

Defines a 2D Rigid Contact Body Made up of NURBS in SOLs 101 and 400

Defines a 2D rigid contact body made up of NURBS used in SOLs 101 and 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
BCNURB2	RBID	NPTU	NORU	NSUB					
	“GRID” or “COORD”	G1 or X1	G2 or Y1	G3 or X2	G4 or Y2	-etc.-			
	“HOMO”	Homo1	Homo2	Homo3	Homo4	-etc.-			
	“KNOT”	Knot1	Knot2	Knot3	Knot4	Knot5	-etc.-		

Examples:

BCNURB2	102	4	1	1					
	GRID	237	101	104	235				
	HOMO	0.0	0.5	0.5	1.0				
	KNOT	0.0	0.2	0.4	0.8	1.0			

BCNURB2	3001	-5	3	50					
	COORD	-.1	.14	-.1	.04	-1.4-16	.04		
		.1	.04	.1	.14				
	HOMO	1.	.707107	1.	.707107	1.			
	KNOT	0.0	0.0	0.0	0.5	0.5	1.0	1.0	
		1.0							

Descriptor Meaning

RBID Unique identification number referenced by a BCRGSRF or BCBODY1 entry.
(Integer > 0) See Remark 1.

NPTU Number of control points. If the control points are entered as coordinates rather than grid IDs NPTU must be set to a negative value whose absolute value is the number of xy coordinates. (Integer; no Default)

NORU Order along U direction. (Integer > 0)

NSUB Number of subdivisions. (Integer > 0)

“GRID” Indicate the start of the list of grid numbers. See Remark 2.

G1, G2, G3, G4,
etc. Grid numbers for each of the NPTU control points. (Integer > 0)

“COORD” Indicate the start of the list of xy coordinates. See Remark 2.

X1, Y1, X2, Y2, etc.	Alternate method to define control points without using GRID points. There must be abs(NPTU) set of (x,y) entries. (Real)
"HOMO"	Indicate the start of the list of homogeneous coordinates.
Homo1,Homo2, Homo3, etc.	Homogeneous coordinates. (Real; 0.0 to 1.0) There must be NPTU entries.
"KNOT"	Indicate the start of the list of knot vectors.
Knot1, Knot2, Knot3, etc.	Knot vectors. (Real; 0.0 to 1.0) There must be (NPTU+NORU) entries.

Remarks:

1. If BCRGSRF entry does not exist, BCNURB2 entry will be referenced by the [BCBODY1](#) entry directly.
2. The list of grid IDs ("GRID") and the list of xy coordinates ("COORD") cannot coexist.

BCNURBS

Defines a Rigid Contact Body Made up of NURBS in SOLs 101 and 400

Defines a rigid contact body made up of NURBS used in SOLs 101 and 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
BCNURBS	RBID	NPTU	NPTV	NORU	NORV	NSUBU	NSUBV		
	“GRID” or “COORD”	G1 or X1	G2 or Y1	G3 or Z1	G4 or X2	G5 or Y2	G6 or Z2	-etc.-	
	“HOMO”	Homo1	Homo2	Homo3	Homo4	Homo5	Homo6	Homo7	
		Homo8	Homo9	-etc.-					
	“KNOT”	Knot1	Knot2	Knot3	Knot4	Knot5	Knot6	Knot7	
		Knot8	Knot9	-etc.-					
	“TRIM”	IDtrim1	IDtrim2	IDtrim3	-etc.-				

Examples:

BCNURBS	48	2	1	2	1	50	50		
	GRID	3005	102						
	HOMO	1.	.3333						
	KNOT	0.0	0.0	0.5	0.5	1.	1.		

BCNURBS	63	-2	2	2	2	50	50		
	COORD	.3	-.6	0.	.3	-.1	0.		
	HOMO	1.	1.	1.	1.				
	KNOT	0.0	0.0	1.	1.	0.	0.		
		1.	1.						
	TRIM	511	2002	87	704				

Descriptor **Meaning**

RBID Unique identification number referenced by a BCRGSRF or BCBODY1 entry.
(Integer > 0) See Remark 1.

NPTU Number of control points in U direction. If the control points are entered as coordinates rather than grid IDs, NPTU must be set to a negative value whose absolute value is the number of xyz coordinates. See Remark 2. (Integer > 0; Required)

NPTV Number of control points in V directions. (Integer > 0; Required)

NORU Order along U direction. (Integer > 0; Required)

NORV	Order along V direction. (Integer > 0; Required)
NSUBU	Number of subdivisions in U direction. (Integer > 0; Required)
NSUBV	Number of subdivisions in V direction. (Integer > 0; Required)
“GRID”	Indicate the start of the list of grid numbers. See Remark 3.
G1, G2, G3, etc.	Grid point IDs defining control points. (Integer > 0) There must be $\text{NPTU} \times \text{NPTV}$ entries.
“COORD”	Indicate the start of the list of xyz coordinates. See Remark 3.
X1, Y1, Z1, X2, Y2, Z2, etc.	Alternate method to define control points without using GRID points. There must be $\text{abs}(\text{NPTU}) \times \text{NPTV}$ sets of (x,y,z) entries.
“HOMO”	Indicate the start of the list of homogeneous coordinates.
Homo1, Homo2, Homo3, etc.	Homogeneous coordinates (0.0 to 1.0). There must be $\text{abs}(\text{NPTU}) \times \text{NPTV}$ entries. (Real)
“KNOT”	Indicate the start of the list of knot vectors.
Knot1, Knot2, Knot3, etc.	Knot vectors (0.0 to 1.0). There must be $(\text{abs}(\text{NPTU}) + \text{NORU}) + (\text{NPTV} + \text{NORV})$ entries. (Real)
“TRIM”	Indicate the start of the list of trimming vector IDs.
IDtrimi	Trimming vector identification number of a BCTRIM entry. (Integer > 0)
Homot1, Homot2, Homot3, etc.	Homogeneous coordinates (0.0 to 1.0) of this trimming vector. There must be NPTUtrim entries. (Real)
Knott1, Knott2, Knott3, etc.	Knot vectors (0.0 to 1.0) of this trimming vector. There must be NPTUtrim+NORUtrim entries. (Real)

Remarks:

1. If BCRGSRF entry does not exist, BCNURBS entry will be referenced by the BCBODY1 entry directly.
2. Enter NPTU grid points G1, G2, G3, etc. (set NPTU to a positive value equal to the number of grid points) or enter X1, Y1, Z1, X2, Y2, Z2, etc. coordinates for $\text{abs}(\text{NPTU})$ points and set NPTU to a negative value.
3. The list of grid IDs (“GRID”) and the list of xyz coordinates (“COORD”) cannot coexist.

BCONECT

Defines a Contact Pair

Defines a contact pair used in SOLs 101, 103, 105, 107, 108, 109, 110, 111, 112, 200, 400 and 700. Only SOL 101, 400 and 700 can support all contact types: touching contact, glued, step glue and permanent glue (see [Chapter 7: Contact Types of Nastran SOL 400 Getting Started Guide](#)). Note that SOL 700 calls it “tie” contact to glue (also step glue or permanent glue). SOL 103-112 and standard 200 (without calling SOL 400) can only support permanent glue. When SOL 200 calls SOL 400 (or say SOL 400 optimization), it can support all contact types.

Format:

1	2	3	4	5	6	7	8	9	10
BCONECT	ID	BCGPID	BCPPID	IDSLAVE	IDMASTER				
	“SLAVES”	IDSL1	IDSL2	IDSL3	IDSL4	IDSL5	IDSL6	IDSL7	
		IDSL8	IDSL9	-etc.-					
	“MASTERS”	IDMA1	IDMA2	IDMA3	IDMA4	IDMA5	IDMA6	IDMA7	
		IDMA8	IDMA9	-etc.-					

Examples:

BCONECT	57	306		2	1002				
---------	----	-----	--	---	------	--	--	--	--

BCONECT	9		108						
	SLAVES	30	26						
	MASTERS	294	135	528					

Descriptor Meaning

- | | |
|----------|---|
| ID | Unique identification number referenced by a BCTABL1 entry (Integer ≥ 0). See Remark 1. |
| BCGPID | Parameter identification number of a BCONPRG entry (Integer ≥ 0 or blank). See Remark 2. |
| BCPPID | Parameter identification number of a BCONPRP entry (Integer ≥ 0 or blank). See Remark 2. |
| IDSLAVE | Identification number of BCBODY1, BCSURF and BCGRID entry defining the touching body (Integer ≥ 0 or blank). See Remarks 3. and 4. |
| IDMASTER | Identification number of BCBODY1 and BCSURF entry defining the touched body (Integer ≥ 0 or blank). See Remarks 5. and 6. |
| “SLAVES” | Indicates the start of the list of the touching bodies. See Remark 4. |

IDSLi	Identification number of BCBODY1 entry defining the touching bodies (Integer ≥ 0 or blank).
	For SOL 700, leaving IDSL1 blank will result in contact for all elements in the model. In this case, you are allowed to use ADAPT=YES.
"MASTERS"	Indicates the start of the list of bodies touched by touching bodies. See Remark 6.
IDMAi	Identification number of BCBODY1 entry defining touched bodies (Integer ≥ 0 or blank).

Remarks:

1. BCONNECT can be selected by the Case Control command BCONTACT=ID to define surface contact if BCTABL1 entry does not exist. See Remarks 2.. and 3.. of BCTABL1 entry.
2. If BCGPID or BCPPID field is blank, then default values are set for the parameters of touching bodies.
3. A short input to define a single touching body exists if the user provides IDSLAVE. On the other hand, if the user leaves IDSLAVE blank, then "SLAVES" descriptor is required and IDSL1 must be specified. Exceptions are for SOL 700 self-contact, which may use a slave IDSL1 of zero and no "MASTERS" entry.
4. "SLAVES" and IDSLi fields will be ignored if IDSLAVE exists. If IDSLAVE field is blank, then "SLAVES" and IDSLi must be specified. In this case, each IDSLi will be processed separately.
5. A short input to define a single touched body exists if the user provides IDMASTER. On the other hand, if the user leaves IDMASTER blank, then "MASTERS" descriptor is required and IDMA1 must be specified.
6. "MASTERS" and IDMAi fields will be ignored if IDMASTER exists. If IDMASTER field is blank, then "MASTERS" and IDMAi must be specified.
7. The concept of Slave and Master relation is important to the node-to-segment contact but not relevant for segment-to-segment contact. In segment-to-segment contact, they are mainly used to define the contact pair(s).
8. If all the BCONPRG that are referenced by a BCONNECT (which is referenced by BCTABL1) have a value of IGLUE > 0 , and this BCONNECT is referenced in the 1st Loadcase (SOL 100*) or the 1st Step (SOL 400) then the connections are considered to be permanent and do not change (unless a BCPARA is used to deactivate the permanent glue).

BCONP

Contact Parameters

Defines the parameters for a contact region and its properties for slideline contact in SOL 106 or SOL 129. SOL 400 is the preferred method for contact analysis, see BCBODY1, BCNECT, BCONPRG, BCONPRP, BCPARA and BCTABL1.

Format:

1	2	3	4	5	6	7	8	9	10
BCONP	ID	SLAVE	MASTER		SFAC	FRICID	PTYPE	CID	

Example:

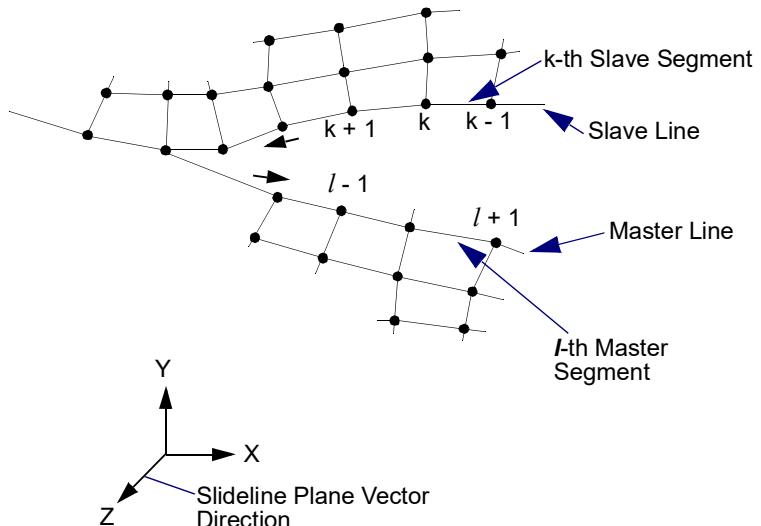
BCONP	95	10	15		1.0	33	1		
-------	----	----	----	--	-----	----	---	--	--

Descriptor	Meaning
ID	Contact region identification number. See Remark 1. (Integer > 0)
SLAVE	Slave region identification number. See Remark 2. (Integer > 0)
MASTER	Master region identification number. See Remark 3. (Integer > 0)
SFAC	Stiffness scaling factor. SFAC is used to scale the penalty values automatically calculated by the program. See Remark 4. (Real > 0.0; Default = 1.0)
FRICID	Contact friction identification number. See Remark 5. (Integer > 0 or blank)
PTYPE	Penetration type. See Remark 6. (Integer 1 or 2; Default = 1) 1 Unsymmetrical (slave penetration only--Default) 2 Symmetrical
CID	Coordinate system identification number to define the slideline plane vector and the slideline plane of contact. See Remark 7. (Integer > 0; Default = 0, which means the basic coordinate system)

Remarks:

1. ID field must be unique with respect to all other BCONP identification numbers.
2. The referenced SLAVE is the identification number in the BLSEG Bulk Data entry. This is the slave line. The width of each slave segment must also be defined to get proper contact stresses. See the Bulk Data entry, [BWIDTH, 1447](#) for the details of specifying widths.
3. The referenced MASTER is the identification number in the BLSEG Bulk Data entry. This is the master line. For symmetrical penetration, the width of each master segment must also be defined. See the Bulk Data entry, [BWIDTH, 1447](#) for the details of specifying widths.

4. SFAC may be used to scale the penalty values automatically calculated by the program. The program calculates the penalty value as a function of the diagonal stiffness matrix coefficients that are in the contact region. In addition to SFAC, penalty values calculated by the program may be further scaled by the ADPCON parameter (see description of ADPCON parameter for more details). The penalty value is then equal to $k \cdot SFAC \cdot |ADPCON|$, where k is a function of the local stiffness. It should be noted that the value in SFAC applies to only one contact region, whereas the ADPCON parameter applies to all the contact regions in the model.
5. The referenced FRICID is the identification number of the BFRIC Bulk Data entry. The BFRLC defines the frictional properties for the contact region.
6. In an unsymmetrical contact algorithm only slave nodes are checked for penetration into master segments. This may result in master nodes penetrating the slave line. However, the error depends only on the mesh discretization. In symmetric penetration both slave and master nodes are checked for penetration. Thus, no distinction is made between slave and master. Symmetric penetration may be up to thirty percent more expensive than the unsymmetric penetration.
7. In [Figure 9-2](#), the unit vector in the Z-axis of the coordinate system defines the slideline plane vector. The slideline plane vector is normal to the slideline plane. Relative motions outside the slideline plane are ignored, and therefore must be small compared to a typical master segment. For a master segment the direction from master node 1 to master node 2 gives the tangential direction (t). The normal direction for a master segment is obtained by the cross product of the slideline plane vector with the unit tangent vector (i.e., $n = z \times t$). The definition of the coordinate system should be such that the normal direction must point toward the slave region. For symmetric, penetration, the normals of master segments and slave segments must face each other. This is generally accomplished by traversing from master line to slave line in a counterclockwise or clockwise fashion depending on whether the slideline plane vector forms a right-hand or left-hand coordinate system with the slideline plane.



- X-Y plane is the slideline plane. Unit normal in the Z-direction is the slideline plane vector.
- Arrows show positive direction for ordering nodes. Counterclockwise from master line to slave line.
- Slave and master segment normals must face each other.

[Figure 9-2](#) A Typical Finite Element Slideline Contact Region

BCONPRG**Geometric Contact Parameters of Touching Bodies**

Defines geometric contact parameters used to determine if contact occurs between bodies. For segment-to-segment contact it also defines additional numerical parameters used to apply the constraints. The parameters defined here are referenced by the BCONECT entry. This entry is used in conjunction with the BCONPRP entry.

Geometric Contact Parameters of Touching Bodies in SOLs 101 and 400 for General Contact or in SOLs 101, 103, 105, 107 - 112, 200 and 400 for the Permanently Glued or Tied Contact.

Format:

1	2	3	4	5	6	7	8	9	10
BCONPRG	BCGPID		PARAM1	VAL1	PARAM2	VAL2	PARAM3	VAL3	
	PARAM4	VAL4	PARAM5	VAL5	-etc.-				

Example:

BCONPRG	90		ICOORD	1	IGLUE	1			
---------	----	--	--------	---	-------	---	--	--	--

Descriptor Meaning

BCGPID Identification number for geometric contact parameters (Integer > 0).

PARAMi Name of a parameter. Allowable names are given in [Table 9-4](#) (Character).

VALi Value of a parameter. See [Table 9-4](#) (Real or Integer).

Table 9-4 Geometric Contact Parameters of Touching Bodies in SOLs 101, 103, 105, 107-112, 200 and 400

Name	Description, Type and Value (Default is 0 for integer, 0.0 for Real Unless Otherwise Indicated)
AUGDIST	Penetration distance beyond which an augmentation will be applied; used by the segment-to-segment contact algorithm only. (Real ≥ 0.0 , see Remark 6. for default)
BIAS	Contact tolerance bias factor. If this field is left blank, the default is the BIAS of the BCPARA entry. A nonblank entry will override the BIAS entered on the BCPARA entry. Note 0.0 is not default, and will override the BIAS on BCPARA. SYSTEM(758)=1 will set 0.0 same as blank. (0.0<=Real<=1.0)
CINTERF	The definition varies depending on OPINTRF. (Real; Default = 0.0) When OPINTRF is set to 0 (default), Interference closure amount, normal to the contact surface. For CINTERF > 0.0, overlap between bodies. For CINTERF < 0.0, gap between bodies. When OPINTRF is set to 1, interference closure magnitude. To clear penetration, set a negative value. When OPINTRF is set to 2, magnitude of translation vector defining in VXINTRF, VYINTRF and VZINTRF. When OPINTRF is set to 3, not used. When OPINTRF is set to 4, not used.
COPTS, COPTM	Flag to indicate how slave and master surfaces may contact. See Remark 2. (Integer; Default = 1011) COPTS and COPTM apply to all slave and master surfaces on the associated BCONECT. COPTS and COPTM are ignored if slave and master in BCONECT are BCSURF or BCGRID entries.
ERROR	Distance below which a node is considered touching a body. Default = blank, automatic calculation. See BCPARA Bulk Data entry for more details. (Real)
HARDS	Hard-soft ratio. This entry is only used if double-sided contact with automatic constraint optimization is used, i.e. ISTYP=2 on the BCBDRP entry. The hard-soft ratio can be used by the program if there is a significant difference in the average stiffness of the contact bodies (expressed by the trace of the initial stress-strain law). If the ratio of the stiffnesses is larger than the hard-soft ratio, the nodes of the softest body are the preferred slave nodes. (Real; Default = 2.0) This parameter is ignored for the permanently glued contact.
FGCFLG	Flag to activate Cohesive (Flexible) Glued Contact: (See Remark 12.) 0 – no Cohesive (Flexible) Glued Contact 1 – activate Cohesive (Flexible) Glued Contact
FGCNST	Equivalent normal contact stiffness of connector for Cohesive (Flexible) Glued Contact
FGCTST	Equivalent tangential contact stiffness of connector for Cohesive (Flexible) Glued Contact
FGCNSTR	ID of table for define contact normal stress versus relative displacement for Cohesive (Flexible) Glued Contact. Only TABL3D allowable.

FGCTSTR	ID of table for define contact tangential stress versus relative displacement for Cohesive (Flexible) Glued Contact. Only TABL3D allowable.
FGCNSTI	ID of table for define contact normal stiffness versus relative displacement or temperature for Cohesive (Flexible) Glued Contact. Only TABL3D allowable.
FGCTSTI	ID of table for define contact tangential stiffness versus relative displacement or temperature for Cohesive (Flexible) Glued Contact. Only TABL3D allowable.
FGCRCEN	ID of grid on which the resultant contact and moment are based.
ICOORD	Enter 1 to modify the coordinates of a node in contact with a deformable body so that stress-free initial contact can be obtained. Enter 2 to extend the tangential error tolerance at sharp corners of deformable bodies to delay sliding off a contacted segment. Enter 3 to have both 1 and 2 active. (Integer; Default = 0)
IGLUE	Flag to activate glue options (Integer ≥ 0). Default is 0, no glue option. See JGLUE option for controlling a glue contact type in details. This field must be specified for the permanently glued contact. See Remarks 4. and 5. A negative value of IGLUE indicates that this method of glue will be used for the entire Step in SOL 400. See Remark 9.
1 or -1	Activates the glue option. In the glue option, all degrees-of-freedom of the contact nodes are tied in case of deformable-deformable contact once the node comes in contact. The relative tangential motion of a contact node is zero in case of deformable-rigid contact. The node will be projected onto the contact body. For the permanently glued contact, this option is recommended when there is no gap or overlap between contact surfaces or initial stress free contact is specified.
2 or -2	Activates a special glue option to insure that there is no relative tangential and normal displacement when a node comes into contact. An existing initial gap or overlap between the node and the contacted body will not be removed, as the node will not be projected onto the contacted body. To maintain an initial gap, ERROR should be set to a value slightly larger than the physical gap.
3 or -3	Insures full moment carrying glue when shells contact. The node will be projected onto the contacted body. For the permanently glued contact, this option is recommended when moments are important and there is no gap or overlap between contact surfaces or initial stress free contact is specified.
4 or -4	Insures full moment carrying glue when shells contact. The node will not be projected onto the contact body and an existing initial gap or overlap between the node and the contacted body will not be removed, as the node will not be projected onto the contacted body.

ISEARCH	<p>Option for contact searching order, from Slave to Master or from Master to Slave, for deformable contact bodies. ISEARCH is not necessary in segment-to-segment contact. (Integer; Default = 0)</p> <p>0 (Double orders search) the search order is from lower BCBODY1 ID's to higher ones first. Then it searches the opposite order. See Remark 3.</p> <p>1 (Single order search) the searching order is from Slave to Master.</p> <p>2 (Single order search) let the program decide which search order. See Remark 3.</p> <p>Note that ISTYP in BCBDPRP or BCBODY is supported with ISEARCH=0 only.</p>
JGLUE	This option is only relevant if the glue option is invoked (JGLUE > 0). Enter 0 if a node should not separate (default). Enter 1 to invoke the standard separation behavior based on the maximum residual force (related to FNTOL parameter of BCPARA or BCONPRP entry). Enter 2 to activate breaking glue (related to BGM, BGN, BGSN and BGST parameters of BCONPRP entry). (Integer; Default = 0) This parameter is ignored for the permanently glued contact or step glued contact, see Remark 9.
PENALT	Augmented Lagrange penalty factor; used by the segment-to-segment contact algorithm only. (Real ≥ 0.0 ; see Remark 7. for default)
SLIDE	Delayed slide off distance. This entry should not be made unless ICOORD ≥ 2 (see above). When using the delayed slide off option, a node sliding on a segment will slide off this segment only if it passes the node (2-D) or edge (3-D) at a sharp corner over a distance larger than the delayed slide off distance. By default, the delayed slide off distance is related to the dimensions of the contacted segment by a 20 percent increase of its isoparametric domain. (Real; Default=0.0)
STKSLP	Maximum allowable slip distance for sticking, beyond it there is no sticking, only sliding exists, used by the segment-to-segment contact algorithm only. (Real ≥ 0.0 ; Default = 0.0) See Remark 8.
TPENALT	Augmented Lagrange penalty factor for sticking part of friction, used by the segment-to-segment contact algorithm only. (Real > 0.0) The default is PENALT/1000, where PENALT parameter is the Augmented Lagrange penalty factor for normal contact.
OPINTRF	The method for interference fit. (Integer ≥ 0 ; Default=0) See Remark 10. 0 small interference fit (solved in one increment) 1 interference fit varying with time; resolved in the normal direction 2 interference fit varying with time in a user specified direction 3 interference fit varying with time using scale factor 4 automatic interference fit varying with time and node location

TBINTRF	TABLED1 id used by interference fit. (Integer>0; Required only when OPINTRF > 0) When OPINTRF=1, TABLED1 id giving variation of interference closure with respect to time When OPINTRF=2, TABLED1 id giving variation of magnitude of translation vector with respect to time When OPINTRF=3, TABLED1 id giving variation of magnitude of scale factor with respect to time When OPINTRF=4, TABLED1 id giving variation of magnitude of penetration vector projected node position on the master segment with respect to time
CBINTRF	Contact body selection for interference fit. Only required when OPINTRF=2, 3 or 4. (Integer ≥ 0 ; Default=0) When OPINTRF=2, Contact body ID for which translation vector to be used When OPINTRF=3, Contact body ID for which scale factors to be used. When OPINTRF=4, Contact body ID of which penetrations to be determined. 0 slave body 1 master body
CDINTRF	Coordinate system ID of VXINTRF, VYINTRF and VZINTRF for interference fit. Only required when OPINTRF=2, 3. (Integer ≥ 0 ; Default=0) When OPINTRF=2, Coordinate system for direction cosines for the translation vector. When OPINTRF=3, Coordinate system for scale factor vector.
VXINTRF	X component of vector for interference fit. Only required when OPINTRF=2, 3. (Real; default=0.0) When OPINTRF=2, Direction cosines for the translation vector. When OPINTRF=3, scale factor vector
VYINTRF	Y component of vector for interference fit. Only required when OPINTRF=2, 3. (Real; default=0.0) When OPINTRF=2, Direction cosines for the translation vector. When OPINTRF=3, scale factor vector.
VZINTRF	Z component of vector for interference fit. Only required when OPINTRF=2, 3. (Real; default=0.0) When OPINTRF=2, Direction cosines for the translation vector. When OPINTRF=3, scale factor vector.
PTINTRF	Penetration search tolerance for interference fit. Only required when OPINTRF=4. (Real ≥ 0.0 ; default=error tolerance*100)
XCINTRF	X component of center of scaling. Only required when OPINTRF=3. (Real; default=0.0) Center is described in global coordinate (Coord0).
YCINTRF	Y component of center of scaling. Only required when OPINTRF=3. (Real; default=0.0) Center is described in global coordinate (Coord0).
ZCINTRF	Z component of center of scaling. Only required when OPINTRF=3. (Real; default=0.0) Center is described in global coordinate (Coord0).

OPINGP	Initial gap or overlap option. See Remark 1. (Integer ≥ 0 ; default=0) 0 not used 1 initial gap or overlap
TOLINGP	Search tolerance of initial gap. See Remark 14.. (Real ≥ 0.0 ; default= average edge length of all contact segments, required only when OPINGP =1)
TOLING1	Search tolerance of initial overlap. See Remark 14. (Real ≥ 0.0 ; default=average edge length of all contact segments, required only when OPINGP =1)
CDINGP	Contact body to be adjusted. (Integer ≥ 0 ; default=0, required only when OPINGP =1) 0 slave body 1 master body
MGINGP	Gap or overlap magnitude. (Real ; default=0.0, required only when OPINGP =1) > 0 gap =0 Preserve initial clearance distance. See Remark 11. < 0 overlap
SFNPNLT	Scale factor of augmented Lagrange penalty factor along contact normal direction; used by segment-to-segment contact method only. (Real>0.0;Default=1.0) SFNPNLT*TPENALT is used in analysis. If this field is left default, value of SFNPNLT entered in BCPARA will be used. A nonblank value entered in SFNPNLT will override the value of SFNPNLT in BCPARA.
SFTPNLT	Scale factor of augmented Lagrange penalty factor along contact tangential direction; used by segment-to-segment contact method only. (Real>0.0;Default=1.0) SFTPNLT*TPENALT is used in analysis. If this field is left default, value of SFTPNLT entered in BCPARA will be used. A nonblank value entered in SFTPNLT will override the value of SFTPNLT in BCPARA.

Remarks:

1. The multipoint constraint equations (MPC equations) internally created from general contact or glued contact can be printed out in standard Nastran punch file by using Case Control command, NLOPRM MPCPCH.
2. COPTS and COPTM are packed numbers designating how the surfaces may contact using the formula

$$\text{COPTx} = A + 10^*B + 1000^*C$$

where the following codes apply:

A: the outside of the solid elements in the body.

- = 1: the outside will be in the contact description (Default).

B: (flexible bodies): the outside of the shell elements in the body.

- = 1: both top and bottom faces will be in the contact description, thickness offset will be included (Default).
- = 2: only bottom faces will be in the contact description, thickness offset will be included.
- = 3: only bottom faces will be in the contact description, shell thickness will be ignored.

- = 4: only top faces will be in the contact description, thickness offset will be included.
- = 5: only top faces will be in the contact description, shell thickness will be ignored.
- = 6: both top and bottom faces will be in the contact description, shell thickness will be ignored.

Note: The choice B = 6 for both bodies in a contact combination is only meaningful for glued contact. If in such cases separation is allowed, separated nodes will not come into contact anymore, unless a new CONTACT TABLE is defined to reset the value of B.

Note: for segment-to-segment contact only use B=1 or B=6.

Note: For thermal contact with 2D shell elements, i.e. uniform temperature gradient across thickness, do not use B=2 or B=3.

B: (rigid bodies): the rigid surface.

- = 1: the rigid surface should be in the contact description (Default).

C: (flexible bodies): the edges of the body.

- = 1: only the beam/bar edges are included in the contact description (Default).
- = 10: only the free and hard shell edges are included in the contact description.
- = 11: both the beam/bar edges and the free and hard shell edges are included in the contact description.

Note that C has no effect if beam-to-beam contact is not switched on. (BEAMB ≠ 1 on BCPARA).

Note that C has no effect if segment-to-segment contact is used.

- When ISEARCH=0 (and ISTYP=0 in default on BCBDRP), the search order is from lower BCBODY1 ID to higher one to create the first set of contact constraints and then add the constraints in the search order from higher BCBODY1 ID to lower one as long as they are not in conflict with the first set.

When ISEARCH=2, the program looks into the smallest element edge at the outer boundary (and the smallest thickness in case of shell elements) of each BCBODY1. Then, the search order of the two deformable contact bodies is determined by the following rule when ID1 < ID2.

CL1= Min(1/20 of the smallest edge,1/4 of the smallest thickness) of BCBODY1 ID1

CL2= Min(1/20 of the smallest edge,1/4 of the smallest thickness) of BCBODY1 ID2

The search order is from lower BCBODY1 ID1 to higher BCBODY1 ID2 if CL1≤1.05*CL2.
Otherwise, if CL1>1.05*CL2 the search order is from BCBODY1 ID2 to BCBODY1 ID1.

- For the permanently glued contact, IGLUE = ±2 or ±4 is favorable to pass the GROUNDCHECK. The initial stress free contact is also available to preserve the six rigid body modes with IGLUE = ±1 or ±3.
- A permanently glued contact for small rotation cannot be used to glue a deformable body to a rigid one. If it is a SOL 101 or SOL 400 job, the permanently glued contact for large deformation and rotation (IGLUE <0) or general contact (IGLUE > 0) with glued option must be performed.
- By default, the threshold value of this penetration distance is 0.05 times the default contact characteristic distance.

AUGDIST=0.05 L

7. The penalty factor used in the augmented Lagrange method is by default derived from the contact characteristic distance and the stiffness of the deformable contact bodies involved (note that the dimension of the penalty factor is force per cubic length).

$$\text{PENALT} = \frac{0.5(S_i + S_j)}{L}$$

The body stiffness (S_i and S_j), are either defined by the average trace of the initial stress-strain law of the elements defining the two contact bodies or by the average bulk modulus for (nearly) incompressible rubber materials, whichever of the two is the largest.

For continuum elements, the characteristic length (L) is given by one half of the average length of all the edges being part of the contact boundary. For shell elements, the characteristic length is given by half of the average thickness of all the shell elements being part of a contact body. When there is contact between a solid and a shell element, then the characteristic length is defined by the shell element.

In case of contact with a rigid body, since there is no body stiffness associated with a rigid body, the default value is related to the deformation body only and is given by

$$\text{PENALT} = \frac{1000S_i}{L}$$

8. If STKSLP is set to 0.0 (Default), the sticking stiffness K1 is equal to the maximum friction force ($\mu \cdot F_{\text{normal}}$, where μ is the friction coefficient) divided by the maximum sticking displacement. Otherwise, K1 is equal to the maximum friction force divided by the value of STKSLP.
9. A negative value of IGLUE enforces the gluing of the pair of bodies in the BCONECT that reference this BCONPRG is based upon the geometric conditions at the beginning of the step and will not change over this step. IGLUE will be changed to negative automatically if SYSTEM(758)=2 when Permanently Glued setting is found with large rotation/deformation effect turned on.
10. For interference fit contact, the following options are available.
CINTERF, OPINTRF, TBINTRF, CBINTRF, CDINTRF, VXINTRF, VYINTRF, VZINTRF, PTINTRF, XCINTRF, YCINTRF and ZCINTRF
11. For setting the initial gap or overlap, the following options are available. OPINGP, TOLINGP, CDINGP and MGINGP.

When OPINGP is 1 in means that there it will try to preserve a distance between the two bodies often called the clearance distance.

When MGINGP is zero, the clearance is not changed from the distance between the original coordinate positions of the nodes. This is call the distance D0

When MGINGP is > 0 it means the distance between the bodies can shrink (i.e. the bodies can get closer together), but when the distance reaches $D0 - MGINGP$. The clearance condition is satisfied and this clearance distance ($D0 - MGINGP$) is preserved.

When MGINGP is < 0 in means that it considers the bodies to be overlapping, so it first tries to separate them by a distance $= |MGINGP|$; then it imposes the clearance condition such that the clearance distance is $D0 + |MGINGP|$

12. Please note that
 - Cohesive Contact is supported for only for segment-to-segment contact.
 - All limitations of segment-to-segment contact is applicable for cohesive contact feature also.
 - Breaking Glue feature is not supported with Cohesive contact.
13. When the initial stress-free(ICOORD=1), Node-to-Segment updates model geometry to close the gap, nodes are projected onto the contact body; but Segment-to-Segment treats initial stress-free as pre-stress in the equation without geometry update, no node projection is considered.
14. TOLINGP and TOLING1 follows the rules below.
 - a. When both of TOLINGP and TOLING1 are not defined, the default value of average edge length of all contact segments will be used for both.
 - b. When TOLINGP is only defined, TOLINGP value will be used for both of TOLINGP and TOLING1.
 - c. When TOLING1 is only defined, TOLING1 value will be used for TOLING1 only, and the default value of average edge length of all contact segments will be used for TOLINGP.
 - d. When both of TOLINGP and TOLING1 are defined, each values are used for each case

BCONPRG-700

Geometric Contact Parameters of Touching Bodies in SOL 700

Defines geometric contact parameters of touching bodies used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10	11
BCONPRP	PID		PARAM1	VAL1	PARAM2	VAL2	PARAM3	VAL3		
	PARAM4	VAL4	PARAM5	VAL5	-etc.-					

Example:

BCONPRP	90		IGLUE	1						
---------	----	--	-------	---	--	--	--	--	--	--

Descriptor Meaning

PID Parameter identification number (Integer > 0).

PARAMi Name of a parameter. Allowable names are given in [Table 9-5](#) (Character).

VALi Value of a parameter. See [Table 9-5](#) (Real or Integer).

Table 9-5 Geometric Contact Parameters of Touching Bodies in SOL 700

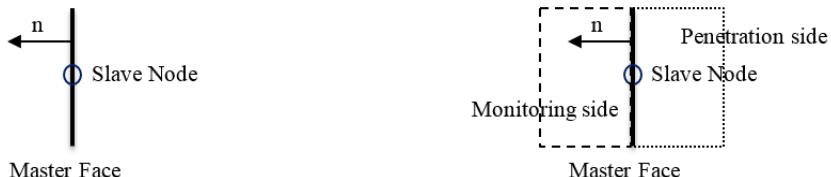
Name	Description, Type and Value (Default is 0 for integer, 0.0 for Real Unless Otherwise Indicated)
IGLUE	<p>Flag to activate glue option (Integer = 0 or 1). Default is 0, no glue option. JGLUE option controls a glue contact type in details. Without assigning IGLUE and JGLUE options, tied contact methods (METHOD=TIEDxxx) can assign glue contact types as well.</p> <ul style="list-style-type: none"> 0. No glue contact. 1. Activates the glue option. In the glue option, all degrees-of-freedom of the contact nodes are tied in case of deformable-deformable contact once the node comes in contact. The relative tangential motion of a contact node is zero in case of deformable-rigid contact. The node will be projected onto the contact body.
JGLUE	<p>This option is only relevant if the glue option is invoked (IGLUE > 0). See METHOD=TIEDxxx. (Integer ≥ 0; Default = 0) The following options are available:</p> <ul style="list-style-type: none"> 0 Slave nodes in contact and which come into contact will permanently stick. Tangential motion is inhibited. This option is only available with METHOD=SS1WAY or SS2WAY and AUTO=YES.
METHOD	<p>Character, Influences the contact type used. See Remark 18. (Character, Default = FULL)</p> <p>Options are:</p> <ul style="list-style-type: none"> FULL: Regular Contact AIRBAG: Single Surface Contact SS1WAY: Surface To Surface One Way SS2WAY: Surface To Surface Two Way RB1WAY: Rigid Body One Way To Rigid Body RB2WAY: Rigid Body Two Way To Rigid Body RNRB: Rigid Nodes To Rigid Body TIEDNS: Tied Nodes to Surface TIEDNSO: Tied Nodes to Surface with Offset
ADAPT	<p>Character, influences the contact type used. (Character, Default = NO)</p> <p>Options are NO or YES.</p> <p>When ADAPT=YES, the BCBODY1 entries IDMAi must be defined as:</p> <ul style="list-style-type: none"> behav=DEFORM bsid references a BCPROP
THICK	Shell thickness scale factor. (Real > 0.0; Default = same as the value in DYPARAM,CONTACT,THICK)

THICKOF	Artificial contact thickness offset. (Real > 0.0; Default = 0.0)
PENV	Overwrites the default maximum penetration distance factor. (Real > 0.0;Default = 1.E20)
MAXPAR	Maximum parametric coordinate in segment. (Real > 0.0; Default = 1.025), search (values 1.025 and 1.20 recommended). Larger values can increase cost. If zero, the default is set to 1.025. This factor allows an increase in the size of the segments. May be useful at sharp corners.
SOFT	<p>Soft constraint option: (Integer \geq 0; Default = 1)</p> <p>1 Soft constraint formulation (same as original Dytran method).</p> <p>In the soft constraint option, the interface stiffness is based on the nodal mass and the global time step size. This method is more suited for contact between two materials where the elastic moduli vary greatly.</p>
IGNORE	<p>Ignore initial penetrations. (Integer, Default = 1)</p> <p>0 Take default from PARAM,DYCONIGNORE*,<value> (Default)</p> <p>1 Allow initial penetrations to exist by tracking it</p> <p>2 Move nodes to eliminate initial penetrations</p>
AUTO	<p>Activation of automatic contact (Character, Default = Yes.)</p> <p>Options are:</p> <p>YES Automatic Contacts Activated</p> <p>NO Non-Automatic Contact Activated. This option is not recommended when Distributed Memory Parallel is activated.</p>
SIDE	<p>Defines which side will be the monitoring side of a master face. The opposite side of the master face will be the penetration side. See Remark 1. (Characters; default=BOTH)</p> <p>BOTH: The side from which a slave node approaches the master face will become the monitoring side.</p> <p>TOP: The monitoring side will always be on the side of the master face that its normal is pointing at.</p> <p>BOTTOM: The monitoring side is always on the opposite side of the master face that its normal is pointing at</p>

WEIGHT	<p>The contact force is multiplied by a mass-weighting factor. The following options are available. See Remark 25. (Character; default=BOTH)</p> <p>BOTH: $M_{scale} = (M_{slave} M_{master}) / (M_{slave} + M_{master})$</p> <p>SLAVE: $M_{scale} = M_{slave}$</p> <p>MASTER: $M_{scale} = M_{master}$</p> <p>NONE: $M_{scale} = 1.0$</p>
MONDIS	<p>Defines the fixed part of the monitoring distance. When the normal distance of a slave node to a master face becomes smaller than the monitoring distance the slave node will tag itself to the master face. The side from which the slave node is moving towards the master face becomes the monitoring region. (Character; default=FACTOR)</p> <p>FACTOR: The monitoring distance is equal to a factor times a characteristic length of the master faces. The factor is specified in MONDISV.</p> <p>DISTANCE: The monitoring distance is specified in MONDISV.</p>
MONDISV	Value of the monitoring distance or value of the FACTOR to calculate the monitoring distance. (Real; default=2.0)
INITMON	Fixed part of the monitoring distance used during the initialization. If not specified, the value of MONDIS is used. (Real > 0.0; default=MONDIS)

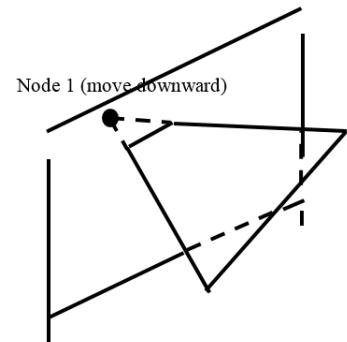
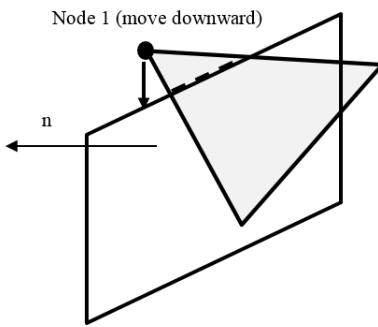
Remarks:

1. When METHOD=RELLIPS is used, BCGRID or BCMATL are only available for SLAVE body and BCELIPS is only available for MASTER body. When BCMATL is used, the MATRIG id or RBE2 id are only acceptable. When Euler solver is deactivated, no output will be generated for the Eulerian elements.
2. The options TOP/BOTTOM are useful in the following cases:
 - a. When a slave node initially is located on the master face (see the picture below), the contact situation is uniquely defined, only if the TOP or BOTTOM side of the master surface is defined:
SIDE=TOP case:



- b. When hooking of slave nodes on the wrong side of a master face might occur. This often is the case when the master face is at the edge of a shell element structure:

SIDE=BOTTOM case: penetration of node 1 (SIDE=BOTH: no penetration of node 1)



3. Recommended usage of WEIGHT:

SLAVE	MASTER	WEIGHT
Deformable	Deformable	BOTH
Deformable	Rigid	SLAVE
Rigid	Deformable	MASTER
Rigid	Rigid	NONE

Default setting when BCELIPS is used in SLAVE or MASTER:

SLAVE	MASTER	WEIGHT
Non-BCELIPS	Non-BCELIPS	NONE
	BCELIPS	

BCONPRP**Physical Contact Parameters of Touching Bodies in SOLs 101 and 400**

Defines physical contact parameters of touching bodies used in SOLs 101 and 400 only. The parameters defined here are referenced by the BCONECT entry. This entry is used in conjunction with the BCONPRG entry.

Format:

1	2	3	4	5	6	7	8	9	10
BCONPRP	PID		PARAM1	VAL1	PARAM2	VAL2	PARAM3	VAL3	
	PARAM4	VAL4	PARAM5	VAL5	-etc.-				

Example:

BCONPRP	90		FRIC	0.3					
---------	----	--	------	-----	--	--	--	--	--

Describer Meaning

PID Parameter identification number (Integer > 0).

PARAMi Name of a parameter. Allowabe names are given in [Table 9-6](#) (Character).

VALi Value of a parameter. See [Table 9-6](#) (Real or Integer).

Table 9-6 Physical Contact Parameters of Touching Bodies in SOLs 101 and 400

Name	Description, Type and Value (Default is 0 for integer, 0.0 for Real Unless Otherwise Indicated)
BGM	Exponent for the tangential stress term in deciding if a glue-contact will break. See Remark 1. (Real; Default = 2.0)
BGN	Exponent for the normal stress term in deciding if a glue-contact will break. See Remark 1. (Real; Default = 2.0)
BGSN	Maximum normal stress for breaking glue. See Remark 1. (Real; Default = 0.0)
BGST	Maximum tangential stress for breaking glue. See Remark 1. (Real; Default = 0.0)
BNC	Exponent associated with the natural convection coefficient for near field behavior. If Real, the value entered is the exponent associated with near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the exponent associated with the near field natural convection coefficient vs. temperature or a TABL3D entry specifying the exponent associated with the near field natural convection coefficient vs. temperature and possibly other variables. Ignored in a pure structural analysis. See Remark 5. (Real or Integer; Default = 1.0.)
BNL	Exponent associated with the nonlinear convective heat flow for near field behavior. If Real, the value entered is the exponent associated with the near field nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 OR TABLEM2 entry specifying the exponent associated with the near field nonlinear convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the near field nonlinear convection coefficient vs temperature and possibly other variables. Ignored in a pure structural analysis. See Remark 5. (Real or Integer; Default = 1.0)
DQNEAR	Distance below which near thermal contact behavior occurs. Used in heat transfer analysis only. See Remark 2. (Real; Default = 0.0; which means near contact does not occur)
EMISS	Emissivity for radiation to the environment or near thermal radiation. If Real, the value entered is the emissivity. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the emissivity vs. temperature or a TABL3D entry specifying the emissivity vs. temperature and possibly other variables. Ignored in a pure structural analysis. See Remark 5. (Real or Integer; Default = 0.0)
FNTOL	Separation force, stress, or fraction above which a node separates from a body. FNTOL is closely related to IBSEP. Its default value is dependent on the IBSEP value. (Real)
FRIC	Friction coefficient. If the value is an integer, it represents the ID of a TABLEM1, TABLEM2 or TABL3D, i.e., a temperature-dependent or multi-dimensional table. (Real ≥ 0.0 or Integer > 0 ; Default = 0.0)
FRLIM	Friction stress limit. This entry is only used for friction type 6 (Coulomb friction using the bilinear model). If the shear stress due to friction reaches this limit value, then the applied friction stress will be reduced so that the maximum friction stress is given by $\min(\mu\sigma_n, \sigma_{limit})$, with μ the friction coefficient and σ_n the contact normal stress. (Real; Default = 1.0E20)

HBL	Separation distance dependent thermal convection coefficient. If Real, the value entered is the convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the convection coefficient vs. temperature or a TABL3D entry specifying the convection coefficient vs. temperature and possibly other variables. Ignored in a pure structural analysis. See Remark 5. (Real or Integer; Default = 0.0)
HCT	Contact heat transfer coefficient. If Real, the value entered is the contact heat transfer coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the contact heat transfer coefficient vs. temperature or a TABL3D entry specifying the contact heat transfer coefficient vs. temperature and possibly other variables. Ignored in a pure structural analysis. See Remark 5. (Real or Integer; Default = 0.0; In a thermal/mechanical coupled analysis a default value of 1.0E6 is used when the mechanical glue option is activated, i.e., when IGLUE > 0.)
HCV	Convection coefficient for near field behavior. If Real, the value entered is the near field convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field convection coefficient vs. temperature or a TABL3D entry specifying the near field convection coefficient vs. temperature and possibly other variables. Ignored in a pure structural analysis. See Remark 5. (Real or Integer; Default = 0.0)
HGLUE	Flag to activate the thermal glue option. When left blank or set to zero, thermal contact conditions will be treated by convective heat transfer between the bodies. When set to 1, the temperature fields of the bodies are tied as soon as they come in contact and there will be no convective heat transfer over the body interfaces. Ignored in a pure structural analysis. See Remarks 3, 4, and 5. (0 ≤ Integer ≤ 1; Default = 0)
HNC	Natural convection coefficient for near field behavior. If Real, the value entered is the near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field natural convection coefficient vs. temperature or a TABL3D entry specifying the near field natural convection coefficient vs. temperature and possibly other variables. Ignored in a pure structural analysis. See Remark 5. (Real or Integer; Default = 0.0).
HNL	Heat transfer coefficient for nonlinear convective heat flow for near field behavior. If Real, the value entered is the near field nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field nonlinear convection coefficient vs. temperature or a TABL3D entry specifying the near field nonlinear convection coefficient vs. temperature and possibly other variables. Ignored in a pure structural analysis. See Remark 5. (Real or Integer; Default = 0.0)

Remarks:

- Breaking Glue provides glued contact to all GRID's at their very 1st contact. This kind of glued-contact will break if

$$(\sigma_n/BGSN)^{**}bgn + (\sigma_t/BGST)^{**}bgm > 1.0$$

When a contact node breaks due to the above criterion, standard contact is activated if it comes into contact again. If BGSN = 0.0 the first term is ignored. Similarly, the second term is ignored if BGST = 0.0. For SOL 400, parameters BGSN, BGST, BGM and BGN are only required if JGLUE = 2. If both BGSN and BGST are equal to 0.0, JGLUE will be set to 0 internally. If Step Glue is used, then Breaking Glue is inactive in this step. If Permanent Glue is active, then Breaking Glue is inactive.

2. For near contact the following convective heat flow law describes the heat exchange per unit area between the body areas (“near” contact is when bodies are not touching each other – but are close enough for convection and radiation.):

$$q = HCV \cdot (T_A - T_B) +$$

$$HNC \cdot (T_A - T_B)^{BNC} +$$

$$HNL \cdot (T_A^{BNL} - T_B^{BNL}) +$$

$$\sigma \cdot EMISS \cdot (T_A^4 - T_B^4) +$$

$$\left[HCT \cdot \left(1 - \frac{dist}{DQNEAR} \right) + HBL \cdot \frac{dist}{DQNEAR} \right] \cdot (T_A^4 - T_B^4)$$

where the last term is only activated when HBL ≠ 0, T_A is the contacting grid temperature and T_B is the face temperature in the contact point in case of a meshed body or the T_{BODY} temperature in case of a rigid geometry.

3. For hard contact (i.e., two bodies are actually touching each other – heat transfer mode between the bodies is conduction), with HGLUE=1:
 - a. The temperature of the contacting grid is tied to the temperatures of the contacted element face or the temperature of the rigid geometry when it has a scalar point associated with it.
 - b. The temperature of the contacting grid is set to the rigid geometry temperature when it has no scalar point associated with it.

Note: “Glued” thermal contact can result in overshoot of the temperatures i.e. Gibbs Phenomenon or the temperature oscillations at the interface, in particular, if two bodies that have non-uniform initial temperatures are placed in contact. The overshoot effect may be damped somewhat if one uses a near contact distance with some convective heat transfer.

4. For hard contact, with HGLUE=0:

The convective heat flow per unit area over the two interfaces is given by:

$$q = HCT \cdot (T_A - T_B)$$

where T_A is the contacting grid temperature and T_B is the face temperature in the contact point in case of a meshed body or the T_{BODY} temperature in case of a rigid geometry.

5. The heat transfer coefficients and associated exponents can all be temperature dependent, when they are entered as an integer value. This integer value is the table ID of a TABLEM1, TABLEM2 or TABL3D entry (formulas are not supported on TABL3D). Table IDs of tables used on the BCBDPRP entry and the BCONPRP entry must be unique.
6. Parameters BGM, BGN, BGSN, BGST and DQNEAR are not supported in segment to segment contact analysis.

BCONPRP-700**Physical Contact Parameters of Touching Bodies in SOL 700**

Defines physical contact parameters of touching bodies used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
BCONPRP	PID		PARAM1	VAL1	PARAM2	VAL2	PARAM 3	VAL3	
	PARAM4	VAL4	PARAM5	VAL5	-etc.-				

Example:

BCONPRP	90		FRIC	0.3					
---------	----	--	------	-----	--	--	--	--	--

Descriptor Meaning

PID Parameter identification number (Integer > 0).

PARAMi Name of a parameter. Allowable names are given in [Table 9-7](#) (Character).

VALi Value of a parameter. See [Table 9-7](#) (Real or Integer).

Table 9-7 Physical Contact Parameters of Touching Bodies in SOL 700

Name	Description, Type and Value (Default is 0 for integer, 0.0 for Real Unless Otherwise Indicated)
FRIC	Static friction coefficient. When it is set to -1.0, different static friction coefficients on properties can be assigned using BCPROP1. (Real \geq -1.0; Default=0.0 for SOL700)
FK	Kinetic coefficient of friction. (Real $>$ 0.0; Default = 0.0)
EXP	Exponential decay coefficient. (Real $>$ 0.0; Default = 0.0)
FACT	Scale factor for the contact forces. (Real $>$ 0.0; Default = 0.1)

TSTART	Time at which the contact is activated. (Real > 0.0; Default = 0.0)
TEND	Time at which the contact is deactivated. (Real > 0.0; Default = 1.e20)
IADJ	<p>Adjacent material treatment option for solid elements. (Integer ≥ 0; Default =1)</p> <p>0 Solid element faces are included only for free boundaries.</p> <p>1 Solid element faces are included if they are on the boundary of the material subset. This option also allows the erosion within a body and the subsequent treatment of contact.</p>
DAMPING	<p>Specifies if a high frequency damping is active or not. The damping force is based on the relative velocity of a slave node with respect to a master face. The damping is preferably turned on in all cases, except for RIGID-RIGID contact. In RIGID-RIGID contact it can result in a substantial loss of energy. VERSION V4 only. (Character; Default=YES).</p> <p>YES damping is activated</p> <p>NO damping is not activated.</p>

BCONUDS**Contact Body User Defined Service**

Allows the user to provide contact routines for use with enhanced SOL 400 contact analysis.

Format:

1	2	3	4	5	6	7	8	9	10
BCONUDS	BID	BTYPE	GROUP	UNAME					
	“INT”	IDATA1	IDATA2	IDATA3	IDATA4	IDATA5	IDATA6	IDATA7	
		IDATA8	IDATA9	IDATAn			
	“REAL”	RDATA1	RDATA2	RDATA3	RDATA4	RDATA5	RDATA6	RDATA7	
		RDATA8	RDATA9	RDATAn			
	“CHAR”	CDATA1	CDATA2	CDATAn			

Example:

In FMS Section of MSC Nastran input stream:

```
CONNECT SERVICE CONTACT 'SCA.MDSolver.Util.Ums'
```

In Bulk Data:

BCONUDS	17	BCBODY	CONTACT	UFRIC					
---------	----	--------	---------	-------	--	--	--	--	--

BCONUDS	17	BCBODY	CONTACT	SEPFOR					
	REAL	.00134	1.467+4	.03					

BCONUDS	17	BCBODY	CONTACT	MOTION					
	REAL	.00134	1.467+4	.03					
	INT	8	3						

Descriptor	Meaning
BID	Contact body identification number defined by BCBODY or BCBODY1. (Integer > 0; Required)
BTYPE	The name of the contact entry. BCBODY/BCBODY1 (Character; no Default)
GROUP	The group name used for the FMS Section CONNECT SERVICE statement. (Character; no Default)
UNAME	User subroutine name associated with the entry. See Remark 6. and 7. (Character)
“INT”	Keyword indicating that the following data is integer. (Character)
IDATAi	Additional user supplied Integer data not already existing on the specified contact property entry. (Integer, no Default)
“REAL”	Keyword indicating that the following data is real. (Character)

Descriptor	Meaning
RDATAi	Additional user supplied Real data not already existing on the specified contact property entry. (Real; no Defaults)
"CHAR"	Keyword indicating that the following data is Character. (Character)
CDATAi	Additional user supplied Character data not already existing on the specified contact property entry. (Character; no Default)

Remarks:

1. This entry triggers the call to a user contact subroutine for advanced nonlinear materials. The GROUP must match the GROUP field of the CONNECT SERVICE FMS statement.
2. On the FMS CONNECT statement, only the CONNECT SERVICE can be used with this entry.
3. The BID must match an existing BID.
4. A CDATAi entry cannot be the Character "REAL", "INT", or "CHAR".
5. Certain user subroutines may require integer or real data input as specified in the User Defined Services manual.
6. UNAME must be truncated to 8 characters in the bulk data field
7. The following user subroutines are currently available for user convenience. See the User Defined Services manual for details.

Type	Uname	Purpose
BCBODY	motion	Define velocity of rigid surfaces
BCBODY	ufric	Define friction coefficients
BCBODY	sepfor	Define separation forces
BCBODY	sepstr	Define separations stresses
BCBODY	ubsqueal	Define friction coefficient and effective stiffness in brake squeal analysis

BCONUDS**Contact Body User Subroutines in SOL 600**

Allows the user to provide contact routines for use with SOL 600 contact analysis.

Format:

1	2	3	4	5	6	7	8	9	10
BCONUDS	BID	BTYPE		UNAME					
	“INT”	IDATA1	IDATA2	IDATA3	IDATA4	IDATA5	IDATA6	IDATA7	
		IDATA8	IDATA9	IDATAn			
	“REAL”	RDATA1	RDATA2	RDATA3	RDATA4	RDATA5	RDATA6	RDATA7	
		RDATA8	RDATA9	RDATAn			

Example:

BCONUDS	17	BCBODY		RTN1					
---------	----	--------	--	------	--	--	--	--	--

BCONUDS	17	BCBODY		RTN2					
	REAL	.00134	1.467+4	.03					

BCONUDS	17	BCBODY		RTN3					
	REAL	.00134	1.467+4	.03					
	INT	8	3						

Descriptor	Meaning
BID	Contact body identification number defined by BCBODY. (Integer > 0; Required)
BTYPE	The name of the contact entry. BCBODY (Character; no Default)
UNAME	The Marc user subroutine name. See Remarks 2. and 5. (Character; Required)
“INT”	Keyword indicating that the following data is integer. (Character)
IDATAi	Additional user supplied Integer data not already existing on the specified MAT entry. See Remark 6. (Integer; no Default)
“REAL”	Keyword indicating that the following data is real. (Character)
RDATAi	Additional user supplied Real data. See Remark 6. (Real; no Defaults)

Remarks:

1. The BID must match an existing BID.
2. Certain user subroutines may require integer or real data input as specified in the UserSubroutine interface document.
3. The following user subroutines are currently available for user convenience. See the User Subroutine interface document for details.

UNAME	Purpose
MOTION	Define velocity of rigid surfaces
UFRIC	Define friction coefficients
DIGEOM	Define rigid surfaces
SEPFOR	Define separation forces
SEPPFORBBC	Define separation forces for beam-beam contact
SEPSTR	Define separations stresses
UFRICBBC	Define the friction coefficients for beam-to-beam contact with node-to-segment contact. This user subroutine is not used for segment-to-segment contact; use UFRIC instead.
UHTNRC	Define thermal near contact film coefficients
UMDCOE	Define variable mass and diffusion coefficients and sink pressures on a free surface
UMDCON	Define variable mass diffusion coefficients on surfaces in contact with other surfaces
UMDNRC	Define mass diffusion coefficients of surfaces in contact with other surfaces
UNDRST	Define normal stresses for elements in contact

4. The BCONUDS entry may be used instead of the USRSUB6 entry. Both entries should not be used in the same run. EVAL will be stored as a character*16 name in common block /userch/
5. UNAME identifies the user subroutine name to be called.
6. IDATA and RDATA are not normally required. They are available should additional arguments beyond those described in Marc Vol D be required. Normally only the first line is entered.

BCPARA

Contact Parameters -SOLs 101 and 400

Defines contact parameters used in SOL 101 and SOL 400. This entry is not available in SOL 700.

Format:

1	2	3	4	5	6	7	8	9	10
BCPARA	ID	Param1	Value1	Param2	Value2	Param3	Value3		
	Param4	Value4	Param5	Value5	etc.				

Example:

BCPARA	ERROR	0.1	BIAS	0.5				
--------	-------	-----	------	-----	--	--	--	--

Descriptor	Meaning
ID	Subcase to which the defined parameters belong. If ID is zero, the parameters belong to all subcases. (Integer ≥ 0 ; no Default)
Param(i)	Name of a parameter. Allowable names are given in Table 9-7 . (Character)
Value(i)	Value of the parameter. See Table 9-7 . (Real or Integer)

Table 9-8 Contact Parameters -SOLs 101 and 400

Name	Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)	
ERROR	Distance below which a node is considered touching a body. (Real; Default = blank). Automatically calculated if left blank. If left blank, the code calculates ERROR as the smallest value resulting from: Either dividing the smallest nonzero element dimension (plates or solids) in the contact body by 20. Or dividing the thinnest shell thickness in the contact body by 4. This value is then used for all contact pairs.	
BIAS	Contact tolerance bias factor. (0.0 ≤ Real ≤ 1.0.); Default = 0.9 for IGLUE=0, if field left blank. Default = 0.0 for IGLUE <>0. Note that when IGLUE<>0, BIAS can only be given by the BCTABLE or BCONPRG. Default = 0.0 for BEHAVE=SYMM on BCBODY, if field left blank or 0.0.	
FGCFLG	Flag to activate Cohesive (Flexible) Glued Contact: (See Remark 12.)	
	0	No Cohesive (Flexible) Glued Contact
	1	Activate Cohesive (Flexible) Glued Contact
FGCNST	Equivalent normal contact stiffness of connector for Cohesive (Flexible) Glued Contact	
FGCTST	Equivalent tangential contact stiffness of connector for Cohesive (Flexible) Glued Contact	
FGCNSTR	ID of table for define contact normal stress versus relative displacement for Cohesive (Flexible) Glued Contact. Only TABL3D allowable.	
FGCTSTR	ID of table for define contact tangential stress versus relative displacement for Cohesive (Flexible) Glued Contact. Only TABL3D allowable.	
FGCNSTI	ID of table for define contact normal stiffness versus relative displacement or temperature for Cohesive (Flexible) Glued Contact. Only TABL3D allowable.	
FGCTSTI	ID of table for define contact tangential stiffness versus relative displacement or temperature for Cohesive (Flexible) Glued Contact. Only TABL3D allowable.	
FGCRCEN	ID of grid on which the resultant contact and moment are based.	
FNTOL	Separation force (or stress if separation is controlled by stress as determined by IBSEP) above which a node separates from a body. Automatically calculated if left blank. (Real; Default = blank). See Remark 10.	
MAXSEP	Maximum number of separations allowed in each increment. After MAXSEP separations have occurred, if the standard convergence tolerance conditions are achieved, the step will converge. (Integer > 0; Default = 9999)	

Name	Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)	
ICSEP	Flag to control separation. Not used for segment-to-segment contact. (Integer ≥ 0 ; Default = 0)	
	0	The node separates and an iteration occurs if the force on the node is greater than the separation force.
	1	If a node which was in contact at the end of the previous increment has a force greater than the separation force, the node does NOT separate in this increment, but separates at the beginning of the next increment.
	2	If a new node comes into contact during this increment, it is not allowed to separate during this increment, prevents chattering.
	3	Both 1 and 2 are in effect.
IBSEP	Flag for separation based on stresses or forces. (Integer ≥ 0 ; Default = 0)	
	0	Separation based on forces.
	1	Separation based on absolute stresses (force/area)
	2	Separation based on absolute stress (extrapolating integration point stresses)
	3	Relative nodal stress (force/area)
	4	Separation based on relative stress (extrapolating integration point stresses)
	Segment to segment contact ignores IBSEP and will use 2 internally. See Remarks 6 and 8 .	
RVCNST	For the bilinear friction model, RVCNST is the slip threshold. (Real ≥ 0.0 ; Default = 0.0).	
	Default=0.0 means that the actual value is automatically calculated.	
FTYPE	Friction type. See Remark 5 . (Integer)	
	0	No friction. (Default)
	6	Bilinear Coulomb friction.
	7	Bilinear Shear friction.
BEAMB	Beam-Beam contact flag. See Remark 7 . (Integer 0 or 1)	
	0	No beam-beam contact. (Default for SOLs 101/400)
	1	Activate beam-beam contact options. (Default for SOL 700)

Name	Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)	
NLGLUE (SOLs 101 and 400 only)	If all slave's for the BCTABLE or BCONPRG corresponding to the first loadcase (first subcase and first step) contain IGLUE >0, permanent glued contact with small rotation condition will be used for all SLAVE entries in all subcases and all steps unless BCPARA,0,NLGLUE,1 is specified. If IGLUE < 0 exists, permanent glued contact for large deformation and large rotation is activated. (Integer; Default = 0)	
NODSEP	Skip separation check in contact for any grid already touched and separated. NODSEP times within the current increment. (Integer; Default = 5)	
	> 0	Keep grid contact status
	< 0	Separate grid from contact
METHOD	Flag to select Contact methods. (Character)	
	NODESURF	Node to segment contact. (Default)
	SEGTOSSEG	Segment to segment contact.
AUGMENT	Augmentation method used in a segment-to-segment contact analysis. (Integer)	
	0	No augmentation (Default)
	1	Augmentation based on a constant Lagrange multiplier field for linear elements and on a (bi)linear Lagrange multiplier field for quadratic elements
	2	Augmentation based on a constant Lagrange multiplier field
	3	Augmentation based on a (bi)linear Lagrange multiplier field
PENALT	Augmented Lagrange penalty factor; used by the segment-to-segment contact algorithm only. (Real > 0.0; see BCTABLE or BCONPRG entries for default)	
AUGDIST	Penetration distance beyond which an augmentation will be applied; used by the segment-to-segment contact algorithm only. (Real > 0; see BCTABLE entry for default)	
SLDLMT	Maximum allowed sliding distance, beyond it the contact segments are redefined, for segment to segment contact analysis with large deformation. (Real \geq 0.0; Default = 0.0) See remark 9.	
SEGSYM	Specify symmetric or non-symmetric friction matrix in segment to segment contact analysis. (Integer 0 = symmetric matrix or 1 = non-symmetric matrix; Default = 0)	
ERRBAS	Error computation option. (Integer 0 = compute error globally or 1 = calculate error based on each pair of slave-master; Default = 0)	
DDULMT	Maximum value of DDU in a Newton-Raphson cycle. (Real \geq 0.0; Default = 0.0, no limitation)	
TAUGMNT	Augmentation for the sticking part of friction in a segment-to-segment contact analysis. (Integer 0 = no augmentation or 1 = use augmentation; Default = 0)	

Name	Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)
TPENALT	Augmented Lagrange penalty factor for sticking part of friction, used by the segment-to-segment contact algorithm only. (Real > 0.0) The default is PENALT/1000, where PENALT parameter is the Augmented Lagrange penalty factor for normal contact.
STKSLP	Maximum allowable slip distance for sticking, beyond it there is no sticking, only sliding exists, used by the segment-to-segment contact algorithm only. (Real \geq 0.0; Default = 0.0) See Remark 11..
THKOFF	Ignore thickness from the tolerance used by ISEARCH=2 in node-to-surface contact or from the characteristic length (for PENALT and AUGDIST) in segment-to-segment contact. (Integer 0 = do not ignore thickness or 1 = remove thickness; Default = 0)
LINCNT	Flag for Linear Contact (under infinitesimal assumption, small sliding with small deformation and rotation). Supports both node-to-segment and segment-to-segment method. See Remark 14.. 0 - (Default) general contact 1 - activate Linear Contact, the contact force is distributed based upon undeformed geometry. -1 - force Linear Contact with large displacement (not recommended).
SFNPNLT	Scale factor of augmented Lagrange penalty factor along contact normal direction; used by segment-to-segment contact method only. (Real>0.0;Default=1.0) SFNPNLT*PENALT is used in analysis. A nonblank value of SFNPNLT entered in BCONPRG will override it.
SFTPNLT	Scale factor of augmented Lagrange penalty factor along contact tangential direction; used by segment-to-segment contact method only. (Real>0.0;Default=1.0) SFTPNLT*TPENALT is used in analysis. A nonblank value of SFTPNLT entered in BCONPRG will override it.
SEGANGL	Minimum angle between segment normal vectors allowing the segments to come into contact; used by segment-to-segment contact only. (Real>90.0 <180.0; Default=120.0).

Remarks:

1. The ID of BCPARA should be left blank or a value of zero. Blank is treated as zero.
2. Only one BCPARA entry should be made. Multiple BCPARA entries with ID=0 cause a Fatal Error.
3. All SOLs only support ID=0, this entry is applied to all the subcases, except the corresponding ones are assigned in BCTABLE or BCONPRG.
4. Refer to the Case Control command [BCONTACT \(Case\)](#), to see the lists of parameters in BCPARA that are not supported by SOLs 101 and 400.

5. In SOLs 101/400, FTYPE only supports options 0, 6, and 7. Friction with segment-to-segment contact is a pre-release capability in the 2012 release. In the 2012.2 version and later, it is fully supported.
6. Shell elements (CQUAD4 CQUAD8 CTRIA3 CTRIA6) in 3D Contact only supports IBSEP = 0, 1 and 3. If IBSEP = 2 or 4 in this case, it will automatically be set to 1 or 3, respectively. In SOLs 101/400, LINQUAD is not supported and it is set to -1 in order that the mid-side nodes are considered in Contact. Quadratic shell elements consider mid-side nodes only in 2D Contact.
 - a. If IBSEP is set to 1, then for bodies consisting of mid-side node elements, IBSEP will automatically be set to 2. This allows for using IBSEP=1 for elements without mid-side nodes and IBSEP=2 for elements with mid-side nodes in one analysis.
 - b. If IBSEP is set to 3, then for bodies consisting of mid-side nodes elements, IBSEP will automatically be set to 4. This allows for using IBSEP=3 for elements without mid-side nodes and IBSEP=4 for elements with mid-side nodes in one analysis.

This rule is always applied to the solid elements with mid-side nodes. However, the mid-side nodes of CQUAD8 and CTRIA6 shell elements are ignored in node-to-segment 3D contact with separation because there is no normal stress in shell elements, and the contact normal forces for these elements do not give a good estimation of the nodal contact stress. Hence it is not recommended to use CQUAD8 and CTRIA6 shell elements in a node-to-segment contact analysis with separation.

7. For SOLs 101/400 if BEAMB=1, Bulk Data entry BCBMRAD must be entered. Those entries are not required in the beam contact of segment-to-segment contact.
8. Parameters MAXSEP, ICSEP, IBSEP, RVCNST and BEAMB are not supported in segment to segment contact analysis.
9. The default of SLDELM parameter is 5 times the default error tolerance. See the description of ERROR parameter for the definition of default error tolerance.
10. IBSEP and FNTOL are related to each other. The Default value of FNTOL is dependent on IBSEP value.
 - a. IBSEP=0 : Separation is based on forces. A contact node will separate if the contact pulling force exceeds FNTOL. If FNTOL is zero it will be taken as the absolute value of the largest component in the residual force vector.
 - b. IBSEP=1 : Separation is based on nominal contact stresses (force/area). A contact node will separate if the nominal contact pulling stress exceeds FNTOL. If FNTOL is zero, it will be 0.1 times of the maximum nominal contact pushing stress of any node of any contact body.
 - c. IBSEP=2 : Separation is based on nodal stress (extrapolating integration point stresses). A contact node will separate if the nodal pulling stress exceeds FNTOL. If FNTOL is zero it will be 0.1 times of the maximum nodal pushing stress of any node of any contact body
 - d. IBSEP=3 : Separation is based on relative nominal contact stress (force/area). A contact node will separate if the nominal contact pulling stress exceeds FNTOL times of maximum nominal contact pushing stress of any node of any contact body. If FNTOL is zero, it will be taken as 0.1
 - e. IBSEP=4 : Separation is based on relative nodal stress (extrapolating integration point stresses). A contact node will separate if the nodal pulling stress exceeds FNTOL times of maximum nodal pushing stress of any node of any contact body. If FNTOL is zero, it will be taken as 0.1

11. If STKSLP is set to 0.0 (Default), the sticking stiffness K1 is equal to the maximum friction force ($\mu \cdot F_{NORMAL}$, where μ is the friction coefficient) divided by the maximum sticking displacement. Otherwise, K1 is equal to the maximum friction force divided by the value of STKSLP.
12. The parameters of cohesive contact defined in BCPARA will be applied to all the contact pairs unless:
 - The contact pair has its own definition for cohesive contact in BCONPRG; or
 - Its IGLUE=0, i.e., general touching contact.Cohesive contact is only applied to glued contact, but not touching contact.
13. If Modules are present then this entry may only be specified in the main Bulk Data section.
14. LINCNT does not support RIGID contact body with velocity control.

BCPARA**Contact Parameters in SOL 600**

Defines contact parameters used in SOL 600.

Format:

1	2	3	4	5	6	7	8	9	10
BCPARA	ID	Param1	Value1	Param2	Value2	Param3	Value3		
	Param4	Value4	Param5	Value5	etc				

Example:

BCPARA		NBODIES	4	BIAS	0.5				
--------	--	---------	---	------	-----	--	--	--	--

Descriptor	Meaning
ID	ID is not used and should be set to zero. Only one BCPARA should be entered and it applies to all subcases. (Integer)
Param(i)	Name of a parameter. Allowable names are given in Table 9-7 . (Character)
Value(i)	Value of the parameter. See the Table 9-7 . (Real or Integer)

Name	Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)	
NBODIES (2,1)	Number of contact bodies defined in the analysis. (Integer ≥ 0 or blank)	
MAXENT (2,2)	Maximum number of entities created for any contact body. (Integer > 0 or blank; default is max element number or 1.5 times the number of nodes whichever is smaller)	
MAXNOD (2,3)	Maximum number of nodes that lie on the periphery of any deformable contact body. (Integer > 0 or blank; default is the number of nodes)	
ERROR (3,2)	Distance below which a node is considered touching a body. Automatically calculated if left blank. If left blank, the code calculates ERROR as the smallest nonzero element dimension divided by 20 or the shell thickness divided by 4. (Real; Default = blank)	
BIAS (3,6)	Contact tolerance bias factor. (Real, $0 \leq \text{BIAS} \leq 1$, Default = 0.9 if field left blank or not entered in the file. To obtain a near zero value, enter 1.0E-16)	
ISPLIT (2,7)	Flag for increment splitting procedure. (Integer ≥ 0 ; Default = 3 for statics and 0 for dynamics)	
	0	Uses increment splitting procedures for the fixed time step procedures.

Name	Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)	
	1	Suppresses splitting for the fixed time step procedures. Any penetration that occurred in the previous increment is adjusted for equilibrium at the start of the next increment. This method may require smaller time steps than the other methods
	2	Suppresses splitting for adaptive time step procedures. Any penetration that occurred in the previous increment is adjusted for equilibrium at the start of the next increment. This method may require smaller time steps than the other methods.
	3	Uses contact procedure which does not require increment splitting (3 is not available for dynamics). If a run does not converge due to an excessive number of “iterative penetration checking” messages, ISPLIT=2 may help, however the time steps may need to be set to smaller values.
FNTOL (3,5)		Separation force (or stress if separation is controlled by stress as determined by IBSEP) above which a node separates from a body. Automatically calculated if left blank. (Real; Default = blank)
MAXSEP (2,6)		Maximum number of separations allowed in each increment. After MAXSEP separations have occurred, if the standard convergence tolerance conditions are achieved, the step will converge. (Integer > 0; Default = 9999)
ICHECK (2,8)		Flag for interference kinematic check and bounding box check. (Integer > 0)
	1	Activates interference kinematic check.
	2	Suppress bounding box checking.
	3	No reset of NCYCLE to zero.
	4	Check for separation only when solution has converged, for analytical surfaces only.
ICSEP (2,9)		Flag to control separation. (Integer ≥ 0 ; Default = 0)
	0	The node separates and an iteration occurs if the force on the node is greater than the separation force.
	1	If a node which was in contact at the end of the previous increment has a force greater than the separation force, the node does NOT separate in this increment, but separates at the beginning of the next increment.
	2	If a new node comes into contact during this increment, it is not allowed to separate during this increment, prevents chattering.
	3	Both 1 and 2 are in effect.
IBSEP (2,12)		Flag for separation based on stresses or forces. (Integer ≥ 0 ; Default = 0)
	0	Separation based on forces.

Name	Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)	
	1	Separation based on absolute stresses (force/area).
	2	Separation based on absolute stress (extrapolating integration point stresses).
	3	Relative nodal stress (force/area).
	4	Separation based on relative stress (extrapolating integration point stresses).
		Only option 2 and 4 can be used with mid-side node elements where the mid-side nodes contact (LINQUAD=-1).
ISHELL (2,10)		Parameter governing normal direction and thickness contribution of shells. (Integer ≥ 0 ; Default = 0)
	0	Check node contact with top and bottom surface.
	1	Nodes only come into contact with bottom layer.
	2	Nodes only come into contact with bottom layer and shell thickness is ignored.
	-1	Nodes only come into contact with top layer.
	-2	Nodes only come into contact with top layer and shell thickness is ignored.
IPRINT (2,11)		Flag to reduce print out of surface definition. (Integer ≥ 0 ; Default = 0)
	0	Full print out.
	1	Reduced print out.
RVCNST (3,1)		Relative sliding velocity between bodies below which sticking is simulated. If FTYPE=5, then the value of RVCNST is the stick-slip transition region.
FTYPE (2,4)		Friction type. (Integer)
	0	No friction. (Default)
	1	Shear friction.
	2	Coulomb Friction.
	3	Shear friction for rolling.
	4	Coulomb friction for rolling.
	5	Stick-slip Coulomb friction.
	6	Bilinear Coulomb friction. (Default if friction is present and FTYPE is not entered.)
	7	Bilinear Shear friction.
FKIND (2,5)		Friction kind. (Integer 0 or 1)

Name	Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)	
	0	Friction based on nodal stress.
	1	Default if friction is present and FKIND is not entered. Friction based on nodal force.
BEAMB (2,13)	Beam-Beam contact flag. (Integer 0 or 1)	
	0	No beam-beam contact. (Default)
	1	Activate beam-beam contact options.
FSSMULT (3,7)	Stick-slip friction coefficient multiplier. Applicable only to stick-slip friction. (The friction coefficient is multiplied by this value for the sticking condition.) (Real ≥ 0 ; Required; Default = 1.05)	
FSSTOL (3,8)	Stick-slip friction force tolerance. Applicable only to stick-slip friction. (Real; Default = 0.05)	
LINQUAD (2,14)	Higher order element contact flag (Integer; Default = 1).	
	1	The outer boundary of a contact body is described by the corner nodes only and mid-side nodes can't come into contact.
	-1	The outer boundary is described by a quadratic field and both corner and mid-side nodes are considered in contact. If this flag is set to -1 and IBSEP is blank, IBSEP will be re-set to 2. This option is only available with Marc 2003 and subsequent releases.
INITCON (2,16)	If INITCON is set, tying relations (MPC's) for surfaces initially in contact will be saved. This option maybe used to model dissimilar meshes. See CONTINUE=101+ on the SOL 600 entry to use these items in the same MSC Nastran execution. The following options are available:	
	1	MPC's in Marc format are saved for initial contact (if any) in file jid.marc.t01.
	3	MPC's for each increment are saved in Marc format in file jid.marc.conmpc_xxxx where xxxx is the increment number.
	4	MPC's for each increment are saved in Nastran MPC format in file jidd.marc.conmpc_xxxx where xxxx is the increment number.
	When initcon=1, the job will stop at the end of increment zero. When initcon=3 initcon=4, the job will run to completion and the information in jid.marc.conmpc_0000 for increment zero will usually not be useful, but the information for increments one and above will contain the proper contact tying relations or mpc's.	
NVSURF	Rigid contact surface ID (BCBODY ID) for which the next 4 approach velocity values apply. Leave this entry and the next 4 out if not applicable. This entry and the next 4 may be repeated as many times as necessary to define all rigid contact surfaces with approach velocity values. (Integer; Default = 0)	

Name	Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)	
VELAPP1 (6,1)	Approach velocity in direction 1 of rigid contact surface - used only in increment zero. Requires PARAM,MRCONVER,11 (Real; Default = 0.0)	
VELAPP2 (6,2)	Approach velocity in direction 2 of rigid contact surface - used only in increment zero. Requires PARAM,MRCONVER,11 (Real; Default = 0.0)	
VELAPP3 (6,3)	Approach velocity in direction 3 of rigid contact surface - used only in increment zero. Requires PARAM,MRCONVER,11 (Real; Default = 0.0)	
VELAPAN (6,3)	Approach angular velocity about local axis through center of rotation of rigid contact surface - used only in increment zero. Requires PARAM,MRCONVER,1 (Real; Default = 0.0)	
NODSEP	If NODSEP is positive, ignore separation of any node that has touched and separated NODSEP times (corresponds to feature, 31xx where xx=nodsep). If NODSEP is negative, NODSEP is the number of separations of a particular node allowed. If more separations of that node actually occur, the node is assumed to be separated (corresponds to feature, -31xx where xx=nodsep). (Integer between 0 and 99; Default if not entered is 2, corresponds to feature,3102)	
METHOD	Flag to select Contact Methods (Character)	
	NODES URF	Regular 3D contact (Default: node to surface contact)
	SEGSMA LL	Segment to segment contact with small sliding
AUGMENT	Augmentation method used in a segment to segment contact analysis. (Integer)	
	0	No augmentation (Default)
	1	Augmentation based on a constant Lagrange multiplier field for linear elements and on a bilinear Lagrange multiplier field for quadratic elements.
	2	Augmentation based on a constant Lagrange multiplier field.
	3	Augmentation based on a bilinear Lagrange multiplier field.
PENALT	Augmented Lagrange penalty factor used by the segment to segment contact algorithm only. (Real > 0; set BCBODY for Default)	
AUGDIST	Penetration distance beyond which an augmentation will be applied. Used by the segment to segment contact algorithm only. (Real > 0.0; set BCBODY for Default)	

Remarks:

- (i,j) refers to data block i and field j of the CONTACT model definition option in Marc.
- Field 2 of the primary line should be left blank or a value of zero should be entered.
- Only one BCPARA entry should be made. If multiple entries are made, the last will be used.

4. For FTYPE, SOLs 400 and 600 differ as follows. For SOL 400, if friction is entered but FTYPE is blank, friction is ignored. For SOL 600, if friction is entered but FTYPE is blank COULOMB friction is used. (FTYPE is reset internally to 2). In addition SOL 400 can only use friction types 0, 6 and 7.

BCPATCH

Defines a Rigid Contact Body Made up of Quadrilateral Patches in SOLs 101 and 400

Defines a rigid contact body made up of quadrilateral patches used in SOLs 101 and 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
BCPATCH	RBID								
	IDP	G1	G2	G3	G4				
	IDP	G1	G2	G3	G4				
	-etc.-								

Example:

BCPATCH	12								
	1	101	102	103	104				

Descriptor Meaning

RBID Unique identification number referenced by a BCRGSRF or BCBODY1 entry (Integer > 0). See Remark 1.

IDP ID of the patch (Integer number 1 through highest value).

G1, G2, G3,
G4 Grid numbers for each of the 4 nodes of the patch.

Remarks:

1. If BCRGSRF entry does not exist, BCPATCH entry will be referenced by the BCBODY1 entry directly.

BCPFLG

Defines a beam section segment flag

Defines a beam section branch and segment for segment to segment beam contact used in SOL 101 and 400 for general contact or in SOLs 101, 103, 105, 107 - 112, 200 and 400 for the permanently glued Contact.

Format:

1	2	3	4	5	6	7	8	9	10
BCPFLG	PID	IBRNCH1	IOUTIN 1	IBRNCH2	IOUTIN2	IBRNCHi	IOUTINi		

Example:

BCPLAG	2	1	2	3	13				
--------	---	---	---	---	----	--	--	--	--

Descriptor Meaning

PID Beam property/section identification number of PBEAML/PBTRL. (Integer > 0; Required)

IBRNCHi Branch selection flag for segment generation. Only required for thin wall sections. See Remark 1, 2, 3 and 4. (Integer; default=0).

0 select all branches for thin wall sections

IOUTINi Beam segment location. See Remark 4, 5, 6 and 7. (Integer; default=0; up to three unique integers may be placed in the field with no embedded blanks.)

0 all segments

1 location 1 of thin wall section or only outer of solid or tube sections

2 location 2 of thin wall section or only inner of tube section

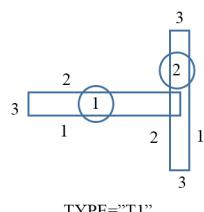
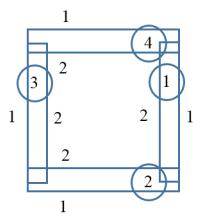
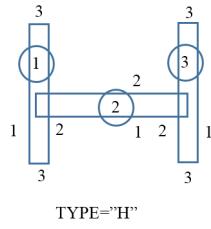
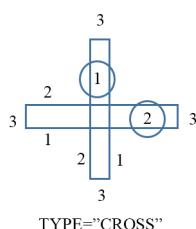
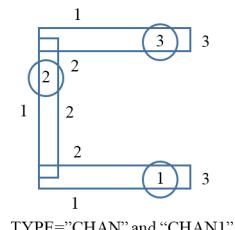
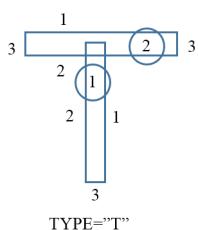
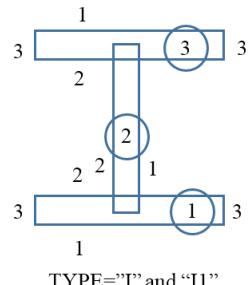
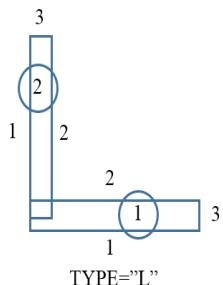
3 location 3 of thin wall section (thickness direction of branch)

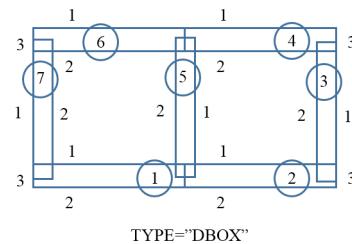
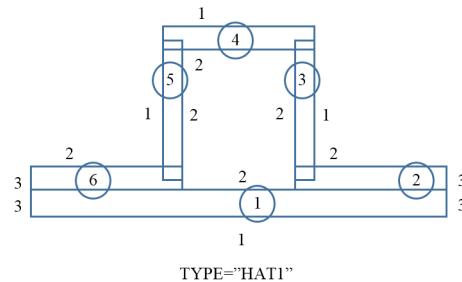
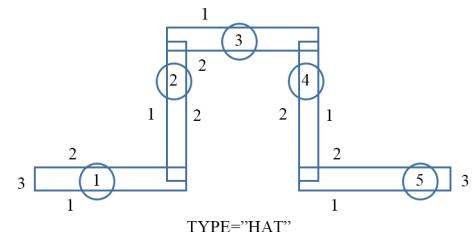
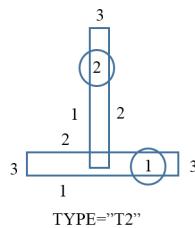
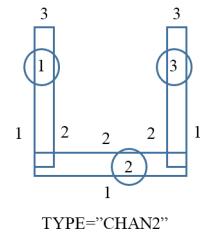
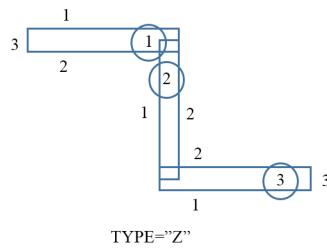
4 segment not including thickness

Remarks:

1. If the card is not defined but the beam property is included in segment to segment contact, all segments will be generated and used. This card is only useful when a user needs to define the specific beam segments for segment to segment contact. It will help the performance of calculation because only selected segments will be considered in the contact calculation.
2. IBRNCHi fields are only required for thin wall beam sections. Solid beam sections such as rectangular, rod, etc or tube sections require only IOUTIN1.
3. Each branch of thin wall beam section has four segments.
4. Segment locations of thin wall beam sections are listed below. TYPE is defined by PBEAML/PBTRL. The number circled is the branch number of each shape and the number without circle is the segment number which is used in IOUTINi. Segment 1 and 2 of each branch are the segments in the branch direction and segment 3's are the segment in the thickness direction of the branch.

Defines a beam section segment flag





5. If there is not thickness, one segment will be generated in the branch.
6. "3" cannot be used alone and "4" must be used only with "1" or "2". IOUTINi must be one of "0", "1", "2", "12", "13", "14", "23", "24" and "123".
7. "14" and "24" generate the identical segment since the thickness is ignored but segment normal direction will be different.

BCPROP**3D Contact Region by Element Properties**

Defines a 3D contact region by element properties. All elements with the specified properties define a contact body used in SOL 101 and SOLs 400 and 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
BCPROP	ID	IP1	IP2	IP3	IP4	IP5	IP6	IP7	
	IP8	IP9	etc.						

Example:

BCPROP	1	101	201	301					
--------	---	-----	-----	-----	--	--	--	--	--

Alternate Format:

BCPROP	ID	IP1	THRU	IP2	IP3	IP4	IP5	IP6	
	IP7	IP8	IP9	etc.					

Alternate Example:

BCPROP	1	101	THRU	102	105	THRU	109	110	
--------	---	-----	------	-----	-----	------	-----	-----	--

Descriptor	Meaning
ID	Identification number of a deformable surface corresponding to a BSID value on the BCBODY entry. All elements corresponding to the property IDs specified that may potentially come into contact. Do not specify mixed property types (use all shell, all solid or all beam properties only). See Remark 1. (Integer > 0)
IPi	Property ID. A minimum of one entry is required. (Integer; no Default)

Remarks:

1. ID must be unique with respect to all other BSURF, BCBOX, BCPROP, and BCMATL entries.
2. The deformable surface may alternately be defined using BSURF, BCBOX, or BCMATL entries.
3. Only one kind of entry (BSURF, BCBOX, BCPROP, or BCMATL) may be used to define a particular deformable surface.
4. All elements corresponding to the IDs entered will be used to define the deformable surface.
5. As many continuation lines as necessary may be used to define all property IDs associated with a particular deformable body.
6. The alternate format is triggered if field 4 contains THRU. The THRU keyword may appear in fields 4 thru 8.
7. BCBOX and BCMATL are not available for SOL 101 or SOL 400

BCPROP**3D Contact Region by Element Properties in SOL 600**

Defines a 3D contact region by element properties. All elements with the specified properties define a contact body used in SOL 600.

Format:

1	2	3	4	5	6	7	8	9	10
BCPROP	ID	IP1	IP2	IP3	IP4	IP5	IP6	IP7	
	IP8	IP9	etc.						

Example:

BCPROP	1	101	201	301					
--------	---	-----	-----	-----	--	--	--	--	--

Alternate Format:

BCPROP	ID	IP1	THRU	IP2					
--------	----	-----	------	-----	--	--	--	--	--

Example for Alternate Format:

BCPROP	25	101	THRU	102					
--------	----	-----	------	-----	--	--	--	--	--

Descriptor Meaning

ID Identification number of a deformable surface corresponding to a BSID value on the BCBODY entry or if the Case Control BCCONTACT = BCPROP is specified. All elements corresponding to the property IDs specified that may potentially come into contact. Do not specify mixed property types (use all shell, all solid or all beam properties only). See Remark 1. (Integer > 0)

IPi Property ID. A minimum of one entry is required. (Integer; no Default)

Remarks:

1. ID must be unique with respect to all other BSURF, BCBOX, BCPROP, and BCMATL entries.
2. The deformable surface may alternately be defined using BSURF, BCBOX, or BCMATL entries.
3. Only one kind of entry (BSURF, BCBOX, BCPROP, or BCMATL) may be used to define a particular deformable surface.
4. All elements corresponding to the IDs entered will be used to define the deformable surface.
5. As many continuation lines as necessary may be used to define all property IDs associated with a particular deformable body.
6. The alternate format is triggered if field 4 contains THRU.

BCRIGID

Defines a Rigid Contact Body in SOLs 101 and 400

Defines a rigid contact body used in SOLs 101 and 400 only.

Format: (SOLs 101 and 400 only)

1	2	3	4	5	6	7	8	9	10
BCRIGID	BCRGID	CGID	CONTROL						
	NLOAD	ANGVEL	DCOS1	DCOS2	DCOS3	VELRB1	VELRB2	VELRB3	
	"APPROV"	A	N1	N2	N3	V1	V2	V3	
	"GROW"	GF1	GF2	GF3	TAB-GF1	TAB-GF2	TAB-GF3		

Example:

BCRIGID	1001	1	1						
	0	0.	0.	0.	1.	1.	1.	1.	

Describer **Meaning**

BCRGID Unique identification number referenced by a BCBODY1 entry. (Integer > 0)

CGID Grid point identification number defining the initial position of the reference point of the rigid body or the point where a concentrated force or moment is applied. (Integer > 0) See Remark 1.

CONTROL Indicates the type of control for the body.

Integer:

= -1 for position control. The coordinates of the final position of GRID Point defined in CGID is given in VELRB_i in the following line.

= 0 for velocity control. (default)

= positive number for "load control". The positive number is the grid number defined in CGID at which translational forces or SPCD are applied. (Note: The rotation in this case is defined by NLOAD in the following line.)

NLOAD Enter a positive number if "load controlled" and rotations are allowed. (Integer) The positive number is the grid number where the moments or rotations are applied. The rotations are specified using SPCD at grid ID NLOAD and can be specified using dof's 1-3 (for translation about x, y, z respectively), or by dof's 4-6 (for rotation about x, y, z respectively). Note: This rotation takes the position of the grid point defined in CGID field as the center of rotation.

ANGVEL Angular velocity or angular position about local axis through center of rotation. If the value is an integer it represents the ID of a TABLED1, TABLED2 or TABL3D, i.e., a time-dependent or multi-dimensional table; however, no log scales, only linear scales. (Real or Integer; Default = 0.0)

DCOSi	Components of direction cosine of local axis if ANGVEL is nonzero. If the value is an integer, it represents the ID of a TABLED1, TABLED2 or TABL3D, i.e., a time-dependent or multi-dimensional table; however, no log scales, only linear scales. (Real or Integer; Default=0.0) In 2D contact only DCOS3 is used and the Default is 1.0.
VELRBi	Translational velocity or final position (depending on the value of CONTROL) of rigid body at the grid point defined in CGID field. For velocity control only, if the value is an integer, it represents the ID of TABLED1, TABLED2 or TABL3D, i.e., a time-dependent or multi-dimensional table; however, no log scales, only linear scales. Only VELRB1 and VELRB2 are used in 2D contact. (Real or Integer; Default = 0.0)
“APPROV”	The entries of this continuation line are for approaching velocity to establish initial contact.
A	Angular velocity about local axis through center of rotation. (Real, Default = 0.0)
Ni	Components of direction cosines of local axis of the angular velocity. The N1, N2, N3 define the axis through the point defined in the CGID field. Only N1 and N2 are used in 2D contact. (Real, Default = 0.0)
Vi	V1, V2 and V3 define the three components of the approaching velocity. Only V1 and V2 are used in 2D contact. (Real; Default = 0.0)
“GROW”	The entries of this continuation line are for rigid body growth. If tables are used for growth, they should either be TABLED1, TABLED2 (growth vs time) or TABL3D (growth vs one or more variables).
GFi	Components of growth factor of rigid body in the coordinate system of the CGID field. (Real, Default = 1.0)
TAB-GFi	Table IDs for growth factor of rigid body in the coordinate system of the CGID field. (Integer > 0 or blank, Default is blank)

Remarks:

1. The grid CGID is the reference grid for the rigid body motion. Loads and enforced motion must be defined in the global coordinate system of CGID. If CGID is not specified, basic coordinate will be used.
2. All continuation lines may be omitted if not required.

BCRGSRF**Rigid Contact Surface List in SOLs 101 and 400**

Defines a list of rigid contact surfaces used in SOLs 101 and 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
BCRGSRF	BSID	RBID1	RBID2	RBID3	RBID4	RBID5	RBID6	RBID7	
	RBID9	-etc.-							

Example:

BCRGSRF	1008	35	2						
---------	------	----	---	--	--	--	--	--	--

Descriptor Meaning

BSID Unique identification number referenced by a BCBODY1 entry. (Integer > 0) See Remark 1.

RBIDi Identification number of BCPATCH, BCBZIER, BCNURB2 or BCNURBS entry. (Integer > 0)

Remarks:

1. If BCRGSRF entry does not exist, BCBODY1 entry can refer to one of the identification number of BCPATCH, BCBZIER, BCNURB2 or BCNURBS entry directly.

BCSCAP

Defines a cap and number of segments

Defines whether a cap is added to a beam section and/or defines the number of segments used to model rod and tube sections for segment to segment beam contact in SOL 101 and 400 for general contact or in SOLs 101, 103, 105, 107 - 112, 200 and 400 for the permanently glued contact

Format:

1	2	3	4	5	6	7	8	9	10
BCSCAP	EID	IESCAP	NSEG						

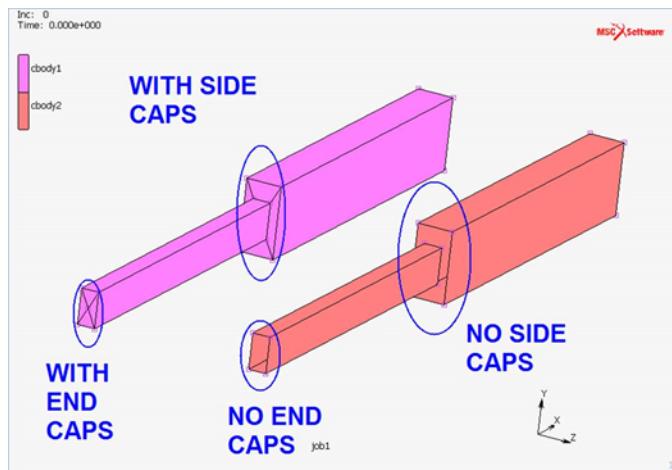
Example:

1	2	3	4	5	6	7	8	9	10
BCSCAP	2	1	20						

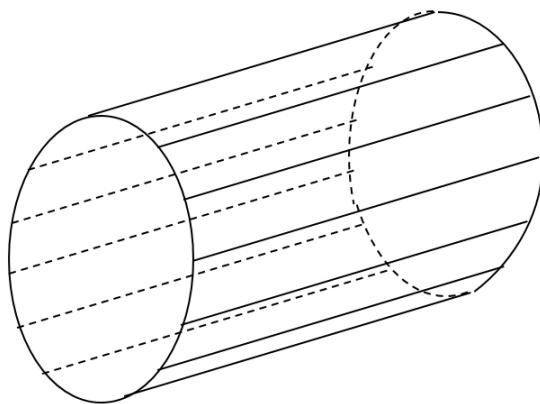
Descriptor	Meaning
EID	Beam element identification number. (Integer > 0; Required)
IESCAP	End and side cap flag (Integer ≥ 0 ; default=0). 0 no end or side cap 1 end cap only 2 side cap only 3 averaged side cap only 4 end cap and side cap 5 end cap and averaged side cap
NSEG	Number of segments for circular sections TYPE=ROD, TUBE or TUBE2 beam sections defined on PBARL or PBEAML. See remarks 2. (Integer ≥ 3 ; default=32)

Remarks:

1. If the card is not defined but the beam is included in segment to segment contact, all caps will be ignored. This card is only useful when a user needs to define the cap of beams for segment to segment contact.



2. The segments for circular cross sections will be generated as the figure below.



BCSEG

Contact Segment Defined Using Grids

Grids which are part of an element to be used in contact analyses in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
BCSEG	ID	IBODY	G1	G2	G3	G4			

Example:

1	2	3	4	5	6	7	8	9	10
BCSEG	100	1005	11	12	13	14			

Descriptor	Meaning
ID	Unique identification number for this BCSEG entry. (Integer > 0; Required)
IBODY	Identification number of a surface that is called out on the 5 th field of a BCBODY entry. (Integer > 0; Required)
G1,G2,G3,G4	GRID point identification numbers of an element on this surface. For quad plates and quad surfaces of solids, enter four grid id's. For triangular plates or triangular faces of solids, leave G4 blank. (Integer > 0; Required)

Remarks:

1. This entry is used as shown in the example below:

BCBODY, 201,,1005

BCSEG,1,1005,11,12,13,14

BCSEG,2,1005,21,22,23,24

BCSEG,3,1005,31,32,33,34

(In the above 11-14, 21-24 and 31-34 are GRID ID's)

BCSURF**Contact entry by element faces**

Define a contact region by element faces for SOLs 101, 103, 105, 107, 108, 109, 110, 111, 112, 200 and 400. This has three forms based the FORM (the 6th) field. “FACE” (default) is to define contact face by element face IDs, “GRID” is to define the same by grid point ids, and “RIGID” is to define a rigid face.

Format 1: FORM="FACE" (default)

1	2	3	4	5	6	7	8	9	10
BCSURF	BID	BPID	DIM		FORM	INCTHK	EDGCNT		
	ESID1	FACEID1	IDTYPE1		ESID2	FACEID2	IDTYPE2		
	ESID3	FACEID3	IDTYPE3		...				
	...								

Format 2: FORM="GRID"

1	2	3	4	5	6	7	8	9	10
BCSURF	BID	BPID	DIM		“GRID”	INCTHK	EDGCNT		
	EID1	G11	G12	G13	EID2	G21	G22	G23	
	EID3	G31	G32	G33	...				
	...								

Format 3: FORM="RIGID"

1	2	3	4	5	6	7	8	9	10
BCSURF	BID	BPID	DIM		“RIGID”				
	BSID	BCRGID							

Example:

FACE form (default)

BCSURF	2	3	3D			NO			
	12	S2			4	S4	SET		

GRID form

BCSURF	3	3	3D		GRID	YES			
	11	1	3	6	16	31	14	15	

RIGID form

BCSURF	4		3D		RIGID				
	101	201							

Descriptor	Meaning
BID	Unique contact face identification number referenced by BCNECT (Integer > 0; Required). See remark 1 .
BPID	Parameter identification number of a BCBDRP entry. (Integer > 0 or blank).
DIM	Dimension of body. (Character; Default= “3D”), “3D” or “2D”.
FORM	Select a type of the entry. (Character; Default =”FACE”). FACE define the contact surface by element face IDs GRID define the contact surface by grid points on an element face RIGID define the contact surface by rigid surface
INCTHK	Option to include shell thickness offset. (Character; Default =”YES”), “YES” or “NO”.
EDGCNT	Option for edge contact. (Integer, default=1), below three values can be specified. 1: only the beam/bar edges are included in the contact description (Default). 10: only the free and hard shell edges are included in the contact description. 11: both the beam/bar edges and the free and hard shell edges are included in the contact description.
ESIDI	Element id or set id. (Integer, no default). If IDTYPEi is “SET”, it is a set id defined by a SET3, and the SET3 must be element type. If IDTYPEi is “ELEM” (default), it is an element id.
FACEIDI	Element face ID. (Character, blank can be used to express BOTH for shell elements and the whole bodies of beam or bar elements). See remarks 6 , 7 , 8 . and 9 .
IDTYPEi	Type of ESIDI. (Character, “ELEM”). ELEM ESIDI is an element id. SET ESIDI is a SET3 id.
EIDI	Element id. (Integer > 0)
GIDIj	Three corner grid point ids of the element face. (Integer, no default). All the three fields of for an EIDI are blanks or zeros mean BOTH for shell elements and the whole bodies of bar or beam elements. See remark 2 .
BSID	Identification number of a BCRGSRF, BCPATCH, BCBZIER, BCNURB2, or BCNURBS. (Integer > 0).
BCRGID	Identification number of a BCRIGID entry. (Integer > 0).

Remarks:

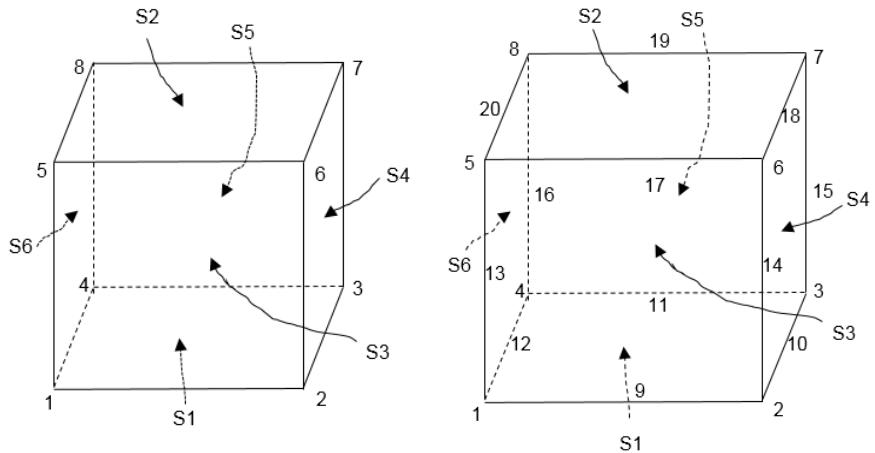
1. BID must be unique among all BCSURF, BCBODY1 and BCGRID entries.

2. BCSURF entries are able to coexist with BCBODY1 entries, however a BCSURF can only be used to construct a contact pair with another BCSURF, itself or a BCGRID in a BCONECT.
3. BCSURF cannot be specified in the continuations of a BCONECT.
4. If a BCSURF or BCGRID is referenced by a BCONECT, COPTS and COPTM in the BCONPRG referenced by the BCONECT are ignored.
5. For GRID form, three corner grid point IDs are required to specify a 3D element face, two corner grid point ids are required to specify a 2D element edge. For shell faces, the sequence of G1,G2,G3 of the element definition means TOP(top) and the reverse order means BTM(bottom), blanks or zeros means BOTH(both).
6. FACEIDi varies depending on element type and DIM. Refer to remark 9 for the list of FACEIDs.
7. For shell elements, TOP, BTM and BOTH are to define the contact face on top, bottom and both sides of the element. A blank is the same as BOTH. It is possible to have TOP and BTM in the same BCSURF entry but not allowed to combine BTM with BOTH or TOP with BOTH.
8. For beam elements, ENDA and ENDB express beam or bar enda and endb points respectively. A blank means the whole beam or bar body.
9. List of face ID.

Refer to following pages

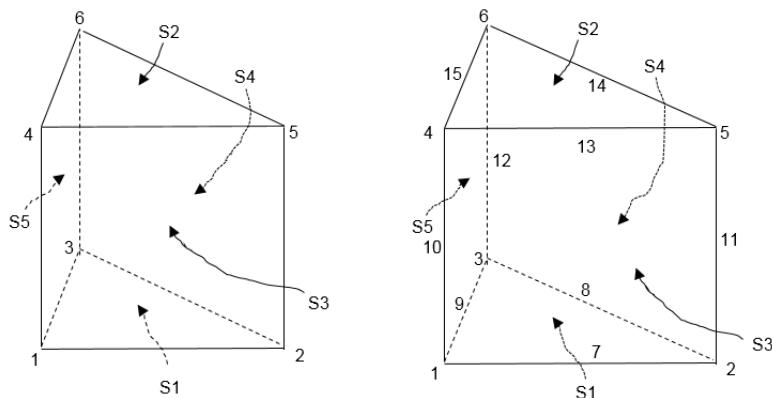
CHEXA

	Linear Element	Quadratic Element
S1	4-3-2-1	4-3-2-1-11-10-9-12
S2	5-6-7-8	5-6-7-8-17-18-19-20
S3	1-2-6-5	1-2-6-5-9-14-17-13
S4	2-3-7-6	2-3-7-6-10-15-18-14
S5	3-4-8-7	3-4-8-7-11-16-19-15
S6	4-1-5-8	4-1-5-8-12-13-20-16



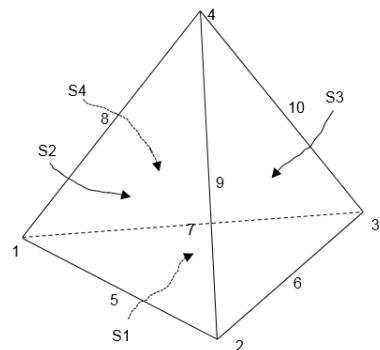
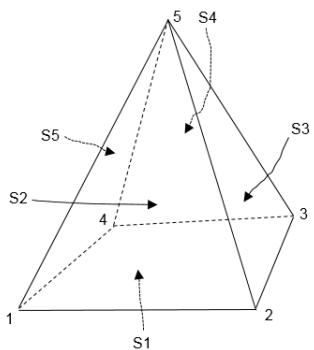
CPENTA

	Linear Element	Quadratic Element
S1	3-2-1	3-2-1-8-7-9
S2	4-5-6	4-5-6-13-14-15
S3	1-2-5-4	1-2-5-4-7-11-13-10
S4	2-3-6-5	2-3-6-5-8-12-14-11
S5	3-1-4-6	3-1-4-6-9-10-15-12



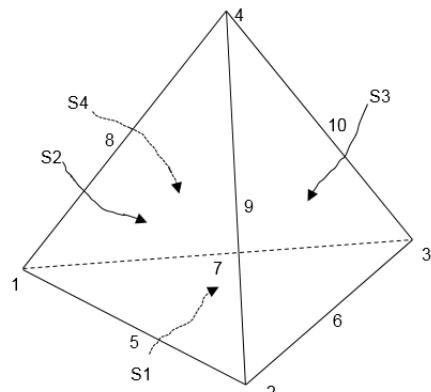
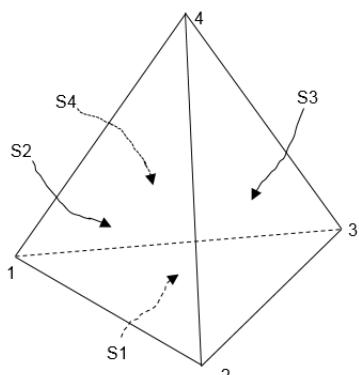
CPYRAM

	Linear Element	Quadratic Element
S1	4-3-2-1	4-3-2-1-8-7-6-9
S2	1-2-5	1-2-5-6-11-10
S3	2-3-5	2-3-5-7-12-11
S4	3-4-5	3-4-5-8-13-12
S5	4-1-5	4-1-5-9-10-13



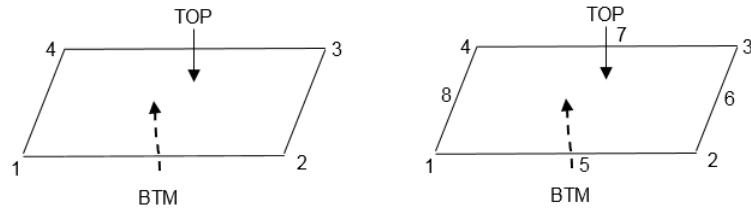
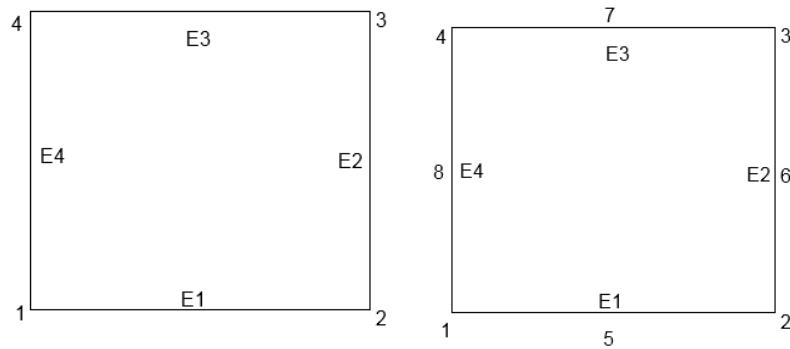
CTETRA

	Linear Element	Quadratic Element
S1	3-2-1	3-2-1-6-5-7
S2	1-2-4	1-2-4-5-9-8
S3	2-3-4	2-3-4-6-10-9
S4	3-1-4	3-1-4-7-8-10



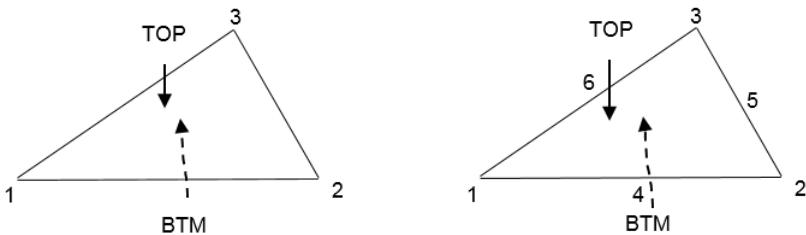
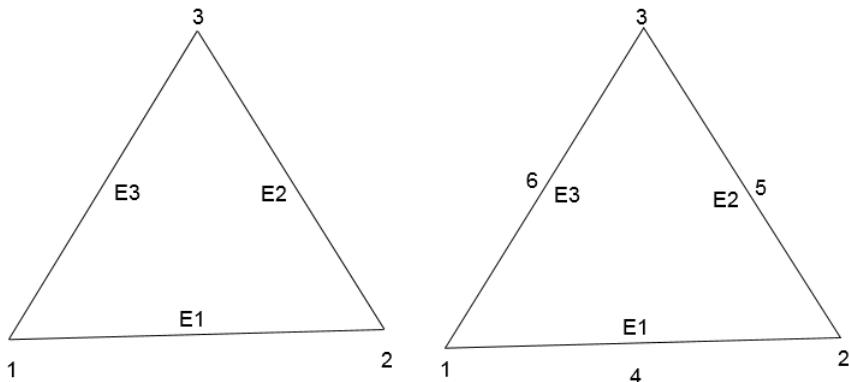
CQUAD4/CQUAD/CQUAD8/CQUADR when DIM is 3D

	Linear Element	Quadratic Element
E1	1- 2	1-2-5
E2	2-3	2-3-6
E3	3-4	3-4-7
E4	4-1	4-1-8
TOP	1-2-3-4	1-2-3-4-5-6-7-8
BTM	4-3-2-1	4-3-2-1-7-6-5-8



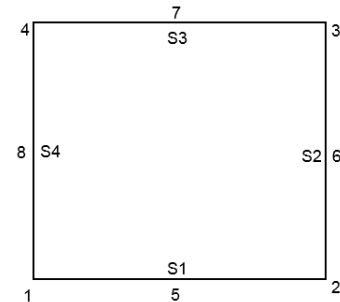
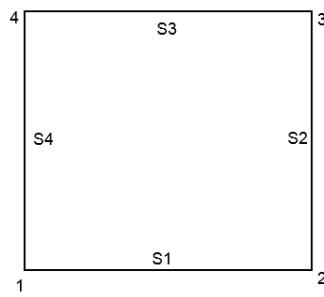
CTRIA3/ CTRIA6/CTRIAR when DIM is 3D

	Linear Element	Quadratic Element
E1	1- 2	1-2-4
E2	2-3	2-3-5
E3	3-1	3-1-6
TOP	-2-3	1-2-3-4-5-6
BTM	3-2-1	3-2-1-5-4-6



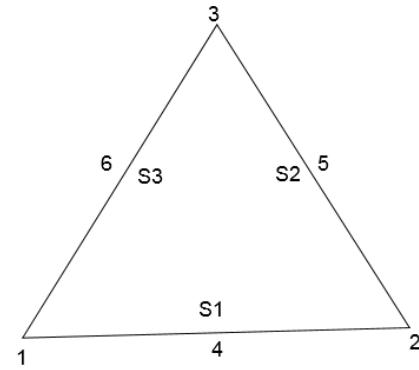
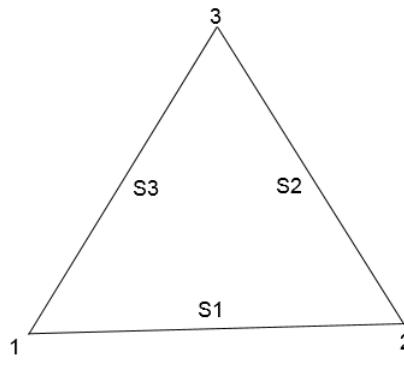
CQUAD4, CQUAD8 and CQUADX when DIM is 2D

	Linear Element	Quadratic Element
S1	1- 2	1-2-5
S2	2-3	2-3-6
S3	3-4	3-4-7
S4	4-1	4-1-8



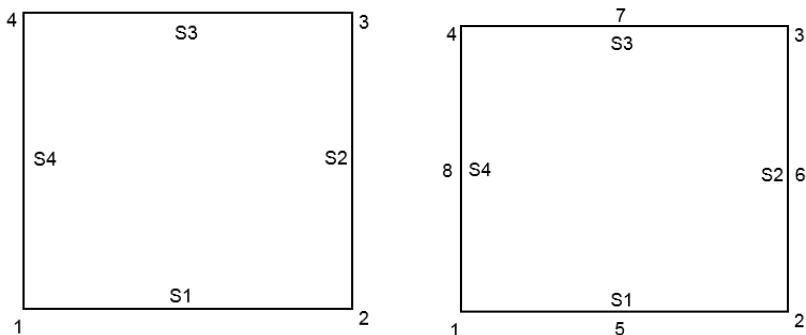
CTRIA3, CTRIA6, CTRIAX, CTRIA6X when DIM is 2D

	Linear Element	Quadratic Element
S1	1- 2	1-2-4
S2	2-3	2-3-5
S3	3-1	3-1-6



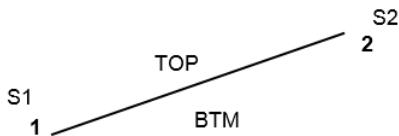
CTRIA3, CTRIA6, CTRIAX, CTRIA6 when DIM is 2D

	Linear Element	Quadratic Element
S1	1- 2	1-2-4
S2	2-3	2-3-5
S3	3-1	3-1-6



CAXISYM

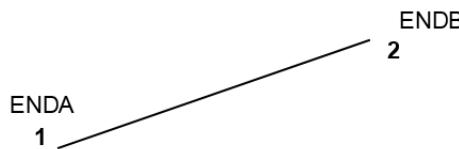
	Linear Element	Quadratic Element
S1	1	1
S2	2	2
TOP	1-2	1-2-3
BTM	2-1	2-1-3



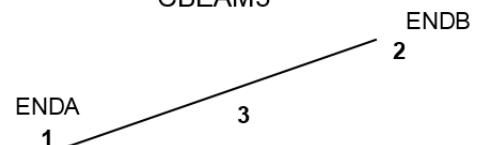
CBAR, CBEAM

	Linear Element	Quadratic Element
ENDA	1	1
ENDB	2	2
(BLANK)	1-2	1-2-3

CBEAM/CBAR



CBEAM3



BCTABLE

Defines a Contact Table for General Contact

Defines a contact table used in SOL 101, 400 and 700. SOL 200 (without calling SOL 400) can only support permanent glue. When SOL 200 calls SOL 400 (or say SOL 400 optimization), it can support all contact types. The BCTABL1 option is the preferred method to define a contact table.

Format: (SOLs 101 and 400 only)

1	2	3	4	5	6	7	8	9	10
BCTABLE	ID	IDSLAVE	IDMAST	NGROUP	COPTS	COPTM			
	“SLAVE”	IDSLA1	ERROR	FNTOL	FRIC	CINTERF	IGLUE		
		ISEARCH	ICOORD	JGLUE		DQNEAR			
		“FBSH”	FRLIM	BIAS	SLIDE	HARDS	COPTS1	COPTM1	
		“BKGL”	BGSN	BGST	BGM	BGN			
		“SEGS”	PENALT	AUGDIST	TPENALT	STKSLP			
		“HHHB”	HCT	HCV	HNC	BNC	EMISS	HBL	
			HNL	BNL	HGLUE				
	“MASTERS”	IDMA1	IDMA2	IDMA3	IDMA4	IDMA5	IDMA6	IDMA7	
		IDMA8	IDMA9	...					

Format: (SOL 700 only)

1	2	3	4	5	6	7	8	9	10
		SIDE	WEIGHT	MONDIS	MONDISV	INITMON	DAMPING		
	"MASTERS"	IDMA1	IDMA2	IDMA3	IDMA4	IDMA5	IDMA6	IDMA7	
		IDMA8	IDMA9	...					

Examples:

BCTABLE	2			3					
	SLAVE	10			0.2				
	MASTERS	20	30						
	SLAVE	20			0.3				
	MASTERS	10							
	SLAVE	30			0.2				
	MASTERS	10							

BCTABLE	0	1	2	0					
---------	---	---	---	---	--	--	--	--	--

Descriptor	Meaning
ID	Identification number referenced by a BCONTACT Case Control command. See Remark 5. (Integer; Required)
IDSLAVE	Identification number of a BCBODY entry defining the touching body. (Integer > 0 or blank)
IDMAST	Identification number of a BCBODY Bulk Data entry defining the touched body. (Integer > 0 or blank)
NGROUP	Flag to indicate that the continuation entries "SLAVE" and "MASTERS" are entered or not. Zero means no continuation entries are entered. Any positive integer means one or more sets of slave/master entries are entered. (Integer; Default = 1).
COPTS, COPTM, COPTS1, COMPTM1	Flag to indicate how slave and master surfaces may contact. See Remark 9. (Integer; Default = 0) COPTS and COPTM apply to all slave and master surfaces on this BCTABLE. To set individual slave/master combination differently, use COPTS1 and/or COPTM1. Ignored by SOL 700.
"SLAVE"	Indicates that this line defines the touching body and its parameters.

Descriptor	Meaning
IDSLA1	Identification number of a BCBODY Bulk Data entry defining the touching body. (Integer > 0) For SOL 700, leaving IDSLA1 blank will result in contact for all elements in the model. In this case, you are allowed to use ADAPT=YES.
ERROR	Distance below which a node is considered touching a body. Default = blank automatic calculation. See the Bulk Data entry BCPARA, 1342 for more details. Ignored by SOL 700. (Real)
FNTOL	Separation force, stress, or fraction above which a node separates from a body. FNTOL is closely related to IBSEP. Its default value is dependent on the IBSEP value. See remark 10 . Ignored by SOL 700. (Real)
FRIC	Friction coefficient. If the value is an integer, it represents the ID of a TABLEM1, TABLEM2 or TABL3D, i.e., a temperature-dependent or multidimensional table. SOL700 does not support the table definition. (Real > 0.0 or Integer > 0; Default = 0.0 for SOL400, Real > 0.0; Default=0.0 for SOL700)
CINTERF	Interference closure amount, normal to the contact surface. Default = 0. For CINTERF > 0, overlap between bodies. For CINTERF < 0., gap between bodies. Ignored by SOL 700. (Real)
IGLUE	Flag to activate glue option (Integer ≠ 0). Default is 0, no glue option. For SOL 700, IGLUE=1 is only acceptable. JGLUE option controls a glue contact type in details. Without assigning IGLUE and JGLUE options, general contact methods (METHOD=TIEDxxx) can assign glue contact types as well. (See Remarks 8 through 16 .)
±1	Activates the glue option. In the glue option, all degrees-of-freedom of the contact nodes are tied in case of deformable-deformable contact once the node comes in contact. The relative tangential motion of a contact node is zero in case of deformable-rigid contact. The node will be projected onto the contact body.
±2	Activates a special glue option to insure that there is no relative tangential and normal displacement when a node comes into contact. An existing initial gap or overlap between the node and the contacted body will not be removed, as the node will not be projected onto the contacted body. To maintain an initial gap, ERROR should be set to a value slightly larger than the physical gap.
±3	Insures full moment carrying glue when shells contact. The node will be projected onto the contacted body.

Descriptor	Meaning
	± 4 Insures full moment carrying glue when shells contact. The node will not be projected onto the contact body and an existing initial gap or overlap between the node and the contacted body will not be removed, as the node will not be projected onto the contacted body.
ISEARCH	Option for contact searching order, from Slave to Master or from Master to Slave, for deformable contact bodies. Ignored by SOL 700. ISEARCH is not necessary in segment-to-segment contact. (Integer; Default = 0) <ul style="list-style-type: none"> 0 (Double orders search) the search order is from lower BCBODOY ID's to higher ones first. Then it searches the opposite order. See Remark 19. 1 (Single order search) the searching order is from Slave to Master 2 (Single order search) let the program decide which search order. See Remark 19. <p>Note that ISTYP in BCBDPRP or BCBODY is supported with ISEARCH=0 only.</p>
ICOORD	Enter 1 to modify the coordinates of a node in contact with a deformable body so that stress-free initial contact can be obtained. Enter 2 to extend the tangential error tolerance at sharp corners of deformable bodies to delay sliding off a contacted segment. Enter 3 to have both 1 and 2 active. Ignored by SOL 700. (Integer; Default = 0)
JGLUE	This option is only relevant if the general glue option is invoked (IGLUE > 0 and =1). Enter 0 if a node should not separate (default). Enter 1 to invoke the standard separation behavior based on the maximum residual force. For SOLs 101 and 400, enter 2 to activate breaking glue. (Integer; Default = 0) <p>For SOL 700 the following options are available:</p> <ul style="list-style-type: none"> 0 Slave nodes in contact and which come into contact will permanently stick. Tangential motion is inhibited. This option is only available with METHOD=SS1WAY or SS2WAY and AUTO=YES.
DQNEAR	Distance below which near thermal contact behavior occurs. Used in heat transfer analysis only. Ignored by SOL 700. (Real; Default = 0; which means near contact does not occur)
"FBSH"	Enter character string FBSH if the line with items FRLIM, BIAS, SLIDE, HARDS is required to change the defaults of any of these values. (SOLs 101 and 400)
FRLIM	Friction stress limit. This entry is only used for friction type 6 (Coulomb friction using the bilinear model). If the shear stress due to friction reaches this limit value, then the applied friction force will be reduced so that the maximum friction stress is given by $\min(\mu\sigma_n, \sigma_{limit})$, with μ the friction coefficient and σ_n the contact normal stress. (Real; Default = 1.0E20)

Descriptor	Meaning
BIAS	Contact tolerance bias factor. If this field is left blank, the default is the BIAS of the BCPARA entry. A nonblank entry will override the BIAS entered on the BCPARA entry. Note 0.0 is not default, and will override the BIAS on BCPARA. SYSTEM(758)=1 will set 0.0 same as blank. (0.0<=Real<=1.0)
SLIDE	Delayed slide off distance. This entry should not be made unless ICOORD \geq 2 (see above). When using the delayed slide off option, a node sliding on a segment will slide off this segment only if it passes the node (2-D) or edge (3-D) at a sharp corner over a distance larger than the delayed slide off distance. By default, the delayed slide off distance is related to the dimensions of the contacted segment by a 20 percent increase of its isoparametric domain. (Real; Default = 0.0)
HARDS	Hard-soft ratio. This entry is only used if double-sided contact with automatic constraint optimization is used, (ISTYP=2 on the BCBODY entry). The hard-soft ratio can be used by the program if there is a significant difference in the (average) stiffness of the contact bodies (expressed by the trace of the initial stress-strain law). If the ratio of the stiffnesses is larger than the hard-soft ratio, the nodes of the softest body are the preferred slave nodes. (Real; Default = 2.0)
“BKGL”	Enter the character string BKGL if the line with items BGST, BGSN, BGM, BGN is required. (SOLs 101 and 400)
BGSN	Maximum normal stress for breaking glue. See Remark 10. (Real; Default = 0.0)
BGST	Maximum tangential stress for breaking glue. See Remark 10. (Real; Default = 0.0)
BGM	Exponent for the tangential stress term in deciding if a glue-contact will break. See Remark 10. (Real; Default = 2.0)
BGN	Exponent for the normal stress term in deciding if a glue-contact will break. See Remark 10. (Real; Default = 2.0)
“SEGS”	Enter the character string SEGS if the line with items PENALT and AUGDIST, TPENALT and STKSLP is required for the segment-to-segment contact.
PENALT	Augmented Lagrange penalty factor; used by the segment-to-segment contact algorithm only. (Real \geq 0.0; see Remark 16. for default)
AUGDIST	Penetration distance beyond which an augmentation will be applied; used by the segment-to-segment contact algorithm only. (Real \geq 0, see Remark 17. for default)
“HHHB”	Enter the character string HHHB if the line with items HCT, HCV, HNC, ..., up to HGLUE is required in a pure thermal analysis or in a coupled thermal/structural analysis. In a pure structural analysis they are ignored.

Descriptor	Meaning
HCT	Contact heat transfer coefficient. If Real, the value entered is the contact heat transfer coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the contact heat transfer coefficient vs temperature or a TABL3D entry specifying the contact heat transfer coefficient vs temperature and possibly other variables. (Real or Integer; Default = 0.0; In a thermal/mechanical coupled analysis a default value of 1.0E6 is used when the mechanical glue option is activated, i.w., when IGLUE > 0.)
HCV	Convection coefficient for near field behavior. If Real, the value entered is the near field convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field convection coefficient vs temperature or a TABL3D entry specifying the near field convection coefficient vs temperature and possibly other variables. (Real or Integer; Default = 0.0).
HNC	Natural convection coefficient for near field behavior. If Real, the value entered is the near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field natural convection coefficient vs temperature or a TABL3D entry specifying the near field natural convection coefficient vs temperature and possibly other variables. (Real or Integer; Default = 0.0).
BNC	Exponent associated with the natural convection coefficient for near field behavior. If Real, the value entered is the exponent associated with near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the exponent associated with the near field natural convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the near field natural convection coefficient vs temperature and possibly other variables. (Real or Integer; Default = 1.0.)
EMISS	Emissivity for radiation to the environment or near thermal radiation. If Real, the value entered is the emissivity. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the emissivity vs temperature or a TABL3D entry specifying the emissivity vs temperature and possibly other variables. (Real or Integer; Default = 0.0)
HBL	Separation distance dependent thermal convection coefficient. If Real, the value entered is the convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the convection coefficient vs temperature or a TABL3D entry specifying the convection coefficient vs temperature and possibly other variables. (Real or Integer; Default = 0.0).
HNL	Heat transfer coefficient for nonlinear convective heat flow for near field behavior. If Real, the value entered is the near field nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field nonlinear convection coefficient vs temperature or a TABL3D entry specifying the near field nonlinear convection coefficient vs temperature and possibly other variables. (Real or Integer; Default = 0.0)

Descriptor	Meaning
BNL	Exponent associated with the nonlinear convective heat flow for near field behavior. If Real, the value entered is the exponent associated with the near field nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 OR TABLEM2 entry specifying the exponent associated with the near field nonlinear convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the near field nonlinear convection coefficient vs temperature and possibly other variables. (Real or Integer; Default = 1.0)
HGLUE	Flag to activate the thermal glue option. When left blank or set to zero, thermal contact conditions will be treated by convective heat transfer between the bodies. When set to 1, the temperature fields of the bodies are tied as soon as they come in contact and there will be no convective heat transfer over the body interfaces. ($0 \leq$ Integer ≤ 1 ; Default = 0)
FK (SOL 700 only)	Kinetic coefficient of friction. (Real ≥ 0.0 ; Default = 0.0)
EXP (SOL 700 only)	Exponential decay coefficient. (Real ≥ 0.0 ; Default = 0.0)
METHOD (SOL 700 only)	Character, Influences the contact type used. See Remark 19. Options are: FULL: Regular Contact (Default) AIRBAG: Single Surface Contact SS1WAY: Surface To Surface One Way SS2WAY: Surface To Surface Two Way RB1WAY: Rigid Body One Way To Rigid Body RB2WAY: Rigid Body Two Way To Rigid Body RNRB: Rigid Nodes To Rigid Body TIEDNS: Tied Nodes to Surface TIEDNSO: Tied Nodes to Surface with Offset RELLIPS: Tied contact between grid points or rigid materials to ATB dummies. See Remark 23. BELT: Suited for modeling contact between a belt element and a rigid structure. Master slave contact only. The contact logic doesn't apply a contact force, but applies an enforced displacement and velocity that keeps the slave nodes exactly on top of the master face. The slave node does not slide relative to the master face when the friction coefficient (FS) is set to 1E20 BELT1: Identical to BELT algorithm, except that the slave nodes are initially repositioned on top of the closest master face. All slave nodes initially penetrated or within a distance of INITMON from a master face, are repositioned.

Descriptor	Meaning				
ADAPT (SOL 700 only)	Character, influences the contact type used. Options are NO or YES. Default = NO When ADAPT=YES, the BCBODY entries IDMAi must be defined as: behav=DEFORM bsid references a BCPROP				
THICK (SOL 700 only)	Shell thickness scale factor. (Real ≥ 0.0 ; Default = same as the value in DYPARAM,CONTACT,THICK)				
THICKOF (SOL 700 only)	Artificial contact thickness offset. (Real ≥ 0.0 ; Default = 0.0)				
PENV (SOL 700 only)	Overwrites the default maximum penetration distance factor. (Real ≥ 0.0 ; Default = 1.E20)				
FACT (SOL 700 only)	Scale factor for the contact forces. (Real ≥ 0.0 ; Default = 0.1)				
TSTART (SOL 700 only)	Time at which the contact is activated. (Real ≥ 0.0 ; Default = 0.0)				
TEND (SOL 700 only)	Time at which the contact is deactivated. (Real ≥ 0.0 ; Default = 1.e20)				
MAXPAR (SOL 700 only)	Maximum parametric coordinate in segment. (Real ≥ 0.0 ; Default = 1.025), search (values 1.025 and 1.20 recommended). Larger values can increase cost. If zero, the default is set to 1.025. This factor allows an increase in the size of the segments. May be useful at sharp corners.				
IADJ (SOL 700 only)	Adjacent material treatment option for solid elements. (Integer ≥ 0 ; Default = 1) <table> <tr> <td>0</td><td>Solid element faces are included only for free boundaries.</td></tr> <tr> <td>1</td><td>Solid element faces are included if they are on the boundary of the material subset. This option also allows the erosion within a body and the subsequent treatment of contact.</td></tr> </table>	0	Solid element faces are included only for free boundaries.	1	Solid element faces are included if they are on the boundary of the material subset. This option also allows the erosion within a body and the subsequent treatment of contact.
0	Solid element faces are included only for free boundaries.				
1	Solid element faces are included if they are on the boundary of the material subset. This option also allows the erosion within a body and the subsequent treatment of contact.				
IGNORE (SOL 700 only)	Ignore initial penetrations. (Integer ≥ 1) <table> <tr> <td>1</td><td>Allow initial penetrations to exist by tracking it</td></tr> <tr> <td>2</td><td>Move nodes to eliminate initial penetrations</td></tr> </table>	1	Allow initial penetrations to exist by tracking it	2	Move nodes to eliminate initial penetrations
1	Allow initial penetrations to exist by tracking it				
2	Move nodes to eliminate initial penetrations				
AUTO (SOL 700 only)	(Character) Options are: YES Automatic Contacts Activated (Default) NO Non-Automatic Contact Activated. This option is not recommended when Distributed Memory Parallel is activated.				

Descriptor	Meaning
SIDE (SOL 700 only)	Defines which side will be the monitoring side of a master face. The opposite side of the master face will be the penetration side. See Remark 24. (Characters; default=BOTH) BOTH: The side from which a slave node approaches the master face will become the monitoring side. TOP: The monitoring side will always be on the side of the master face that its normal is pointing at. BOTTOM: The monitoring side is always on the opposite side of the master face that its normal is pointing at.
WEIGHT (SOL 700 only)	The contact force is multiplied by a mass-weighting factor. The following options are available. See Remark 25. (Character; default=BOTH) BOTH: $M_{scale} = \frac{M_{slave} M_{master}}{M_{slave} + M_{master}}$ SLAVE: $M_{scale} = M_{slave}$ MASTER: $M_{scale} = M_{master}$ NONE: $M_{scale} = 1.0$
MONDIS (SOL 700 only)	Defines the fixed part of the monitoring distance. When the normal distance of a slave node to a master face becomes smaller than the monitoring distance the slave node will tag itself to the master face. The side from which the slave node is moving towards the master face becomes the monitoring region. (Character; default=FACTOR) FACTOR: The monitoring distance is equal to a factor times a characteristic length of the master faces. The factor is specified in MONDISV.
DISTANCE	The monitoring distance is specified in MONDISV.
:	
MONDISV (SOL 700 only)	Value of the monitoring distance or value of the FACTOR to calculate the monitoring distance. (Real; default=2.0)
INITMON (SOL 700 only)	Fixed part of the monitoring distance used during the initialization. If not specified, the value of MONDIS is used. (Real > 0.0; default=MONDIS)
DAMPING (SOL 700 only)	Specifies if a high frequency damping is active or not. The damping force is based on the relative velocity of a slave node with respect to a master face. The damping is preferably turned on in all cases, except for RIGID-RIGID contact. In RIGID-RIGID contact it can result in a substantial loss of energy. VERSION V4 only. (Character; Default=YES) YES damping is activated N damping is not activated

Descriptor	Meaning
"MASTERS"	Indicates the start of the list of bodies touched by touching body IDSLA1.
IDMAi	Identification numbers of BCBODY Bulk Data entries defining touched bodies. (Integer > 0)

Remarks:

1. BCTABLE defines surface contact.
2. If BCTABLE is not given, the default for contact analysis is assumed if BCONTACT=ALLBODY in Case Control Section, and, every body detects the possibility of contact relative to all other bodies and itself if it is a deformable body. If BCTABLE is given, the default for every body is overwritten. The touching body does not contact itself unless requested. When the touched body is deformable, double-sided contact is applied by default. BCTABLE is useful for deactivating or activating bodies to reduce computational effort and to change contact conditions between subcases.
3. A short input to define two contact bodies exists if the user provides IDSLAVE and IDMAST. Then it is assumed that there are only two contact bodies, NGROUP is ignored and continuation entries are not allowed. Default values are set for the parameters on the continuation entry.
4. If the user leaves IDSLAVE and IDMAST blank, then NGROUP is normally required and continuation entries are usually expected for NGROUP SLAVE/MASTER combinations. Exceptions are for SOL 700 self-contact may be designated using a slave IDSLA1 of zero and no MASTER entry.
5. For SOLs 101 and 400, the BCTABLE with ID=0 will be used in loadcase 0 automatically that does not need a corresponding Case Control command BCONTACT=0. The loadcase 0 is purely elastic and it can also be used to (1) move rigid contact bodies so that they just touch flexible contact bodies, and/or (2) remove any prestressed condition by adjusting the coordinates of the active nodes, which are the Grid Points on the surface of all deformable BCBODY's. BCTABLE, 0 is not used to define the contact relationship for any loadcases. ONLY BCTABLE, n(n>0), will dominate the contact analysis for each loadcase. To place an entry in any physical loadcase (SUBCASE or STEP), the ID must be selected by the Case Control command BCONTACT=ID. When BCONTACT=ALLBODY, there is no ID of BCTABLE specified; therefore, the default values of all entries of BCTABLE are assumed.
6. The line starting with "HHHB" is used for heat transfer or thermal contact analyses only. Also, see Remark 17.
7. It is not necessary to enter all continuation lines between SLAVE and MASTERS. For example, if an entry is required on the 4th SLAVE line, the first 3 must be entered with some being blank and those after the 4th may be omitted. All continuation lines prior to the last needed must be entered.
8. Many slave/masters may be entered up to the number specified in Remark 5. A new slave entry may not begin until the masters from the previous entry are finished (as shown in the example). The number of master surfaces for any given slave surface is limited by the number specified in Remark 5., however most GUI's produce one slave surface and one master surface per slave/masters pairs.

9. COPTS, COPTM, COPTS1 and COPTM1 are packed numbers designating how the surfaces may contact using the formula

$$\text{COPxxx} = A + 10^*B + 1000^*C$$

where the following codes apply:

A: the outside of the solid elements in the body

- = 1: the outside will be in the contact description (Default)

B (flexible bodies): the outside of the shell elements in the body

- = 1: both top and bottom faces will be in the contact description, thickness offset will be included (Default)
- = 2: only bottom faces will be in the contact description, thickness offset will be included
- = 3: only bottom faces will be in the contact description, shell thickness will be ignored
- = 4: only top faces will be in the contact description, thickness offset will be included
- = 5: only top faces will be in the contact description, shell thickness will be ignored
- = 6: both top and bottom faces will be in the contact description, shell thickness will be ignored

Note: The choice B = 6 for both bodies in a contact combination is only meaningful for glued contact. If in such cases separation is allowed, separated nodes will not come into contact anymore, unless a new CONTACT TABLE is defined to reset the value of B.

B (rigid bodies): the rigid surface

- = 1: the rigid surface should be in the contact description (Default)

C (flexible bodies): the edges of the body

- = 1: only the beam/bar edges are included in the contact description (Default)
- = 10: only the free and hard shell edges are included in the contact description
- = 11: both the beam/bar edges and the free and hard shell edges are included in the contact description

Note that C has no effect if beam-to-beam contact is **not** switched on (BEAMB ≠ 1 on BCPARA).

Note that C has no effect if segment-to-segment contact is used.

10. Breaking Glue provides glued contact to all GRID's at their very 1st contact. This kind of glued-contact will break if

$$(\sigma_n/BGSN)^{**}bgn + (\sigma_t/BGST)^{**}bgm > 1.0$$

When a contact node breaks due to the above criterion, standard contact is activated if it comes into contact again. If BGSN = 0.0 the first term is ignored. If BGST is zero, the second term is ignored.

11. For hard contact, with HGLUE=1:

- a. The temperature of the contacting grid is tied to the temperatures of the contacted element face or the temperature of the rigid geometry when it has a scalar point associated with it.

- b. The temperature of the contacting grid is set to the rigid geometry temperature when it has no scalar point associated with it.

Note:

“Glued” thermal contact can result in overshoot of the temperatures at the interface, in particular, if two bodies that have non-uniform initial temperatures are placed in contact. The overshoot effect may be damped somewhat if one uses a near contact distance with some convective heat transfer.

12. For hard contact, with HGLUE=0:

The convective heat flow per unit area over the two interfaces is given by:

$$q = HCT \cdot (T_A - T_B)$$

where T_A is the contacting grid temperature and T_B is the face temperature in the contact point in case of a meshed body or the T_{BODY} temperature in case of a rigid geometry.

13. For near contact:

$$\begin{aligned} q = & HCV \cdot (T_A - T_B) + \\ & HNC \cdot (T_A - T_B)^{BNC} + \\ & HNL \cdot (T_A^{BBL} - T_B^{BBL}) + \\ & \sigma \cdot EMISS \cdot (T_A^A - T_B^A) + \\ & \left[HCT \cdot \left(1 - \frac{dist}{DQNEAR} \right) + HBL \cdot \frac{dist}{DQNEAR} \right] \cdot (T_A \cdot T_B) \end{aligned}$$

where the last term is only activated when $HBL \neq 0$, T_A is the contacting grid temperature and T_B is the face temperature in the contact point in case of a meshed body or the T_{BODY} temperature in case of a rigid geometry.

14. The heat transfer coefficients and associated exponents can all be temperature dependent, when they are entered as an integer value. This integer value is the table ID of a TABLEM1, TABLEM2 or TABL3D entry (formulas are not supported on TABL3D).
15. Table IDs of tables used on the BCTABLE and the BCBODY entry must be unique.
16. The penalty factor used in the augmented Lagrange method is by default derived from the contact characteristic distance and the stiffness of the deformable contact bodies involved (note that the dimension of the penalty factor is force per cubic length).

$$PENALT = \frac{0.5(S_i + S_j)}{\bar{L}}$$

The body stiffness (S_i and S_j), are either defined by the average trace of the initial stress-strain law of the elements defining the two contact bodies or by the average bulk modulus for (nearly) incompressible rubber materials, whichever of the two is the largest.

For continuum elements, the characteristic length (\bar{L}) is given by one half of the average length of all the edges being part of the contact boundary. For shell elements, the characteristic length is given by half of the average thickness of all the shell elements being part of a contact body. When there is contact between a solid and a shell element, then the characteristic length is defined by the shell element.

In case of contact with a rigid body, since there is no body stiffness associated with a rigid body, the default value is related to the deformation body only and given by:

$$\text{PENALT} = \frac{1000S_i}{\bar{L}}$$

17. By default, the threshold value of this penetration distance is 0.05 times the default contact characteristic distance.

$$\text{AUGDIST} = 0.05\bar{L}$$

18. The multipoint constraint equations (MPC equations) internally created from body contact can be printed out in standard Nastran punch file by using Case Control command, NLOPRM MPCPCH. This capability is good for all solutions except SOL 700.
19. When ISEARCH=0 (and ISTYP=0 in default on BCBODY), the search order is from lower BCBODOY ID to higher one to create the first set of contact constraints and then add the constraints in the search order from higher BCBODY ID to lower one as long as they are not in conflict with the first set.

When ISEARCH=2, the program looks into the smallest element edge at the outer boundary (and the smallest thickness in case of shell elements) of each BCBODY. Then, the search order of the two deformable contact bodies is determined by the following rule when ID1 < ID2

CL1= Min(1/20 of the smallest edge,1/4 of the smallest thickness) of BCBODY ID1

CL2= Min(1/20 of the smallest edge,1/4 of the smallest thickness) of BCBODY ID2

CL1 and CL2 refers to characteristic length of BCBODY ID1 and ID2.

The search order is from lower BCBODY ID1 to higher BCBODY ID2 if CB1<=1.05*CB2.

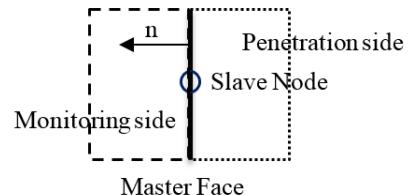
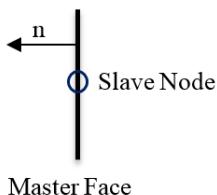
Otherwise, if CB1>1.05*CB2 the search order is from BCBODY ID2 to BCBODY ID1.

20. Fields ISEARCH, DQNEAR and "BKGL" are not supported in segment to segment contact analysis.
21. If STKSLP is set to 0.0 (Default), the sticking stiffness K1 is equal to the maximum friction force ($\mu \cdot F_{NORMAL}$, where μ is the friction coefficient) divided by the maximum sticking displacement. Otherwise, K1 is equal to the maximum friction force divided by the value of STKSLP.
22. When the initial stress-free(ICORD=1), Node-to-Seg updates model geometry to close the gap, nodes are projected onto the contact body; but Seg-to-Seg treats initial stress-free as pre-stress in the equation without geometry update, no node projection is considered.
23. When METHOD=RELLIPS is used, BCGRID or BCMATL are only available for SLAVE body and BCELIPS is only available for MASTER body. When BCMATL is used, the MATRIG id or RBE2 id are only acceptable.

24. The options TOP/BOTTOM are useful in the following cases:

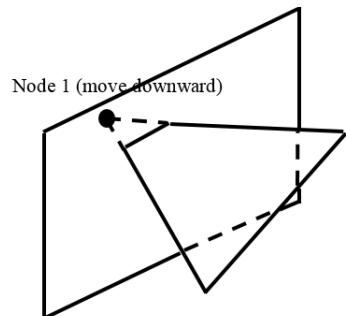
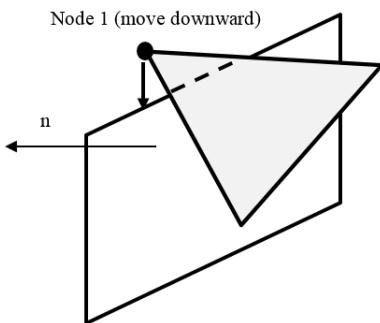
- a. When a slave node initially is located on the master face (see the picture below), the contact situation is uniquely defined, only if the TOP or BOTTOM side of the master surface is defined:

SIDE=TOP case:



- b. When hooking of slave nodes on the wrong side of a master face might occur. This often is the case when the master face is at the edge of a shell element structure:

SIDE=BOTTOM case: penetration of node 1 (SIDE=BOTH: no penetration of node 1)



25. Recommended usage of WEIGHT:

SLAVE	MASTER	WEIGHT
Deformable	Deformable	BOTH
Deformable	Rigid	SLAVE
Rigid	Deformable	MASTER
Rigid	Rigid	NONE

Default setting when BCELIPS is used in SLAVE or MASTER:

SLAVE	MASTER	WEIGHT
Non-BCELIPS	BCELIPS	SLAVE
BCELIPS	Non-BCELIPS	NONE
	BCELIPS	

BCTABLE - Glued Option

Defines a Contact Table for Permanently Glued Contact

Defines a glued contact used in SOLs 101, 103, 105, 107, 108, 109, 110, 111, 112, 200 and SOL 400 for the permanently-glued or general contact.

Format: (SOLs 101, 103, 105, 107, 108, 109, 110, 111, 112, 200 and 400 only)

1	2	3	4	5	6	7	8	9	10
BCTABLE	ID			NGROUP	COPTS	COPTM			
	“SLAVE”	IDSLA1	ERROR			CINTER F	IGLUE		
		ISEARCH	ICOORD						
		“FBSH”		BIAS	SLIDE		COPTS1	COPTM1	
	“MASTERS”	IDMA1	IDMA2	IDMA3	IDMA4	IDMA5	IDMA6	IDMA7	
		IDMA8	IDMA9	...					

Examples:

BCTABLE	101			2	61	61			
	SLAVE	20	0.002		0.2		1		
		1							
		FBSH		0.01			60	60	
	MASTERS	31	32	33					
	SLAVE	40					1		
	MASTERS	51	52	53	54				

Descriptor	Meaning
ID	Identification number of a BCTABLE entry to be selected by BCONTACT Case Control command. See Remark 5. for the general contact. Case Control, BCONTACT=ALLBODY, cannot be used for the permanently glued contact. See Remark 1. (Integer ≥ 0 ; Required)
NGROUP	Flag to indicate the number of pairs of “SLAVE” and “MASTERS” entries in the continuation lines. (Integer > 0 ; Required)
COPTS1, COPTM	Flag to indicate how slave and master surfaces may contact. See Remark 6. for the general contact.
“SLAVE”	Indicates that this line defines the slave or touching body and its parameters.
IDSLA1	Identification number of a BCBODY Bulk Data entry defining the touching body. (Integer > 0 ; Required)

Descriptor	Meaning
ERROR	Distance below which a node is considered touching a body. When this field is left blank or 0.0, the value is taken from BCPARA = 0, ERROR, if it is specified. Otherwise, Nastran automatically calculates the value. See the Bulk Data entry BCPARA, 1342 for more details. (Real)
CINTERF	Interference closure amount, normal to the contact surface. For CINTERF > 0.0, overlap between bodies, CINTERF < 0.0, gap between bodies. (Real; Default = 0.0)
IGLUE	Flag to activate the glue options. See the descriptions of various options in BCTABLE for the general contact. This field must be specified for the glued contact. See Remark 4. (Integer > 0; Required). <ol style="list-style-type: none"> 1. Activates the glue option. In the glue option, all degrees-of-freedom of the contact nodes are tied in case of deformable-deformable contact once the node comes in contact. The relative tangential motion of a contact node is zero in case of deformable-rigid contact. This option is recommended when there is no gap or overlap between contact surfaces or initial stress free contact is specified. 2. Activates a special glue option to insure that there is no relative tangential and normal displacement when a node comes into contact. An existing initial gap or overlap between the node and the contacted body will not be removed, as the node will not be projected onto the contacted body. To maintain an initial gap, ERROR should be set to a value slightly larger than the physical gap. 3. Insures full moment carrying glue when shells contact. This option is recommended when moments are important and there is no gap or overlap between contact surfaces or initial stress free contact is specified. 4. Insures full moment carrying glue when shells contact. The node will not be projected onto the contact body and an existing initial gap or overlap between the node and the contacted body will not be removed, as the node will not be projected onto the contacted body.
ISEARCH	Option for contact searching order, from a slave (touching) to a master (touched) body or visa versa, for deformable contact. See the options in BCTABLE for the general contact. (Integer; Default = 0) <ul style="list-style-type: none"> 0 (Double orders search) 0(Double orders search) the search order is from lower BCBODOY ID's to higher ones first. Then it searches the opposite order. See Remark 19. in BCTABLE. 1 (Single order search) the searching order is from Slave to Master 2 (Single order search) let the program decide which search order. See Remark 19. in BCTABLE.

Descriptor	Meaning
ICOORD	Enter 1 to modify the coordinates of a node in contact with a deformable body so that stress-free initial contact can be obtained. Enter 2 to extend the tangential error tolerance at sharp corners of deformable bodies to delay sliding off a contacted segment. Enter 3 to have both 1 and 2 active. (Integer; Default = 0)
"FBSH"	Character string used to introduce BIAS, COPTS1, and COPTM1.
BIAS	Contact tolerance bias factor. If this field is left blank, the default is the BIAS of the BCPARA entry. A nonblank entry will override the BIAS entered on the BCPARA entry. Note 0.0 is not default, and will override the BIAS on BCPARA. SYSTEM(758)=1 will set 0.0 same as blank. (0.0<=Real<=1.0)
SLIDE	Delayed slide off distance. This entry should not be made unless ICOORD \geq 2 (see above). When using the delayed slide off option, a node sliding on a segment will slide off this segment only if it passes the node (2-D) or edge (3- D) at a sharp corner over a distance larger than the delayed slide off distance. By default, the delayed slide off distance is related to the dimensions of the contacted segment by a 20 percent increase of its isoparametric domain. (Real; Default = 0.0)
COPTS1, COPTM1	Flag to indicate how individual pair of slave and master contact surfaces may contact. These two fields override COPTS and COPTM. See Remark 6.
"MASTERS"	Indicates the start of the list of bodies touched by touching body IDSLA1.
IDMAi	Identification numbers of BCBODY Bulk Data entries defining touched bodies. (Integer > 0; Required)

Remarks:

1. This BCTABLE entry is tailored specifically for what we called the permanently-glued contact (or tied contact), from the entry of BCTABLE for the general contact, where the glued is an option in the approaching-and-touched contact. Most of the descriptions and remarks on the fields in this entry can be found in the BCTABLE for the general contact.
2. Permanent glued contact for small rotation condition is defined when all IGLUE fields of BCTABLE, reference by the very first loadcase, (the 1st STEP of the 1st SUBCASE in SOL 400 or the 1st SUBCASE in others) is set to larger than 0, IGLUE(>0). In BCPARA with ID=0, the value of NLGLUE may not be 1.

For large deformation and large rotation condition, permanent glued contact is activated by negative IGLUE (<0) and works in corresponding subcase or step in the corresponding subcase or step where it is defined, no matter what is set (1 or 0). This type of permanent glued contact with large deformation and large rotation may be applied together with combination of general glued contact and touching contact. IGLUE will be changed to negative automatically if SYSTEM(758)=2 when Permanently Glued setting is found with large rotation/deformation effect turned on.

3. The MPC equations internally created for the glued contact can be printed out in Nastran standard punch file by using Case Control command, NLOPRM MPCPCH=BEGN.

4. IGLUE=±2 or ±4 is favorable to passing the GROUNDCHECK. The initial stress-free contact is also available to preserve the six rigid-body modes with IGLUE = 1 or 3.
5. For SOLs 101 and 400, the BCTABLE with ID=0 will be used in loadcase automatically that does not need a corresponding Case Control command BCONTACT=0. The loadcase 0 is purely elastic and it can also be used to (1) move rigid contact bodies so that they just touch flexible contact bodies, and/or (2) remove any prestressed condition by adjusting the coordinates of the active nodes, which are the Grid Points on the surface of all deformable BCBODY's. To place an entry in any physical loadcase (SUBCASE or STEP), the ID must be selected by the Case Control command BCONTACT=ID. When BCONTACT=ALLBODY, there is no ID of BCTABLE specified; therefore, the default values of all entries of BCTABLE are assumed.
6. COPTS, COPTM COPTS1 and COPTM1 are packed numbers designating how the surfaces may contact using the formula

$$\text{COPxxx} = A + 10^*B + 1000^*C$$

where the following codes apply:

A: the outside of the solid elements in the body

- = 1: the outside will be in the contact description (Default)

B (flexible bodies): the outside of the shell elements in the body

- = 1: both top and bottom faces will be in the contact description, thickness offset will be included (Default)
- = 2: only bottom faces will be in the contact description, thickness offset will be included
- = 3: only bottom faces will be in the contact description, shell thickness will be ignored
- = 4: only top faces will be in the contact description, thickness offset will be included
- = 5: only top faces will be in the contact description, shell thickness will be ignored
- = 6: both top and bottom faces will be in the contact description, shell thickness will be ignored

Note: The choice B = 6 for both bodies in a contact combination is only meaningful for glued contact. If in such cases separation is allowed, separated nodes will not come into contact anymore, unless a new CONTACT TABLE is defined to reset the value of B.

B (rigid bodies): the rigid surface

- = 1: the rigid surface should be in the contact description (Default)

C (flexible bodies): the edges of the body

- = 1: only the beam/bar edges are included in the contact description (Default)
- = 10: only the free and hard shell edges are included in the contact description
- = 11: both the beam/bar edges and the free and hard shell edges are included in the contact description

Note that C has no effect if beam-to-beam contact is **not** switched on (BEAMB ≠ 1 on BCPARA)

7. It is not supported that a permanently-glued contact is used to glue a deformable body to a rigid one. If it is a SOL 101 or SOL 400 job, the general contact with glued option must be performed.

BCTABLE

Defines a Contact Table in SOL 600

Defines a contact table in SOL 600.

For heat transfer or thermal contact do not enter parameter PARAM,MRCONVER,11 in the bulk data. For structural analysis, enter PARAM,MRCONVER,11 if SLAVE lines "FBSH" or beyond are used.

Format:

1	2	3	4	5	6	7	8	9	10
BCTABLE	ID	IDSLAVE	IDMAST	NGROUP					
	"SLAVE"	IDSLA1	ERROR	FNTOL	FRIC	CINTERF	IGLUE		
		ISEARCH	ICOORD	JGLUE	TOLID	DQNEAR	DISTID		
		"FBSH"	FRLIM	BIAS	SLIDE	HARDS	COPTS1	COPTM1	
		"BKGL"	BGSN	BGST	BGM	BGN			
		"HHHB"	HCT	HCV	HNC	BNC	EMISS	HBL	
		"SEGS"	PENALT	AUGDIST					
	"MASTERS"	IDMA1	IDMA2	IDMA3	IDMA4	IDMA5	IDMA6	IDMA7	
		IDMA8	IDMA9	...					

Examples:

BCTABLE	2			3					
	SLAVE	10			0.2				
	MASTERS	20	30						
	SLAVE	20			0.3				
	MASTERS	10							
	SLAVE	30			0.2				
	MASTERS	10							

BCTABLE	0	1	2	0					
---------	---	---	---	---	--	--	--	--	--

Descriptor	Meaning
ID	Identification number referenced by a BCONTACT Case Control command. See Remark 6. (Integer; Required)
IDSLAVE (3,1) [3,1]	Identification number of a BCBODY entry defining the touching body. (Integer > 0 or blank)
IDMAST (5,1) [15,1]	Identification number of a BCBODY Bulk Data entry defining the touched body. (Integer > 0 or blank)

Descriptor	Meaning
NGROUP (2,1) [2,1]	Flag to indicate that the continuation entries “SLAVE” and “MASTERS” are entered or not. Zero means no continuation entries are entered. Any positive integer means one or more sets of slave/master entries are entered. (Integer; Default = 1).
COPTS1, COMPTM1	Flag to indicate how slave and master surfaces may contact. See Remark 10. (Integer; Default = 0).
“SLAVE”	Indicates that this line defines the touching body and its parameters.
IDSLA1 (3,1) [3,1]	Identification number of a BCBODY Bulk Data entry defining the touching body. (Integer > 0)
ERROR (3,2) [3,2]	Distance below which a node is considered touching a body. (Real for BCTABLE with ID = 0, Real or Integer for BCTABLE with ID > 0). If the value is an Integer, it is the ID of a TABLED1 or TABL3Di providing error vs time or other variables. Default = blank automatic calculation of ERROR without any table dependence. See the Bulk Data entry BCPARA, 1349 for more details.
FNTOL (3,3) [5,1]	Separation force or stress (depending BCPARA IBSEP) on above which a node separates from a body. Default is maximum residual force. (Real)
FRIC (3,4) [5,2]	Friction coefficient. If the value is an integer, it is the ID of a TABLED1 or TABL3Di providing friction vs time or friction vs other variables. (Real > 0.0 or Integer; Default = 0.0)
CINTERF (3,5) [5,3]	Interference closure amount, normal to the contact surface. CINTERF > 0.0, designates an overlap between bodies, CINTERF < 0.0 designates a gap between bodies. If the value is an integer, it is the ID of a TABLED1 or TABL3Di providing interference closure vs time or friction vs other variables. (Real or Integer; Default = 0.0)

Descriptor	Meaning
IGLUE (3,7) [3,7]	<p>Flag to activate glue option (Integer ≥ 0). Default is 0, no glue option,</p> <ul style="list-style-type: none"> 1. Activates the glue option. In the glue option, all degrees-of-freedom of the contact nodes are tied in case of deformable-deformable contact once the node comes in contact. The relative tangential motion of a contact node is zero in case of deformable-rigid contact. The node will be projected onto the contact body. 2. Activates a special glue option to insure that there is no relative tangential and normal displacement when a node comes into contact. An existing initial gap or overlap between the node and the contacted body will not be removed, as the node will not be projected onto the contacted body. To maintain an initial gap, ERROR should be set to a value slightly larger than the physical gap. 3. Insures full moment carrying glue when shells contact. The node will be projected onto the contacted body. 4. Insures full moment carrying glue when shells contact. The node will not be projected onto the contact body and an existing initial gap or overlap between the node and the contacted body will not be removed, as the node will not be projected onto the contacted body.
ISEARCH (3,8) [3,8]	<p>Option for contact searching order, from Slave to Master or from Master to Slave, for deformable contact bodies. ISEARCH is not supported in segment-to-segment contact. (Integer; Default = 0)</p> <ul style="list-style-type: none"> 0 Double orders search) the search order is from lower BCBODDY ID's to higher ones first. Then it searches the opposite order. See Remark 19. in BCTABLE. 1 (Single order search) the searching order is from Slave to Master 2 (Single order search) let the program decide which search order. See Remark 19. in BCTABLE.
ICOORD (3,9) [3,9]	Enter 1 to modify the coordinates of a node in contact with a deformable body so that stress-free initial contact can be obtained. Enter 2 to extend the tangential error tolerance at sharp corners of deformable bodies to delay sliding off a contacted segment. Enter 3 to have both 1 and 2 active. (Integer; Default = 0)
JGLUE (3,10) [3,10]	This option is only relevant if the glue option is invoked (IGLUE > 0). Enter 0 if a node should not separate (Default). Enter 1 to invoke the standard separation behavior based on the maximum residual force. (Integer; Default = 0)
TOLID (4,2) [4,2]	Enter 2 to activate breaking glue.
	Contact tolerance table ID of a TABLED <i>i</i> . Used in heat transfer analysis only. (Integer; Default = 0 which means no table ID)

Descriptor	Meaning
DQNEAR (3,3) [3,3]	Distance below which near thermal contact behavior occurs. Used in heat transfer analysis only. Used in heat transfer analysis only. If the value is an integer, it is the ID of a TABLED1 or TABL3Di providing friction vs time or friction vs other variables. (Real; or integer; Default = 0.0; which means near contact does not occur)
DISTID (4,3) [4,3]	Contact near distance table ID of a TABLED1. Used in heat transfer only. (Integer; Default = 0 which means near contact does not occur) (Integer; Default = 0 which means no table ID)
"FBSH"	Enter character string FBSH if the line with items FRLIM, BAIS, SLIDE, HARDS is required to change the defaults of any of these values.
FRLIM [5,4]	Friction stress limit. This entry is only used for friction type 6 (Coulomb friction using the bilinear model). If the shear stress due to friction reaches this limit value, then the applied friction force will be reduced so that the maximum friction stress is given by $\min(\mu\sigma_n, \sigma_{limit})$, with μ the friction coefficient and σ_n the contact normal stress. If the value is an integer, it is the ID of a TABLED1 or TABL3Di providing friction vs time or friction vs other variables. (Real or integer; Default = 1.0E20)
BIAS [5,5]	Contact tolerance bias. A non-blank entry will override the BIAS entered on the BCBODY entry. (Real value between 1.0E-10 and 1.0; Default = 0.9)
SLIDE [5,6]	Delayed slide off distance. This entry should not be made unless ICOORD \geq 2 (see above). When using the delayed slide off option, a node sliding on a segment will slide off this segment only if it passes the node (2-D) or edge (3-D) at a sharp corner over a distance larger than the delayed slide off distance. By default, the delayed slide off distance is related to the dimensions of the contacted segment by a 20 percent increase of its isoparametric domain. (Real; Default)
HARDS [5,7]	Hard-soft ratio. This entry is only used if double-sided contact with automatic constraint optimization is used, (ISTYP=2 on the BCBODY entry). The hard-soft ratio can be used by the program if there is a significant difference in the (average) stiffness of the contact bodies (expressed by the trace of the initial stress-strain law). If the ratio of the stiffnesses is larger than the hard-soft ratio, the nodes of the softest body are the preferred slave nodes. (Real; Default = 2.0)
"BKGL"	Enter the character string BKGL if the line with items BGST, BGSN, BGM, BGN is required.
BGSN	Maximum normal stress for breaking glue. If the value is an integer, it is the ID of a TABLED1 or TABL3Di providing friction vs time or friction vs other variables. See Remark 10. (Real or integer; Default = 0.0)
BGST	Maximum tangential stress for breaking glue. If the value is an integer, it is the ID of a TABLED1 or TABL3Di providing friction vs time or friction vs other variables. See Remark 10. (Real or integer; Default = 0.0)

Descriptor	Meaning
BGM	First exponent for breaking glue. If the value is an integer, it is the ID of a TABLED1 or TABL3Di providing friction vs time or friction vs other variables. See Remark 10. (Real or integer; Default = 2.0)
BGN	Second exponent stress for breaking glue. If the value is an integer, it is the ID of a TABLED1 or TABL3Di providing friction vs time or friction vs other variables. See Remark 10. (Real or integer; Default = 2.0)
"HHHB"	Enter the character string HHHB if the line with items HCT, HCV, HNC, etc. is required.
HCT [7,1]/[8,1]	Contact heat transfer coefficient. If real, the value entered is the contact heat transfer coefficient. If integer, the value entered is the ID of a TABLEMi entry specifying the contact heat transfer coefficient vs temperature. (Real or Integer; Default = 0.0 for a heat transfer problem, omit for a structural problem.)
HCV [7,2]/[8,2]	Convection coefficient for near field behavior. If real, the value entered is the near field convection coefficient. If integer, the value entered is the ID of a TABLEMi entry specifying the near field convection coefficient vs temperature. (Real or Integer; Default = 0.0 for a heat transfer problem, omit for a structural problem).
HNC [7,3]/[8,3]	Natural convection coefficient for near field behavior. If real, the value entered is the near field natural convection coefficient. If integer, the value entered is the ID of a TABLEMi entry specifying the near field natural convection coefficient vs temperature. (Real or Integer; Default = 0.0 for a heat transfer problem, omit for a structural problem).
BNC [7,4]/[8,4]	Exponent associated with the natural convection coefficient for near field behavior. If real, the value entered is the exponent associated with near field natural convection coefficient. If integer, the value entered is the ID of a TABLEMi entry specifying the exponent associated with the near field natural convection coefficient vs temperature. (Real or Integer; Default = 1.0 for a heat transfer problem, omit for a structural problem.)
EMISS [7,5]/[8,5]	Emissivity for radiation to the environment or near thermal radiation. If real, the value entered is the emissivity. If integer, the value entered is the ID of a TABLEMi entry specifying the emissivity vs temperature. (Real or Integer; Default = 1.0 for a heat transfer problem, omit for a structural problem.)
HBL [7,6]/[8,6]	Separation distance dependent thermal convection coefficient. If real, the value entered is the separation distance dependent thermal convection coefficient. If integer, the value entered is the ID of a TABLEMi entry specifying the separation distance dependent thermal convection coefficient. (Real or Integer; Default = 0.0 for a heat transfer problem, omit for a structural problem).
"SEGS"	Enter the character string SEGS if segment-to-segment contact is desired (note that for MSC Nastran 2012 segment-to-segment contact cannot include friction.)

Descriptor	Meaning
PENALT	Penalty factor (stiffness) used for segment-to-segment contact analysis. (Real; Default = leave blank, meaning the program will calculate the value)
AUGDIST	Penetration distance above which additional penalty stiffness will be added. (Real; Default = leave blank, meaning the program will calculate the value)
“MASTERS”	Indicates the start of the list of bodies touched by touching body IDSLA1.
IDMAi (4,i) [15,1]	Identification numbers of BCBODY Bulk Data entries defining touched bodies. (Integer > 0)

Remarks:

1. BCTABLE defines surface contact.
2. If BCTABLE is not given, the default for contact analysis is assumed, every body detects the possibility of contact relative to all other bodies and itself if it is a deformable body. If BCTABLE is given, the default for every body is overwritten. The touching body does not contact itself unless requested. When the touched body is deformable, double-sided contact is applied by default. BCTABLE is useful for deactivating or activating bodies to reduce computational effort and to change contact conditions between subcases.
3. A short input to define two contact bodies exists if the user provides IDSLAVE and IDMAST. Then it is assumed that there are only two contact bodies, NGROUP is ignored and continuation entries are not allowed. Default values are set for the parameters on the continuation entry.
4. If the user leaves IDSLAVE and IDMast blank, then NGROUP is normally required and continuation entries are usually expected for NGROUP SLAVE/MASTER combinations. An exception if no contact is desired in increment zero or a particular subcase, fields 1 and 2 of the primary BCTABLE entry for that subcase are entered, all other fields left blank and no continuation lines are entered. The no contact condition may be achieved in either of two ways - set Case Control BCONTACT=ID and enter a matching BCTABLE with that ID in field 2 and all other fields blank or set BCONTACT=NONE and do not enter BCTABLE for that subcase.
5. The maximum number of slave-master combinations is 99 for versions prior to MSC Nastran 2005 and 999 for current versions. For MSC Nastran 2005 r3 and subsequent versions, the limit is 999.
6. The BCTABLE with ID=0 will be used in “increment 0”. This phase of Marc is purely elastic and is sometimes used to move rigid contact bodies so that they just touch flexible contact bodies. If no BCTABLE entries are desired in “increment 0”, do not enter any BCTABLEs with ID=0. In the examples shown, the first BCTABLE is used in subcase 2 and the second BCTABLE is used in phase 0.
7. The BCTABLE formats (primary or alternate) shown in the *MSC Nastran 2005 r3 Quick Reference Guide* may still be used in which case PARAM,MRCONVER,11 should not be entered if the “primary” format shown is used.

8. Many slave/masters may be entered up to the number specified in Remark 5. A new slave entry may not begin until the masters from the previous entry are finished (as shown in the example). The number of master surfaces for any given slave surface is limited by the number specified in Remark 5., however most GUI's produce one slave surface and one master surface per slave/masters pairs.
9. (i,j) indicates the ith datablock jth field of MARC's CONTACT TABLE (without tables) history definition. [i,j] indicates Marc's Contact Table (with Tables).
10. COPTS, COPTM COPTS1 and COPTM1 are packed numbers designating how the surfaces may contact using the formula

$$\text{COPxxx} = A + 10^*B + 1000^*C$$

where the following codes apply:

A: the outside of the solid elements in the body

- = 1: the outside will be in the contact description (Default)

B (flexible bodies): the outside of the shell elements in the body

- = 1: both top and bottom faces will be in the contact description, thickness offset will be included (Default)
- = 2: only bottom faces will be in the contact description, thickness offset will be included
- = 3: only bottom faces will be in the contact description, shell thickness will be ignored
- = 4: only top faces will be in the contact description, thickness offset will be included
- = 5: only top faces will be in the contact description, shell thickness will be ignored
- = 6: both top and bottom faces will be in the contact description, shell thickness will be ignored
- Note: For flexible bodies, B=6 for both bodies in a contact combination can only be used for glued contact. If the contact bodies are not glued and both bodies specify B=6, a FATAL ERROR will be issued. Should this fatal error occur, please choose a different B option and re-run the analysis.
- = 1: the rigid surface should be in the contact description (Default)

C (flexible bodies): the edges of the body

- = 1: only the beam/bar edges are included in the contact description (Default)
- = 10: only the free and hard shell edges are included in the contact description
- = 11: both the beam/bar edges and the free and hard shell edges are included in the contact description

Note that C has no effect if beam-to-beam contact is **not** switched on (BEAMB ≠ 1 on BCPARA)

11. Breaking Glue provides glued contact to all GRID's at their very 1st contact. This kind of glued-contact will break if

$$(\sigma_n/BGSN)^{**}bgn + (\sigma_t/BGST)^{**}bgm > 1.0$$

When a contact node breaks due to the above criterion, standard contact is activated if it comes into contact again. If BSGN = 0.0 the first term is ignored. If BGST is zero, the second term is ignored.

12. If the "SEGS" entry is used, "CTYPE" on BCBODY must also be entered and parameters PARAM,MRCONVER,11 and PARAM,MRCONTAB,11 must be entered.

BCTABL1

Defines a Contact Table

BCTABL1 specifies a list of contact pairs through the BCNECT option for SOLs 101, 103, 105, 107, 108, 109, 110, 111, 112, 200, 400 and 700. Only SOL 101, 400 and 700 can support all contact types: touching contact, glued, step glue and permanent glue (see [Chapter 7: Contact Types](#) of *Nastran SOL 400 Getting Started Guide*). Note that SOL 700 calls it “tie” contact to glue (also step glue or permanent glue). SOL 103, 105, 107 - 112 and standard 200 can only support permanent glue. When SOL 400 optimization, it can support all contact types. This contact table is activated in the BCONTACT Case Control command.

Format:

1	2	3	4	5	6	7	8	9	10
BCTABL1	BCID	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	-etc.-						

Examples:

BCTABL1	2	198	62	75	8	159	31	82	
	17								

BCTABL1	0	23	56						
---------	---	----	----	--	--	--	--	--	--

BCTABL1	5	31	35	THRU	40	163	THRU	169	
	1077	1088							

Describer	Meaning
BCID	Unique identification number referenced by a BCONTACT Case Control command. See Remark 3. (Integer ≥ 0 ; default=0)
IDi	Identification number of BCNECT entry (Integer > 0 or “THRU”; ID1 is required). See Remark 4. and 5.

Remarks:

1. BCTABL1 defines surface contact. If BCTABL1 does not exist, the Case Control command BCONTACT=BCID may refer to the BCNECT Bulk Data entry directly.
2. If neither BCTABL1 nor BCNECT is given, the default for contact analysis is assumed, every body detects the possibility of contact relative to all other bodies and itself if it is a deformable body. If BCTABL1 or BCNECT is given, the default for every body is overwritten. The touching body does not contact itself unless requested. When the touched body is deformable, double-sided contact is applied by default. BCTABL1 or BCNECT is useful for deactivating or activating bodies to reduce computational effort and to change contact conditions between subcases.

3. For SOLs 101 and 400, the BCTABL1 or BCONECT with ID=0 will be used in loadcase 0 automatically that does not need a corresponding Case Control command BCONTACT=0. The loadcase 0 is purely elastic and it can also be used to (1) move rigid contact bodies so that they just touch flexible contact bodies, and/or (2) remove any prestressed condition by adjusting the coordinates of the active nodes, which are the Grid Points on the surface of all deformable BCBODY1's. To place an entry in any physical loadcase (SUBCASE or STEP), the BCID must be selected by the Case Control command BCONTACT=BCID or BCONECT=BCID. When BCONTACT=ALLBODY, there is no BCID of BCTABL1 or BCONECT specified; therefore, the default values of all entries of BCONPRG and BCONPRP are assumed. Case Control command BCONTACT=ALLBODY cannot be used for permanently glued contact.
4. When the "THRU" option is used, all BCONECT entries associated with intermediate BCONECT IDs must exist. The word "THRU" may not appear in field 3 or 9 (2 or 9 for continuations).
5. If Modules are present then this entry may only be specified in the main Bulk Data section.

BCTRIM**Defines the Geometry of a Trimming Curve**

Defines the geometry of a trimming curve used to specify a NURBS for a rigid contact body.

Format:

1	2	3	4	5	6	7	8	9	10
BCTRIM	IDtrim	NPTUtrim	NORUtrim	NSUBtrim					
	"COORD"	Xisoparam1	Yisoparam1	Xisoparam2	Yisoparam2	-etc.-			
	"HOMO"	Homot1	Homot2	Homot3	-etc.-				
	"KNOT"	Knott1	Knott2	Knott3	-etc.-				

Examples:

BCTRIM	202	2	2	50					
	COORD	0.0	0.0	1.0	0.0				
	HOMO	1.	1.						
	KNOT	0.0	0.0	1.	1.				

Descriptor	Meaning
IDtrim	Unique trimming vector identification number referenced by a BCNURBS entry. (Integer > 0)
NPTUtrim	Number of control points for this trimming vector. (Integer > 0)
NORUtrim	Order for this trimming vector. (Integer > 0)
NSUBtrim	Number of subdivisions for this trimming vector. (Integer > 0)
"COORD"	Indicate the start of the list of isoparametric coordinates of points.
Xisoparam1, Yisoparam1, Xisoparam2, Yisoparam2, etc.	First and second coordinates of point in isoparametric space. There must be NPTUtrim set of (Xisoparam, Yisoparam) entries. (Real)
"HOMO"	Indicate the start of the list of homogeneous coordinates of this trimming vector.
Homot1, Homot2, Homot3, etc.	Homogeneous coordinates of this trimming vector. There must be NPTUtrim entrie. (Real; 0.0 to 1.0)
"KNOT"	Indicate the start of the list of knot vectors of this trimming vector.
Knot1, Knot2, Knot3, etc.	Knot vectors of this trimming vector. There must be (NPTUtrim+NORUtrim) entries. (Real; 0.0 to 1.0)

Remarks:

1. BCTRIM is referenced by a BCNURBS entry to define the trimming curve of a NURBS for a rigid contact body.

BDYLIST

Fluid Boundary List

Defines the boundary between a fluid and a structure.

Format:

1	2	3	4	5	6	7	8	9	10
BDYLIST	RHO	IDF1	IDF2	IDF3	IDF4	IDF5	IDF6	IDF7	
	IDF8	-etc.-							

Example:

BDYLIST	.037	AXIS	432	325	416	203	256	175	
---------	------	------	-----	-----	-----	-----	-----	-----	--

Descriptor	Meaning
RHO	Fluid mass density at boundary. (Real ≥ 0.0 or blank; Default is DRHO on the AXIF entry)
IDF1	Identification number of a RINGFL entry. (Integer > 0 or Character = “AXIS” or “LAXIS”) See Remark 7.
IDF2-IDFi	Identification number of additional RINGFL entries. (Unique Integers > 0)

Remarks:

1. This entry is allowed only if an AXIF entry is also present.
2. Each entry defines a boundary if $RHO \neq 0.0$. The order of the points must be sequential with the fluid on the right with respect to the direction of travel.
3. The word “AXIS” defines an intersection with the polar axis of the fluid coordinate system.
4. There is no limit to the number of BDYLIST entries specified. If the fluid density varies along the boundary, there must be one BDYLIST entry for each interval between fluid points.
5. The BDYLIST entry is not required and should not be used to specify a rigid boundary where structural points are not defined. Such a boundary is automatically implied by the omission of a BDYLIST.
6. If $RHO=0.0$, no boundary matrix terms will be generated to connect the GRIDB points to the fluid. This option is a convenience for structural plotting purposes. GRIDB points may be located on a fluid ring (RINGFL entry) only if the rings are included in a BDYLIST.
7. If the polar axis of the fluid coordinate system is to occur at the first point use AXIS. If the polar axis of the fluid coordinate system is to occur at the last point use LAXIS. See Remark 2.

BDYOR**CHBDYi Entry Default Values**

Defines default values for the CHBDYP, CHBDYG, and CHBDYE entries.

Format:

1	2	3	4	5	6	7	8	9	10
BDYOR	TYPE	IVIEWF	IVIEWB	RADMIDF	RADMIDB		PID	GO	
	CE	E1	E2	E3					

Example:

BDYOR	AREA4	2	2	3	3		10		
-------	-------	---	---	---	---	--	----	--	--

Descriptor	Meaning
TYPE	Default surface type. See Remark 2. (Character)
IVIEWF	Default identification number of front VIEW entry. (Integer > 0 or blank)
IVIEWB	Default identification number of back VIEW entry. (Integer > 0 or blank)
RADMIDF	Default identification number of a RADM entry for front face. (Integer ≥ 0 or blank)
RADMIDB	Default identification number of a RADM entry for back face. (Integer ≥ 0 or blank)
PID	Default PHBDY property entry identification number. (Integer > 0 or blank)
GO	Default orientation grid point. (Integer ≥ 0 ; Default = 0)
CE	Default coordinate system for defining the orientation vector. (Integer ≥ 0 or blank)
E1, E2, E3	Default components of the orientation vector in coordinate system CE. The origin of this vector is grid point G1 on a CHBDYP entry. (Real or blank)

Remarks:

- Only one BDYOR entry may be specified in the Bulk Data Section.
- TYPE specifies the type of CHBDYi element surface; allowable values are: POINT, LINE, REV, AREA3, AREA4, ELCYL, FTUBE, AREA6, AREA8, and TUBE.
- IVIEWF and IVIEWB are specified for view factor calculations only (see VIEW entry).
- GO is only used from BDYOR if neither GO nor the orientation vector is defined on the CHBDYP entry and GO is > 0 .
- E1, E2, E3 is not used if GO is defined on either the BDYOR entry or the CHBDYP entry.

BEADVAR**Topography Design Variable**

Defines design region for topography (bead or stamp) optimization.

Format:

1	2	3	4	5	6	7	8	9	10
BEADVAR	ID	PTYPE	PID	MW	MH	ANG	BF	SKIP	
	"DESVAR"	NORM/XD	YD	ZD	CID	XLB	XUB	DELXV	
	"GRID"	NGSET	DGSET						

Example using NORM:

BEADVAR	10	PSHELL	101	10.0	20.0	70.0			
---------	----	--------	-----	------	------	------	--	--	--

Example using "DESVAR" and "GRID":

BEADVAR	10	PSHELL	101	10.0	20.0	70.0		NONE	
	DESVAR	2.0	3.0	4.0		-1.0	1.0		
	GRID	100							

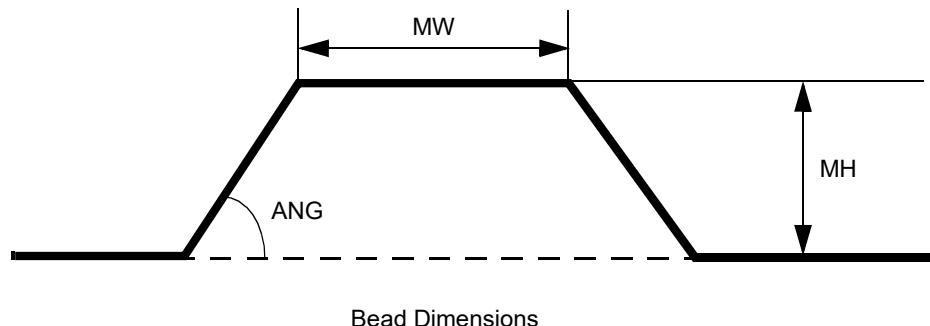
Descriptor	Meaning
ID	Unique topography design region identification number. (Integer > 0)
PTYPE	Property entry type. Used with PID to identify the element nodes to be designed. (Character: "PSHELL", "PSHEAR", "PCOMP", or "PCOMPG".)
PID	Property entry identifier. See Remark 1. (Integer > 0)
MW	Minimum bead width. This parameter controls the width of the beads. The recommended value is between 1.5 and 2.5 times the average element width. See Remark 2. (Real > 0.0)
MH	Maximum bead height (Real > 0.0). This parameter sets the maximum height of the beads when XUB=1.0 (as Default). See Remark 2.
ANG	Draw angle in degrees (0.0 < Real < 90.0). This parameter controls the angle of the sides of the beads. The recommended value is between 60 and 75 degrees.
BF	Buffer zone ('yes' or 'no'; Default='yes'). This parameter creates a buffer zone between elements in the topography design region and elements outside the design region when BF='yes'. See Remark 3.

Descriptor	Meaning
SKIP	Boundary skip (“bc”, “load”, “both”, or “none”; Default = “both”). This parameter indicates which element nodes are excluded from the design region. “bc” indicates all nodes referenced by “SPC” and “SPC1” are omitted from the design region. “load” indicates all nodes referenced by “FORCE”, “FORCE1”, “FORCE2”, “MOMENT”, “MOMENT1”, “MOMENT2”, and “SPCD” are omitted from the design region. “both” indicates nodes with either “bc” or “load” are omitted from the design region. “none” indicates all nodes associated with elements referencing PID specified in field 4 are in the design region.
“DESVAR”	Indicates that this line defines bead design variables that are automatically generated.
NORM/XD, YD, ZD	Bead vector (draw direction). Norm indicates the shape variables are created in the normal directions to the elements. If XD, YD, and ZD are provided, the shape variables are created in the direction specified by the xyz vector defined by XD/YD/ZD that is given in the basic coordinate system or CID. See Remark 4. (Character or Real; Default = blank = norm).
CID	Coordinate system ID used for specifying draw direction (Blank or Integer > 0; Default = blank = basic coordinate system)
XLB	Lower bound. (Real < XUB or blank; Default = blank=0.0). This ensures the lower bound on grid movement equal to XLB*MH. See Remark 5.
XUB	Upper bound. (Real > XLB or blank; Default = 1.0). This sets the upper bound of the beads equal to XUB*MH. See Remark 5.
DELXV	Fractional change allowed for the design variable during approximate optimization. See Remark 3. (Real > 0.0; Default = 0.2)
“GRID”	Indicates this line defines what element nodes can be added and/or removed from topography design regions.
NGSET	All grids listed on a Bulk Data entry SET1 = NGSET are removed from topography design regions.
DGSET	All grids listed on a Bulk Data entry SET1 = DGSET are added to topography design regions.

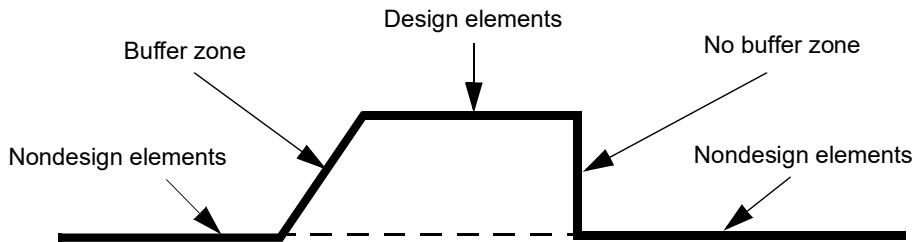
Remarks:

1. Multiple BEADVAR's are allowed in a single file. Combined topometry, topology, topography, sizing, and shape optimization is supported in a single file.

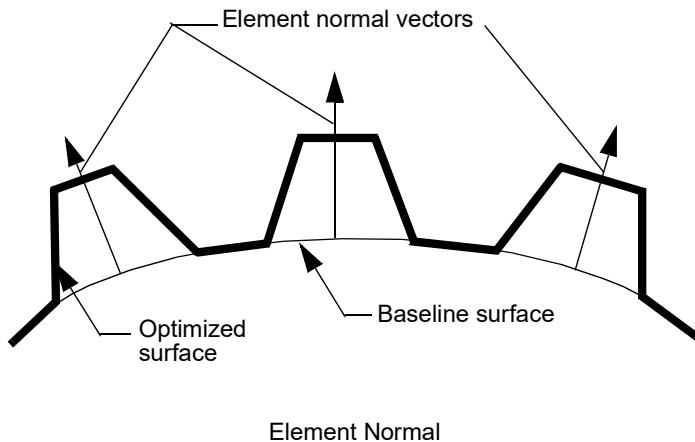
2. The user can provide allowable bead dimensions.

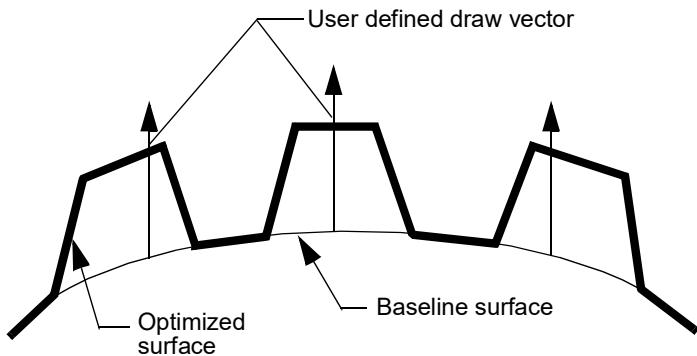


3. It is recommended to set buffer zone = yes to maintain a good quality of mesh during topography optimization.

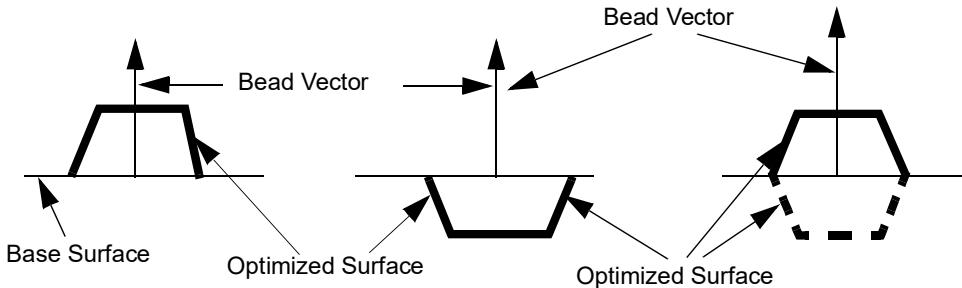


4. The grids moves in the normal direction. All element grids referenced by one BEADVAR entry must follow the right hand rule.



**User's Provided Draw Direction**

5. To force the grids to move only in the positive bead vector direction (one side of the surface), use $XLB = 0.0$. To force the grids to move only in the negative bead vector direction (another side of the surface), use $XUB = 0.0$. To allow grids to move in both positive and negative bead vector directions, use $XLB < 0.0$ and $XUB > 0.0$. For example,



(a) $XLB = 0.0$ and $XUB = 1.0$ (b) $XLB = -1.0$ and $XUB = 0.0$ (c) $XLB = -1.0$ and $XUB = 1.0$

6. The jobname.h5 file (created by specifying MDLPRM,HDF5,0 in the bulk data input) has topology optimization results (shape changes) that can be viewed in Patran. The text file jobname.pch also has updated grid coordinates that can be copied to the original file, replace the original grids, and imported to Patran and other post-processors to view topography optimization results.
7. The BEADVAR entry cannot be used with thermal loads.

BEAMOR**CBEAM Entry Default Values**

Defines default values for field 3 and fields 6 through 9 of the CBEAM entry.

Format:

1	2	3	4	5	6	7	8	9	10
BEAMOR		PID			X1	X2	X3	OFFT	

Example:

BEAMOR		39			0.6	2.9	-5.87	GOG	
--------	--	----	--	--	-----	-----	-------	-----	--

Alternate Format and Example:

BEAMOR		PID			G0			OFFT	
--------	--	-----	--	--	----	--	--	------	--

BEAMOR		39			86			GOG	
--------	--	----	--	--	----	--	--	-----	--

Descriptor	Meaning
PID	Property identification number of the PBEAM or PBCOMP entry. (Integer > 0 or blank)
X1, X2, X3	Components of orientation vector \vec{v} , from GA, in the displacement coordinate system at GA (default), or in the basic coordinate system. See Remark 5. (Real)
G0	Alternate method to supply the orientation vector \vec{v} , using grid point G0. Direction of \vec{v} is from GA to G0. \vec{v} is then translated to End A. (Integer > 0; G0 ≠ GA or GB on CBEAM entry)
OFFT	Offset vector interpretation flag. See Remark 5. (Character or blank)

Remarks:

1. The contents of fields on this entry will be assumed for any CBEAM entry with corresponding fields that are blank.
2. Only one BEAMOR entry is allowed.
3. For an explanation of beam element geometry, see the CBEAM entry description.
4. If X1 or G0 is integer, G0 is used. If X1 or G0 is blank or real, then X1, X2, X3 is used.
5. For p-version CBEAM elements, field 9 contains the value of the built-in twist measured in radians. The OFFT option cannot be used for p-elements. Otherwise, OFFT in field 9 is a character string code that describes how the offset and orientation vector components are to be interpreted. By default (string input is GGG or blank), the offset vectors are measured in the displacement coordinate

systems at grid points A and B and the orientation vector is measured in the displacement coordinate system of grid point A. At user option, the offset vectors can be measured in an offset system relative to grid points A and B, and the orientation vector can be measured in the basic system as indicated in the following table:

String	Orientation Vector	End A Offset	End B Offset
GGG	Global	Global	Global
BGG	Basic	Global	Global
GGO	Global	Global	Offset
BGO	Basic	Global	Offset
GOG	Global	Offset	Global
BOG	Basic	Offset	Global
GOO	Global	Offset	Offset
BOO	Basic	Offset	Offset

Any attempt to specify invalid combinations results in a bulk data entry input error message. For example, a value of OOO (indicating offset and orientation vectors are specified in an offset reference system) results in a fatal error since the orientation vector cannot be specified in an offset system. The offset system x-axis is defined from GA to GB. The orientation vector \hat{v} and the offset system x-axis are then used to define the z and y axes of the offset system. (Note: The character “O” in the table replaces the obsolete character “E”.)

BFRIC**Contact Friction**

Defines frictional properties between two bodies in contact using the slideline contact in SOL 106 and SOL 129. SOL 400 is the recommended approach for contact analysis.

Format:

1	2	3	4	5	6	7	8	9	10
BFRIC	FID		FSTIF	MU1					

Example:

BFRIC	33			0.3					
-------	----	--	--	-----	--	--	--	--	--

Descriptor	Meaning
FID	Friction identification number. See Remark 1. (Integer > 0)
FSTIF	Frictional stiffness in stick. See Remarks 2. and 3. (Real > 0.0; Default = automatically selected by the program)
MU1	Coefficient of static friction. (Real > 0.0)

Remarks:

1. This identification number must be unique with respect to all other friction identification numbers. This is used in the FRICID field of BCONP Bulk Data entry.
2. The value of frictional stiffness requires care. A method of choosing its value is to divide the expected frictional strength ($MU1 * \text{expected normal force}$) by a reasonable value of the relative displacement that may be allowed before slip occurs. The relative value of displacement before slip occurs must be small compared to expected relative displacements during slip. A large stiffness value may cause poor convergence, while too small a value may cause poor accuracy.
Frictional stiffness specified by the user is selected as the initial value. If convergence difficulties are encountered during the analysis, the frictional stiffness may be reduced automatically to improve convergence.
3. The stiffness matrix for frictional slip is unsymmetric. However, the program does not use the true unsymmetric matrix. Instead the program uses only the symmetric terms. This is to avoid using the unsymmetric solver to reduce CPU time.

BIAS**MESH Bias Definition**

Specifies a variation of the mesh-size in one direction for use in the MESH entry in SOL 700. The MESH entry can create a biased or non-uniform mesh. A uniform mesh consists of a number of planes separated by a fixed distance, but for a biased mesh the distance between subsequent planes can differ. The BIAS definition allows specifying the locations of planes in one direction. For a number of intervals the density of planes can be specified.

Format:

1	2	3	4	5	6	7	8	9	10
BIAS	BID								
	X0	GROWTH0	N0	DXS0	DXE0				
	X1	GROWTH1	N1	DXS1	DXE1				
	Xi	GROWTHi	Ni	DXSi	DXEi				

Example:

BIAS	100								
	-4.5	0.2	15						
	-1	1	20						
	1			0.1	0.46				

Descriptor	Meaning
BID	Unique bias number. Must be referenced from MESH entry. (Integer; Required)
Xi	Begin coordinate of an interval. The interval ends at the X_{i+1} entry. (Real; Default = 0.0)
GROWTHi	GROWTHi is the ratio between the step size at the beginning of the interval and at the end of the interval. If it is smaller than 1.0 then the mesh refines when going from the beginning of the interval to the end of the interval. (Real>0) See Remarks 1.- 3.
Ni	Ni is the number of elements inside the interval. (Integer>0) See Remarks 1.- 3.
DXSi	DXSi is the start element size of the interval. (Real>0) See Remarks 1.- 3.
DXEi	DXEi is the end element size of the interval. (Real>0) See Remarks 1.- 3.

Remarks:

1. The begin point of the first interval has to be equal to the X0 field of the MESH entry and may be left unspecified. The end point of the last interval is given by X0 + DX as specified by the MESH entry. In the example above, the first interval is given by [-4.5,-1], the second one by [-1,1] and the last one by [1,4.5], assuming that X0 + DX = 4.5 on the MESH entry that references the bias definition from IBIDX.

2. To define the bias in an interval four fields are available. These are GROWTHi, Ni, DXSi and DXEi. To specify the bias inside an interval two of these four variables have to be specified. The other two variables have to be left blank. In addition the mesh size can be chosen constant by defining Ni and leaving GROWTHi, DXSi and DXEi blank.

This gives seven methods of specifying a bias;

1. Define GROWTHi and Ni. DXSi and DXEi have to be left blank.
2. Define DXSi and DXEi. GROWTHi and Ni have to be left blank.
3. Define DXSi and Ni. GROWTHi and DXEi have to be left blank.
4. Define DXEi and Ni. GROWTHi and DXSi have to be left blank.
5. Define GROWTHi and DXSi. Ni and DXEi have to be left blank.
6. Define GROWTHi and DXEi. Ni and DXSi have to be left blank.
7. Only define Ni and leave GROWTHi, DXSi and DXEi blank.

For method 2 it can happen that the biased elements do not exactly fit into the interval.

To get a good fit a small change to defined start and end step sizes is made. These changes in general amount to a few percents.

For method 5 and 6 a small change to the specified growth factor can be made so that the biased elements fit into the interval.

3. The algorithm for each method first determines an appropriate growth factor so that the biased elements fit into the interval. In addition the number of planes is determined. For method 2, 3 and 4 this is done by using bisection. The location of the planes is given by:

$$X(i) = X(i-1) + \Delta X(i-1)$$

$$\Delta X(i) = GROWFAC \times \Delta X(i-1)$$

$$GROWTHi = GROWFAC^{N-1}$$

Here $X(i)$ denotes the location of the Euler plane, ΔX_i denotes the step size of the interval and N denotes the number of Euler planes. The index i runs across the Euler planes. The variable GROWFAC denotes the grow factor between planes within the interval. The locations of the planes $X(i)$ are written to the OUT file as part of MESH output. In addition, the growth of the element sizes is written out in the next column. This is given as the ratio in element size between the layer of elements to the right of the plane and to the left of the plane. Let x_0, x_1 and x_2 denote three subsequent planes, then the element size to the left of the x_1 -plane is given x_1-x_0 and to the right it is given by x_2-x_1 . The ratio by which the element size grows if one goes across the x_1 -plane is:

$$\frac{x_2 - x_1}{x_1 - x_0}$$

To get physically meaningful results, this value should not exceed 1.3 or be smaller than 0.7.

Within each interval the ratio in element size equals GROWFAC. But the element size between two adjacent elements that are in two different intervals can differ from GROWFAC. Here each interval has a distinct GROWFAC variable.

For both method 2, 5 and 6 The growth factor GROWFAC and start and end step sizes that are actually used can be obtained from this plane summary in the OUT file. In this summary for each plane a growth factor and step size is specified.

Also the total number of elements is written out.

BJOIN

Defines (multiple) pairs of grid points of one-dimensional and/or shell elements to be joined during the analysis in SOL 700. When the failure criterion for a grid-point pair is satisfied, the grid-point pair is removed from the join and the grid-point motion is computed for the separate grid points. The join ceases to exist when all pairs of the join have failed, after which all of the grid points of the join are treated as separate grid points.

Format:

1	2	3	4	5	6	7	8	9	10
BJOIN	BID	SID	TOL	TYPE		SN		SS	
					MULTI				
		TF							

Example:

BJOIN	1	2		SPOTWELD		1.E3		1.E3	
-------	---	---	--	----------	--	------	--	------	--

Descriptor	Meaning	Type	Default
BID	BJOIN number.	Integer > 0	Required
SID	BCGRID ID of set of grid points.	Integer > 0	Required
TOL	Tolerance used in matching grid point pairs.	Real > 0.0	1.E-4
TYPE	Type of failure criterion. SPOTWELD Spotweld-type failure.	C	SPOTWELD
SN	Failure force in tension.	Real > 0.0	No failure
SS	Failure force in shear.	Real > 0.0	No failure
MULTI	Multiple breakable joins, where the grid points must be entered as a sequence of BJOIN pairs. YES The grid points are entered on the BCGRID entry as a sequence of BJOIN pairs. NO The code creates BJOIN pairs for every two grid points entered on the BCGRID entry when the grid point positions fall within the tolerance (TOL).	C	YES
TF	Failure time for nodal constraint set.	Real	1.0E20

Remarks:

1. Nodes connected by a spot weld cannot be members of another constraint set that constrain the same degrees-of-freedom, a tied interface, or a rigid body, i.e., nodes cannot be subjected to multiple, independent, and possibly conflicting constraints. Also, care must be taken to ensure that single point constraints applied to nodes in a constraint set do not conflict with the constraint sets constrained degrees-of-freedom.
2. When the failure time, TF , is reached the spot weld becomes inactive and the constrained nodes may move freely.
3. Note that the shell elements do not have rotary stiffness in the normal direction and, therefore, this component cannot be transmitted.
4. For MULTI=YES, a spotweld or rivet will always be created, regardless if their distance is larger than the value of TOL. TOL will only be used for MULTI=NO.

BLSEG**Boundary Line Segments**

Defines a curve that consists of a number of line segments via grid numbers that may come in contact with another body.

Format:

1	2	3	4	5	6	7	8	9	10
BLSEG	ID	G1	G2	G3	G4	G5	G6	G7	

Alternate Format:

BLSEG	ID	G1	"THRU"	G2	"BY"	INC			
-------	----	----	--------	----	------	-----	--	--	--

The Continuation Entry formats may be used more than once and in any order. They may also be used with either format above.

Continuation Entry Format 1:

	G8	G9	G10	G11	-etc.-				
--	----	----	-----	-----	--------	--	--	--	--

Continuation Entry Format 2:

	G8	"THRU"	G9	"BY"	INC				
--	----	--------	----	------	-----	--	--	--	--

Example:

BLSEG	15	5	THRU	21	BY	4			
	27	30	32	33					
	35	THRU	44						
	67	68	72	75	84	93			

Descriptor	Meaning
ID	Line segments identification number. See Remark 2. (Integer > 0)
Gi	Grid point identification numbers on a curve in a continuous topological order so that the normal to the segment points toward the other curve. See Remark 3. (Integer > 0)
INC	Grid point identification number increment. See Remark 3. (Integer or blank)

Remarks:

1. A line segment is defined between every two consecutive grid points. Thus, the number of line segments defined is equal to the number of grid points specified minus 1. A corresponding BWIDTH Bulk Data entry may be required to define the width/thickness of each line segment. If the corresponding BWIDTH is not present, the width/thickness for each line segment is assumed to be unity.

2. ID must be unique with respect to all other BLSEG entries. Each line segment has a width in 3-D sideline and a thickness in a 2-D slideline contact to calculate contact stresses. The width/thickness of each line segment is defined via BWIDTH Bulk Data entry. The ID in BLSEG must be same as the ID specified in the BWIDTH. That is, there must be a one to one correspondence between BLSEG and BWIDTH. BWIDTH Bulk Data entry may be omitted only if the width/thickness of each segment is unity.
3. For automatic generation of grid numbers, the default increment value is 1 if grid numbers are increasing or -1 if grid numbers are decreasing (i.e., the user need not specify BY and the increment value).
4. The normal to the segment is determined by the cross product of the slideline plane vector (i.e., the Z direction of the coordinate system defined in the 'CID' field of BCOMP Bulk Data entry) and the tangential direction of the segment. The tangential direction is the direction from node 1 to node 2 of the line segment.
5. A curve may be closed or open. A closed curve is specified by having the last grid point identification number the same as the first grid number.
6. See BCBODY for use of BLSEG in 3D contact.

BNDFIX

Fixed Boundary Degrees-of-Freedom

Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

1	2	3	4	5	6	7	8	9	10
BNDFIX	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Example:

BNDFIX	2	135	14	6					
--------	---	-----	----	---	--	--	--	--	--

Descriptor	Meaning
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component number. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points. No embedded blanks.)

Remarks:

1. BNDFIX and BSET entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BSETi/BNDFIXi or CSETi/BNDFREE/BNDFRE1 entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi/BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are only CSETi/BNDFREE/BNDFRE1 entries present or both BSETi/BNDFIXi and CSETi/BNDFREE/BNDFRE1 entries present, the c-set degrees-of-freedom are defined by the CSETi/BNDFREE/BNDFRE1 entries, and any remaining a-set points are placed in the b-set.
3. Degrees-of-freedom specified on BSETi/BNDFIXi entries form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See the [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
 - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
 - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

BNDFIX1

Fixed Boundary Degrees-of-Freedom, Alternate Form of BNDFIX Entry

Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

1	2	3	4	5	6	7	8	9	10
BNDFIX1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	ID10	-etc.-					

Example:

BNDFIX1	2	135	14	6	23	24	25	26	
	122	127							

Alternate Format and Example:

BNDFIX1	C	ID1	"THRU"	ID2					
BNDFIX1	3	6	THRU	32					

Descriptor	Meaning
C	Component numbers. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.)
IDi	Grid or scalar point identification numbers. (Integer > 0; For "THRU" option, ID1< ID2)

Remarks:

1. BNDFIX1 and BSET1 entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BSETi/BNDFIXi or CSETi/BNDFREE/BNDFRE1 entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi/BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are only CSETi/BNDFREE/BNDFRE1 entries present or both BSETi/BNDFIXi and CSETi/BNDFREE/BNDFRE1 entries present, the c-set degrees-of-freedom are defined by the CSETi/BNDFREE/BNDFRE1 entries, and any remaining a-set points are placed in the b-set.
3. Degrees-of-freedom specified on BSETi/BNDFIXi entries form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See the [Degree-of-Freedom Sets, 1111](#) for a list of these entries.

4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
 - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
 - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

BNDFREE

Free Boundary Degrees-of-Freedom

Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during generalized dynamic reduction or component modes calculations.

Format:

1	2	3	4	5	6	7	8	9	10
BNDFREE	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Example:

BNDFREE	124	1	5	23	6	16			
---------	-----	---	---	----	---	----	--	--	--

Descriptor	Meaning
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component numbers. (Integer zero or blank for scalar points, or any unique combination of the Integers 1 through 6 for grid points with no embedded blanks.)

Remarks:

1. BNDFREE and CSET entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BSETi/BNDFIXi or CSETi/BNDFREE/BNDFRE1 entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi/BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are only CSETi/BNDFREE/BNDFRE1 entries present or both BSETi/BNDFIXi and CSETi/BNDFREE/BNDFRE1 entries present, the c-set degrees-of-freedom are defined by the CSETi/BNDFREE/BNDFRE1 entries, and any remaining a-set points are placed in the b-set.
3. Degrees-of-freedom specified on CSETi/BNDFREE/BNDFRE1 entries form members of the mutually exclusive c-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
 - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
 - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

BNDFRE1

Free Boundary Degrees-of-Freedom, Alternate Form of BNDFREE Entry

Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during generalized dynamic reduction or component modes calculations.

Format:

1	2	3	4	5	6	7	8	9	10
BNDFRE1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	-etc.-						

Example:

BNDFRE1	124	1	5	7	6	9	12	122	
	127								

Alternate Formats and Examples:

BNDFRE1	C	ID1	"THRU"	ID2					
BNDFRE1	3	6	THRU	32					

BNDFRE1		"ALL"							
BNDFRE1		ALL							

Descriptor	Meaning
C	Component number. (Integer zero or blank for scalar points, or any unique combination of the Integers 1 through 6 for grid points with no embedded blanks)
IDi	Grid or scalar point identification number. (Integer > 0; For THRU option, ID1 < ID2)
ALL	All a-set degrees-of-freedom will be set free (included in c-set).

Remarks:

1. BNDFRE1 and CSET1 entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BSETi/BNDFIXi or CSETi/BNDFREE/BNDFRE1 entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi/BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are only CSETi/BNDFREE/BNDFRE1 entries present or both BSETi/BNDFIXi and CSETi/BNDFREE/BNDFRE1 entries present, the c-set degrees-of-freedom are defined by the CSETi/BNDFREE/BNDFRE1 entries, and any remaining a-set points are placed in the b-set.
3. Degrees-of-freedom specified on CSETi/BNDFREE/BNDFRE1 entries form members of the mutually exclusive c-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.

4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
 - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
 - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

BNDGRID**Boundary Grid Points**

Specifies a list of grid point identification numbers on design boundaries or surfaces for shape optimization (SOL 200).

Format:

1	2	3	4	5	6	7	8	9	10
BNDGRID	C	GP1	GP2	GP3	GP4	GP5	GP6	GP7	
	GP8	-etc.-							

Example:

BNDGRID	123	41	42	43	44	45	46	47	
	49								

Alternate Format and Example:

BNDGRID	C	GP1	"THRU"	GP2					
BNDGRID	123	41	THRU	49					

Descriptor	Meaning
C	Component number (any unique combination of integers 1 through 6 with no embedded blanks). See Remark 1.
GPi	Shape boundary grid point identification number. (0 < Integer < 1000000; For THRU option, GP1 < GP2)

Remarks:

1. C specifies the components for the listed grid points for which boundary motion is prescribed.
2. Multiple BNDGRID entries may be used to specify the shape boundary grid point identification numbers.
3. Both fixed and free shape boundary grid point identification numbers are listed on this entry.
4. The degrees-of-freedom specified on BNDGRID entries must be sufficient to statically constrain the model.
5. Degrees-of-freedom specified on this entry form members of the mutually exclusive s-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1111](#) for a list of these entries.

BOLT

Defines the Multi-Point Constraints for a Bolt

Defines a rigid bolt by a set of MPC constraints.

Format:

1	2	3	4	5	6	7	8	9	10
BOLT	ID	GRIDC							
	TOP	GT1	GT2	GT3	GT4	GT5	GT6	GT7	
		GT8	GT9	etc.					
	BOTTOM	GB1	GB2	GB3	GB4	GB5	GB6	GB7	
		GB8	GB9	etc.					

Example:

BOLT	100	1025							
	TOP	101	102	103	104	105			
	BOTTOM	1	2	3	4	5			

Descriptor	Meaning
ID	Element ID of the bolt. (Integer; Required; no Default)
GRIDC	Control GRID ID where forces or displacements are applied. (Integer; no Default; Required)
TOP	Enter the character string TOP to define the start of the entry that defines all of the grids at the “top” of the bolt intersection with the structure. (Integer; no Default)
GT1, GT2, etc.	Grid IDs of the grid points at the top of the bolt intersection. (Integer; no Default)
BOTTOM	Enter the character string BOTTOM to define the start of the entry that defines all of the grids at the “bottom” of the bolt intersection with the structure (do not enter the ID for GRIDC). (Integer; no Default)
GB1, GB2, etc.	Grid IDs of the grid points at the bottom of the bolt intersection. (Integer; no Default)

Remarks:

1. The GRIDS entries of the TOP and BOTTOM keywords are open-ended.
2. GRIDC is the control grid point and usually not connected to any element.
3. (GT_i, GB_i) are pairs of grid on top and bottom.
4. To each pair of (GT_i, GB_i) and GRIDC, MPCs are created internally to all 6 DOFs. Since the GBs always belong to dependent-DOFs, they cannot be applied to any SPC, SPC1, SPCD and SPCR.
5. Same number of grid points in TOP and BOTTOM. They should be coincident but it is not required. Users who do otherwise do so at their own risk since the current design does not consider the initial offset between them.

6. Bolt loads, including enforced motion, are usually prescribed on GRIDC to represent the pre-tension, overlap or loading of the bolt. BOLT relative displacements are given in the global coordinate system of the control node.
7. Global Coordinate System may have to be defined at the Control Node if the bolt direction is not a Basic Coordinate direction and the user wants to apply the loads along the shaft of the bolt.
8. Loads in directions other than the shaft of the bolt direction are possible.
9. The internally written MPC relationship is of the form:

$$u_{GB} = u_{GT} + u_{GC}$$

10. In 3D Contact analysis, it replaces GBi (Bottom bolt segment) by GTi (Top bolt segment) on the internally generated contact surface, which makes contact surface continuous across the mesh split between them.
11. Force output is obtained through Case Control MPCFORCE.
12. The internal MPCs generated by the BOLT entry are valid for small rotations only.
13. The BOLT entry must be defined in the residual structure only.
14. Sufficient boundary conditions must be placed on the BOLT to prevent mechanisms.
15. In contact analysis, care should be taken that the preload displacement does not exceed the local element width.
16. Pretension of a BOLT is a three step process in SOL 400:

SUBCASE 1

STEP 1

- LOAD=n Applies a pretension load to the bolt on the control grid.
 SPC=m If any required to prevent motion of structure do not include the control grid.

STEP 2

- LOAD=p1 Load entry pointing to a SPCR (relative value of 0.0) and any other desired type of loading.
 SPC=q1 SPC or SPC1 includes control grid and any other desired SPC requirements.

STEP 3

- LOAD=p2 Load entry pointing to a SPCR and any other additional desired type of loading.
 SPC=q2 SPC or SPC1 includes control grid and any other additional desired SPC requirements.

17. The BOLT may also be used in SOL101 and SOL103. In this case PARAM, AUTOMSET, YES is recommended.

BOUTPUT

Output for Slideline Contact

Defines slave nodes at which output is requested.

Format:

1	2	3	4	5	6	7	8	9	10
BOUTPUT	ID	G1	G2	G3	G4	G5	G6	G7	

Alternate Format:

BOUTPUT	ID	G1	"THRU"	G2	"BY"	INC			
---------	----	----	--------	----	------	-----	--	--	--

The Continuation Entry formats may be used more than once and in any order. They may also be used with either format above.

Continuation Entry Format 1:

	G8	G9	G10	G11	-etc.-				
--	----	----	-----	-----	--------	--	--	--	--

Continuation Entry Format 2:

	G8	"THRU"	G9	"BY"	INC				
--	----	--------	----	------	-----	--	--	--	--

Example:

BOUTPUT	15	5	THRU	21	BY	4			
	27	30	32	33					
	35	THRU	44						
	67	68	72	75	84	93			

Format and Example Using "ALL" (No continuation entry is allowed):

BOUTPUT	ID	ALL							
BOUTPUT	15	ALL							

Descriptor	Meaning
ID	Contact region identification number of a BCOMP entry for which output is desired, or the contact Grid ID, in 3D contact. (Integer > 0)
Gi	Slave node numbers for which output is desired. (Integer > 0)
INC	Grid point identification number increment. See Remark 1. (Integer or blank)

Remark:

1. For automatic generation of grid numbers, the default increment value is 1 if grid numbers are increasing or -1 if grid numbers are decreasing (i.e., the user need not specify BY and the increment value).

BRKSQ**Specifies Data for Brake Squeal Calculations In SOL 600**

Specifies data for brake squeal calculations in SOL 600 only. Obsolete, please see [BSQUEAL, 1439](#) for an enhanced version.

Format:

1	2	3	4	5	6	7	8	9	10
BRKSQ	METH	AVSTIF				GLUE	ICORD		
	R1	R2	R3	X	Y	Z			
	NASCMD								
	RCFILE								

Example:

BRKSQ	1	5.34E6							
	0.0	0.0	1.0	2.0	3.0	4.0			
	tran								
	nastb								

Descriptor	Meaning
METH	Method flag corresponding to the type of brake squeal calculations to be performed. (Integer; Default = 1) 0 = Performs brake squeal calculations before any nonlinear analysis has taken place (corresponds to Marc feature, 4302). 1 = Performs brake squeal calculations after all nonlinear load cases (corresponds to Marc feature, 4304). -1 = Same as ID=0 except it corresponds to Marc feature, 4301 (not recommended).
AVSTIF	Approximate average stiffness per unit area between the pads and disk. Corresponds to Marc's PARAMETERS fifth datablock, field 1. This value is also known as the initial friction stiffness in Marc Volume C documentation. AVSTIF is used as a penalty contact stiffness for brake squeal, it needs to be a large value but not so large that numerical instabilities result. If AVSTIF is large enough, increasing it by a few orders of magnitude will not appreciably affect the squeal modes. (Real; no Default. Required field)
GLUE	Flag specifying whether MPC for non-pad/disk surfaces with glued contact are used or ignored. A value of 0 means ignore the MPC; a value of 1 means include the MPCs. See Remark 6. (Integer; Default = 1)

Descriptor	Meaning
ICORD	Flag indicating whether coordinates are updated or not. A value of 0 means coordinates are not updated. A value of 1 means coordinates are updated using the formula Cnew=Corig+Defl where Cnew are updated coordinates, Corig are original coordinates, and Defl are the final displacements from last Marc increment. (Integer; Default = 0)
R1	X direction cosine (basic coord system) of axis of rotation; corresponds to Marc ROTATION A second datablock. (Real; no Default. Required field)
R2	Y direction cosine (basic coord system) of axis of rotation; corresponds to Marc ROTATION A second datablock
R3	Z direction cosine (basic coord system); corresponds to Marc ROTATION A second datablock. (Real; no Default. Required field)
X	X coordinate in basic coord system of a point on the axis of rotation; corresponds to Marc ROTATION A third datablock. (Real; no Default. Required field)
Y	Y coordinate in basic coord system of a point on the axis of rotation; corresponds to Marc ROTATION A third datablock. (Real; no Default. Required field)
Z	Z coordinate in basic coord system of a point on the axis of rotation; corresponds to Marc ROTATION A third datablock. (Real; no Default. Required field)
NASCMD	Name of a command to run Nastran (limited to 64 characters) -- used in conjunction with the CONTINUE options on the SOL 600 entry. The full path of the command to execute Nastran should be entered. Enter the string entirely in lower case. The string will be converted to lower case. See Remark 2. (Character; Default=nastran)
RCFILE	Name of an RC file to be used with a secondary Nastran job (limited to 8 characters) -- used in conjunction with the CONTINUE options on the SOL 600 entry. An extension of ".rc" will automatically be added. Enter the string entirely in lower case. See Remark 2. (Character; Default=nastb.rc)

Remarks:

1. This entry is used to calculate complex eigenvalues for brake squeal using unsymmetric stiffness friction matrices calculated by Marc. Options exist to obtain the unsymmetric stiffness matrices using the undeformed geometry (initial contact) or after all specified nonlinear subcases.
2. SOL 600 performs brake squeal calculations, using the following approach. The main (original) Nastran job with input file jid.dat or jid.bdf spawns Marc just as it does for any other SOL 600 job. Marc calculates unsymmetric friction stiffness matrices that are saved on a file (jid.marc.bde with associated file jid.marc.ccc). The primary Nastran job then creates input data for a second Nastran job (jid.nast.dat) to use the unsymmetric stiffness matrices in an complex eigenvalue extraction. The primary Nastran job spawns a second Nastran job to calculate the complex eigenvalues. The complex eigenvalues and eigenvectors are found in jid.nast.f06, jid.nast.op2, etc.

NASCMD is the name of the command to execute the secondary Nastran job. NASCMD can be up to 64 characters long and must be left justified in field 2. The sting as entered will be used as is - except that it will be converted to lower case regardless of whether it is entered in upper or lower case.

RCFILE is the name of an RC file to be used for the secondary Nastran job. Normally it should be similar to the RC file used for the primary run except that additional memory will normally be necessary to calculate the complex eigenvalues and batch=no should also be specified for Linux systems. RCFI^L is limited to 8 characters and an extension of ".rc" will be added automatically. This entry will be converted to upper case in Nastran but will be converted to lower case before spawning the complex eigenvalue run. This RC file must be located in the same directory as the Nastran input file. This entry is the same as specifying PARAM,MRRCFILE. Only one or the other should be used.

3. MPCs are produced for contact surfaces with glued contact. DMIGs are produced for contact surfaces without glued contact. The brakes and drums should not use glued contact; other regions of the structure can use glued contact.
4. The continuation lines may be omitted if defaults are appropriate.
5. When a BRKSQ^L entry is used, PARAM,MRMTXNAM and PARAM,MARCFIL1 should not be entered.
6. When brake squeal matrices are output by Marc, unsymmetric friction stiffness matrices are output for non-glued contact surfaces. For surfaces with glued contact, MPCs are output. The GLUE flag signals SOL 600 to look for these MPCs and combine them with other MPCs that might be in the model using MPCADD, or if no MPCs were originally used to add the MCPs due to glued contact. Glued contact surfaces may not be used for the disk-rotor interface. If IGLUE is zero or blank, the MPCs for glued contact in the Marc brake squeal bde file (if any) will be ignored. Sometimes, Marc puts out MPCs with only one degree-of-freedom defined. Such MPCs will be ignored; otherwise Nastran will generate a fatal error.
7. If METH=1, a Marc t19 file will be produced.
8. The names NASCMD and RCFI^L must be entered in small fixed field and start in column 9 (i.e., left justified in the field).
9. The Nastran input file name used for a brake squeal analysis may only contain lower case letters and the underscore and/or dash characters.
10. Brake squeal is not available with DDM (parallel processing). Do not enter a PARAMMARC when using the BRKSQ^L entry.

BSET

Fixed Analysis Degrees-of-Freedom

Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

1	2	3	4	5	6	7	8	9	10
BSET	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Example:

BSET	2	135	14	6					
------	---	-----	----	---	--	--	--	--	--

Descriptor	Meaning
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component number. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points. No embedded blanks.)

Remarks:

1. BSET and BNDFIX entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BSETi/BNDFIXi or CSETi/BNDFREE/BNDFRE1 entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi/BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are only CSETi/BNDFREE/BNDFRE1 entries present or both BSETi/BNDFIXi and CSETi/BNDFREE/BNDFRE1 entries present, the c-set degrees-of-freedom are defined by the CSETi/BNDFREE/BNDFRE1 entries, and any remaining a-set points are placed in the b-set.
3. Degrees-of-freedom specified on BSETi/BNDFIXi entries form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See the [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
 - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
 - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

BSET1

Fixed Analysis Degrees-of-Freedom, Alternate Form of BSET Entry

Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

1	2	3	4	5	6	7	8	9	10
BSET1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	ID10	-etc.-					

Example:

BSET1	2	135	14	6	23	24	25	26	
	122	127							

Alternate Format and Example:

BSET1	C	ID1	"THRU"	ID2					
BSET1	3	6	THRU	32					

Descriptor	Meaning
C	Component numbers. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.)
IDi	Grid or scalar point identification numbers. (Integer > 0; For "THRU" option, ID1< ID2)

Remarks:

1. BSET1 and BNDFIX1 entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BSETi/BNDFIXi or CSETi/BNDFREE/BNDFRE1 entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi/BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are only CSETi/BNDFREE/BNDFRE1 entries present or both BSETi/BNDFIXi and CSETi/BNDFREE/BNDFRE1 entries present, the c-set degrees-of-freedom are defined by the CSETi/BNDFREE/BNDFRE1 entries, and any remaining a-set points are placed in the b-set.
3. Degrees-of-freedom specified on BSETi/BNDFIXi entries form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See the [Degree-of-Freedom Sets, 1111](#) for a list of these entries.

4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
 - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
 - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

BSQUEAL**Specifies Data for Brake Squeal Analysis Using SOLs 400 and 600**

Defines data for brake squeal calculations involving one wheel (Primary Format) or multiple wheels (Alternate Format) as well as other rotating or stationary sliding objects used in SOLs 400 and 600.

Primary Format (One Wheel, SOL 400 and SOL 600):

1	2	3	4	5	6	7	8	9	10
BSQUEAL	ID	OMETH	AVSTIF			BSONLY	IGLUE	ICORD	
	RX	RY	RZ	X	Y	Z			
	NASCMD								
	RCFILE								

Alternate Format (Multiple Wheels, SOL 600 Only):

1	2	3	4	5	6	7	8	9	10
BSQUEAL	ID	OMETH	NGROUP	ICORD	ITYPE				
	“BODY”	ID1	ID2			AVSTIF	T(AVSTIF)		
		IVEC	IDRF1	IDRF2	IGLUE				
		RX1	RY1	RZ1	X1	Y1	Z1		
		RX2	RY2	RZ2	X2	Y2	Z2		
	“BODY”	ID1	ID2			AVSTIF	T(AVSTIF)		
		IVEC	IDRF1	IDRF2	IGLUE				
		RX1	RY1	RZ1	X1	Y1	Z1		
		RX2	RY2	RZ2	X2	Y2	Z2		
	NASCMD								
	RCFILE								

Example, Primary Format (SOL 400):

1	2	3	4	5	6	7	8	9	10
BSQUEAL	100	0.2	5.34E6			NO			
	0.0	0.0	1.0	2.0	3.0	4.0			

Example, Primary Format (SOL 600):

1	2	3	4	5	6	7	8	9	10
BSQUEAL		1.0	5.34E6						
	0.0	0.0	1.0	2.0	3.0	4.0			
	nastran								
	nastb.rc								

Example, Alternate Format (SOL 600):

1	2	3	4	5	6	7	8	9	10
BSQUEAL		1.0	2	1	1				
	BODY	2	3			60000.	0		
		0			1				
		0.0	0.0	1.0	10.0	10.0	25.0		
		0.0	0.0	-1.0	10.0	10.0	30.0		
	BODY	6	8			1.0	5		
		0	110	120	1				
	nastran								
	nastb.rc								

Descriptor	Meaning	
ID	Identification number of a corresponding BSQUEAL Case Control command (Integer > 0). Ignored by SOL 600.	
OMETH	Specifies the corresponding load factor (or time step) where the brake squeal analysis is to be performed. (Real; Default = 0.0 in SOL 400, see Remark 5., for SOL 600 the value must be 0.0 or 1.0, Remark 11.)	
NGROUP	Number of pairs of contact bodies (Integer > 0; Default = 1)	
ICORD (SOL 600 Only)	Flag indicating whether coordinates are updated or not. A value of 0 means coordinates are not updated. A value of 1 means coordinates are updated using the formula Cnew=Corig+Defl where Cnew are updated coordinates, Corig are original coordinates, Defl are the final displacements from last Marc increment. (Integer; Default = 0)	
ITYPE (SOL 600 Only)	Flag determining type of analysis. See Figure 9-3 and Figure 9-4 (Integer; Default = 1)	
	1	Brake squeal
	2	Complex eigenvalue analysis following general sliding contact analysis.
BSONLY (SOL 400 Only)	Brake-Squeal-Only flag to control whether or not to continue nonlinear iterations after brake squeal analysis is performed. BSONLY=YES means to perform brake squeal analysis only and exit nonlinear iteration immediately; BSONLY=NO means to continue nonlinear analysis. (Character; Default = YES)	
“BODY”	Character string that signals the start of a pair of contact bodies for which instability calculations are to be made (Character; Required)	
ID1	BCBODY ID of the first contact body. (Integer > 0; no Default; bcbody id1 must exist)	
ID2	BCBODY ID of the second contact body. (Integer > 0; no Default; bcbody id2 must exist)	

Descriptor	Meaning				
AVSTIF	Approximate average stiffness per unit area between the two surfaces. AVSTIF is used a penalty contact stiffness, it needs to be a large value but not so large that numerical instabilities result. If AVSTIF is large enough, increasing it by a few orders of magnitude will not appreciably affect the squeal modes. (Real; no Default; Required field)				
T(AVSTIF)	Table ID of a TABL3Di providing temperature or spatial variation of the average stiffness (Integer; Default = 0 which means FC is constant)				
IVEC	Flag specifying whether friction vector at center of master contact body is in the same direction as that of the slave nodes. (Integer; Default = 0) <table> <tr> <td>0</td><td>In same direction</td></tr> <tr> <td>1</td><td>Slave vector is in tangential direction</td></tr> </table>	0	In same direction	1	Slave vector is in tangential direction
0	In same direction				
1	Slave vector is in tangential direction				
IDRFi (SOL 600 Only)	ID of a matching RFORCE entry (if any) for body 1 or 2. (Integer; Default = 0, which means this body is not rotating)				
IGLUE (SOL 600 Only)	Flag specifying whether MPC's for non-pad/disk surfaces with glued contact are used or ignored. A value of 0 means ignore the MPC's, a value of 1 means include the MPC's. See Remarks 12. and 14. (Integer; Default = 1)				
RXi	X direction cosine (basic coord system) of axis of rotation for body i if IDRFi is zero or blank (Real; or blank)				
RYi	Y direction cosine (basic coord system) of axis of rotation for body i if IDRFi is zero or blank (Real; or blank)				
RZi	Z direction cosine (basic coord system) of axis of rotation for body i if IDRFi is zero or blank (Real; or blank)				
Xi	X coordinate in basic coord system of a point on the axis of rotation for body i if IDRFi is zero or blank (Real; or blank)				
Yi	Y coordinate in basic coord system of a point on the axis of rotation for body i if IDRFi is zero or blank (Real; or blank)				
Zi	Z coordinate in basic coord system of a point on the axis of rotation for body i if IDRFi is zero or blank (Real; or blank)				
NASCMD (SOL 600 Only)	Name of a command to run Nastran (limited to 64 characters) - used in conjunction with the CONTINUE options on the SOL 600 statement. The full path of the command to execute Nastran should be entered. Enter the string entirely in lower case. (Character; Default = nastran) If the string exceeds 8 characters, do not enter this string using free-field also see Remark 9.				
RCFILE (SOL 600 Only)	Name of a RC file to be used with a secondary Nastran job (limited to 8 characters) - used in conjunction with the CONTINUE options on the SOL 600 statement. An extension of ".rc" will automatically be added. Enter the string entirely in lower case. (Character; Default=nastb.rc) If the string exceeds 8 characters, do not enter this string using free-field also see Remark 9.				

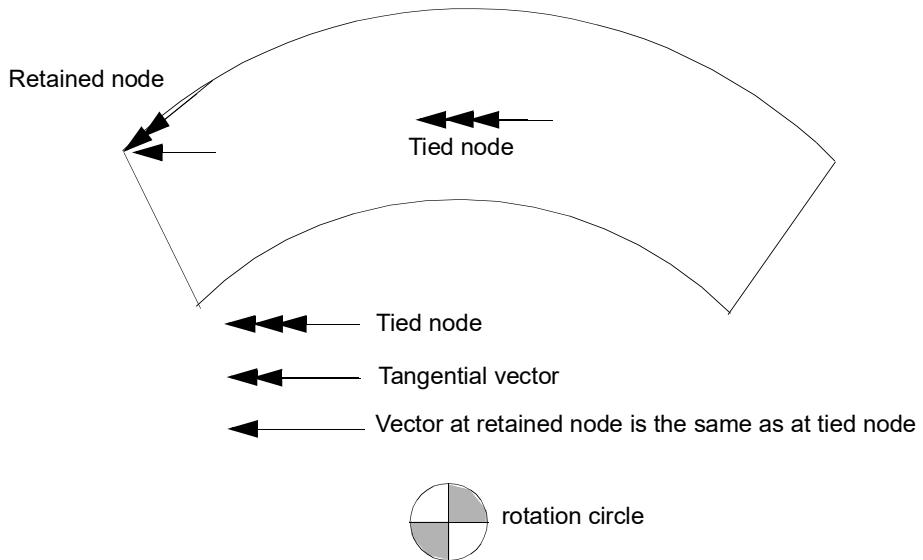


Figure 9-3

Vectors of frictional stiffness at tied and retained node.

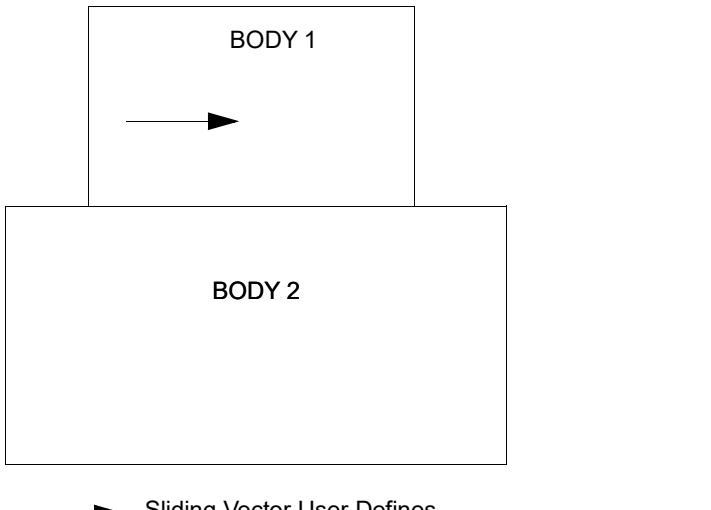


Figure 9-4

Sliding model

Remarks:

1. This entry is used to perform multiple-body brake squeal analysis or other contact with friction sets of bodies using unsymmetric stiffness friction matrices.

2. One or more pairs of bodies may be specified. Each pair of bodies is described by the four lines beginning with the header "BODY". For example, there may be one rotating body, such as brake disk or rotor, and one contact body or bodies, such as brake pad under each "BODY" keyword. There may be other bodies or parts in the assembly.
3. The disks should not be glued with pads. When bodies are intended to be glued, turn on BCTABLE / IGLUE or BCONPRG/IGLUE for those contact bodies.
4. Entries ICORD, ITYPE, IDBFI, IGLUE, NASCMD and RCFI are used by SOL 600 only.
5. Entry BSONLY is used by SOL 400 only.
6. Brake squeal is not available with segment-to-segment contact.

SOL 400 Only

7. For nonlinear static analysis, $0.0 < \text{OMETH} < 1.0$, OMETH is overridden by the load factor from Case Control command, NLIC or its default, when a brake squeal analysis is performed in a separate SUBCASE-STEP other than ANALYSIS = NLSTATIC.
8. User subroutine UBSQUEAL is available using the BCONUDS entry.

SOL 600 Only

9. SOL 600 performs brake squeal calculations as follows. The main (original) Nastran job with input deck jid.dat or jid.bdf spawns Marc just as done for any other SOL 600 job. Marc calculates unsymmetric friction stiffness matrices which are saved on a file (jid.marc.bde with associated file jid.marc.ccc). The primary Nastran job then creates input data for a second Nastran job (jid.nast.dat) to use the unsymmetric stiffness matrices in an complex eigenvalue extraction. The primary Nastran job spawns a second Nastran job to calculate the complex eigenvalues. The complex eigenvalues and eigenvectors are found in jid.nast.f06, jid.nast.op2, etc.
NASCMD is the name of the command to execute the secondary Nastran job. NASCMD can be up to 64 characters long and must be left justified in field 2. The sting as entered will be used as is - except that it will be converted to lower case regardless of whether it is entered in upper or lower case. RCFI is the name of an RC file to be used for the secondary Nastran job. Normally it should be similar to the RC file used for the primary run except that additional memory will normally be necessary to calculate the complex eigenvalues and batch=no should also be specified. RCFI is limited to 8 characters and an extension of ".rc" will be added automatically. This entry will be converted to upper case in Nastran but will be converted to lower case before spawning the complex eigenvalue run. This RC file must be located in the same directory as the Nastran input file. This entry is the same as specifying PARAM,MRRCFILE. Only one or the other should be used.
10. OMETH can only be one of the following two values in SOL 600.
0.0=Perform brake squeal calculations before any nonlinear analysis has taken place
1.0=Perform brake squeal calculations after all nonlinear load cases
11. Options exist to obtain the unsymmetric stiffness matrices using the undeformed geometry (initial contact) or after all specified nonlinear subcases.
12. MPC's are produced for contact surfaces with glued contact. DMIG's are produced for contact surfaces without glued contact. The brakes and drums may not use glued contact, other regions of the structure can used glued contact.

13. When the BSQUEAL entry is used, PARAM,MRMTXNAM and PARAM,MARCFIL1 should not be entered.
14. When brake squeal matrices are output by Marc, unsymmetric friction stiffness matrices are output for non-glued contact surfaces. For surfaces with glued contact, MPC's are output. The GLUE flag signals SOL 600 to look for these MPC's and combine them with other MPC's that might be in the model using MPCADD, or if no MPC's were originally used to add the MCP's due to glued contact. Glued contact surfaces may not be used for the disk-rotor interface. If IGLUE is zero or blank, the MPC's for glued contact in the Marc brake squeal bde file (if any) will be ignored. Sometimes, Marc puts out MPC's with only one degree-of-freedom defined. Such MPC's will be ignored otherwise Nastran will generate a fatal error.
15. The names NASCMD and RCFILE must be entered in small fixed field and start in column 9 (i.e. left justified in the field).
16. The Nastran input file name used for a brake squeal analysis may only contain lower case letters and the underscore and/or dash characters.
17. Brake squeal is not available with DDM (parallel processing). Do not enter a PARAMMARC when using the BSQUEAL entry.
18. The Bulk Data entry BRKSQ is the alternate format in SOL 600 when there is only one body.

BSURF

Contact Body or Surface

Defines a contact body or surface by Element IDs.

Format:

1	2	3	4	5	6	7	8	9	10
BSURF	ID	ELID1	ELID2	ELID3	ELID4	ELID5	ELID6	ELID7	
	ELID8	ELID9	etc.						

Alternate Format:

BSURF	ID	ELID1	THRU	ELID2	BY	INC			
	ELID3	THRU	ELID4	BY	INC2				

Example:

BSURF	15	5	THRU	21	BY	4			
	27	30	32	33					
	35	THRU	44						
	67	68	72	75	84	93			

Descriptor	Meaning
ID	Identification of a deformable surface corresponding to a BSID value on the BCBODY entry. See Remark 2. (Integer > 0)
ELIDI	Element identification numbers. If the curve or surface is defined with element ids only, the direction of the normal depends on the grid point numbering. Keywords THRU and BY can be used to assist the listing. For SOL 600 and SOL 700 the Alternate Format must be used with Remarks 6. and 7.
INC	Identification number increment. See Remark 3. (Integer or blank)

Remarks:

1. BSURF can be used in SOL101, 103, 105, 107, 108, 109, 110, 111, 112, 400, 600, and 700.
2. ID must be unique with respect to all other BSURF, BCBOX, BCPROP, and BCMATL entries.
3. For automatic generation of element IDs, the default increment value is 1 if element numbers are increasing or -1 if element numbers are decreasing (i.e., the user need not specify BY and the increment value).
4. The deformable surface may alternately be defined using BCBOX, BCPROP, or BCMATL entries.
5. Only one kind of entry (BSURF, BCBOX, BCPROP, or BCMATL) may be used to define a particular deformable surface.

6. For SOLs 600 and 700, Format 1 and the Alternate Format cannot be mixed for a particular BSURF entry.
7. For SOLs 600 and 700 if the Alternate Format is used, THRU must be in column 4 for entries with the BSURF header and in column 3 for continuation entries. If BY is not used, columns 6-9 of the BSURF entry and columns 5-9 of continuation entries must be blank.

BWIDTH**Boundary Line Segment Width or Thickness**

Defines widths or thicknesses for line segments in 3-D or 2-D slideline contact defined in the corresponding BLSEG Bulk Data entry for SOL 106 or SOL 129. SOL 400 general contact capability, defined with BCONTACT, BCTABL1, etc. is the recommended approach for contact analysis.

Format:

1	2	3	4	5	6	7	8	9	10
BWIDTH	ID	W1	W2	W3	W4	W5	W6	W7	

Alternate Format:

BWIDTH	ID	W1	"THRU"	W2	"BY"	INC			
--------	----	----	--------	----	------	-----	--	--	--

The continuation entry formats may be used more than once and in any order. They may also be used with either format above.

Continuation Entry Format 1:

	W8	W9	W10	W11	-etc.-				
--	----	----	-----	-----	--------	--	--	--	--

Continuation Entry Format 2:

	W8	"THRU"	W9	"BY"	INC				
--	----	--------	----	------	-----	--	--	--	--

Example:

BWIDTH	15	2.0	THRU	5.0	BY	1.0			
	2.0	2.0	2.0	2.0					
	35.	THRU	44.						
	1.5	3.4	7.6	0.4	0.7				

Descriptor	Meaning
ID	BLSEG entry identification number. (Integer > 0)
Wi	Width values for the corresponding line segments defined in the BLSEG entry. See Remark 1. (Real > 0.0)
INC	Width value increment. See Remark 2. (Real or blank)

Remarks:

1. BWIDTH may be omitted if the width of each segment defined in the BLSEG entry is unity. The number of widths to be specified is equal to the number of segments defined in the corresponding BLSEG entry.

2. The default value for INC is 1.0 if the width is increasing or -1.0 if the width is decreasing. That is, the user need not specify BY and the increment value. If the number of widths specified is less than the number of segments defined in the corresponding BLSEG entry, the width for the remaining segments is assumed to be equal to the last width specified.
3. If there is only one grid point in the corresponding BLSEG entry, there is no contributory area associated with the grid point. To compute correct contact stresses an area may be associated with the single grid point by specifying the area in field W1.

Entries CA - CM

CAABSF

Frequency-Dependent Acoustic Absorber Element

Defines a frequency-dependent acoustic absorber element in coupled fluid-structural analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CAABSF	EID	PID	G1	G2	G3	G4			

Example:

CAABSF	44	38	1	10	20				
--------	----	----	---	----	----	--	--	--	--

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number that matches a PAABSF entry. (Integer > 0; Default = EID)
Gi	Grid point identification number of fluid connection points. (Integer ≥ 0 or blank)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. If only G1 is specified then a point impedance is assumed. If G1 and G2 are specified then a line impedance is assumed. If G1, G2, and G3 are specified, then an impedance is associated with the area of the triangular face. If G1 through G4 are specified, then an impedance is associated with the quadrilateral face. See [Figure 9-5](#).
3. The CAABSF element must connect entirely to fluid points on the fluid-structure boundary.
4. This element is used only in frequency response and is ignored in all other solutions.

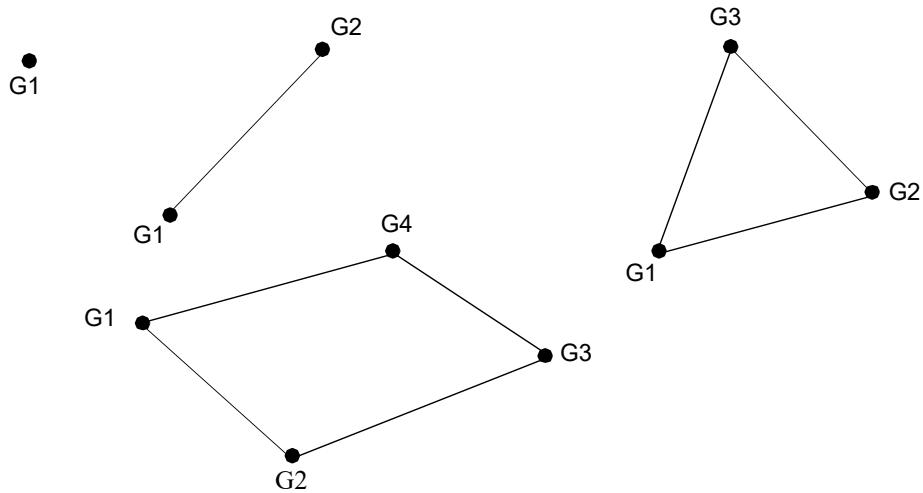


Figure 9-5 Four Types of CAABSF Elements

CACINF3**Acoustic Conjugate Infinite Element Base Connection**

Defines an acoustic conjugate infinite element with triangular base.

Format:

1	2	3	4	5	6	7	8	9	10
CACINF3	EID	PID	G1	G2	G3				

Example:

CACINF3	111	10	1004	1008	1011				
---------	-----	----	------	------	------	--	--	--	--

Descriptor	Meaning
EID	Element Identification Number. (0 < Integer < 100,000,000)
PID	Property Identification Number of a PACINF entry. (Integer > 0)
Gi	Grid Point Identification Numbers of Element Base Connection Points. (Integer > 0)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. The element must be connected to acoustic grid points.
3. The element normal is defined by the right-hand rule. If the normal does not point into the exterior domain, the element orientation will be changed automatically, and an information message will be written to the .f06 file.

CACINF4

Acoustic Conjugate Infinite Element Base Connection

Defines an acoustic conjugate infinite element with quadrilateral base.

Format:

1	2	3	4	5	6	7	8	9	10
CACINF4	EID	PID	G1	G2	G3	G4			

Example:

CACINF4	275	10	1027	1032	1056	1021			
---------	-----	----	------	------	------	------	--	--	--

Descriptor	Meaning
EID	Element Identification Number. (0 < Integer < 100,000,000)
PID	Property Identification Number of a PACINF entry. (Integer > 0)
Gi	Grid Point Identification Numbers of Element Base Connection Points. (Integer > 0)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. The element must be connected to acoustic grid points.
3. The element normal is defined by the right-hand rule. If the normal does not point into the exterior domain, the element orientation will be changed automatically, and an information message will be written to the .f06 file.

CAERO1

Aerodynamic Panel Element Connection

Defines an aerodynamic macro element (panel) in terms of two leading edge locations and side chords. This is used for Doublet-Lattice theory for subsonic aerodynamics and the ZONA51 theory for supersonic aerodynamics.

Format:

1	2	3	4	5	6	7	8	9	10
CAERO1	EID	PID	CP	NSPAN	NCHORD	LSPAN	LCHORD	IGID	
	X1	Y1	Z1	X12	X4	Y4	Z4	X43	

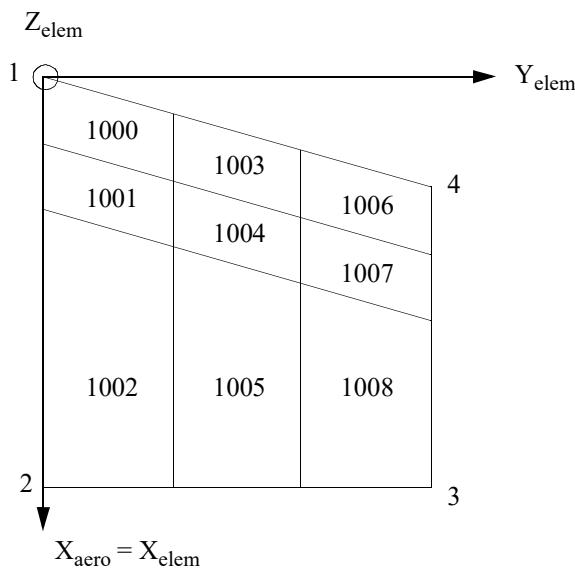
Example:

CAERO1	1000	1		3			2	1	
	0.0	0.0	0.0	1.0	0.2	1.0	0.0	0.8	

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PAERO1 entry; used to specify associated bodies. Required even if there are no associated bodies. (Integer > 0)
CP	Coordinate system for locating points 1 and 4. (Integer ≥ 0 ; Default = 0)
NSPAN	Number of spanwise boxes; if a positive value is given NSPAN, equal divisions are assumed; if zero or blank, a list of division points is given at LSPAN, field 7. (Integer ≥ 0)
NCHORD	Number of chordwise boxes; if a positive value is given NCHORD, equal divisions are assumed; if zero or blank, a list of division points is given at LCHORD, field 8. (Integer ≥ 0)
LSPAN	ID of an AEFACT entry containing a list of division points for spanwise boxes. Used only if NSPAN, field 5 is zero or blank. (Integer > 0)
LCHORD	ID of an AEFACT data entry containing a list of division points for chordwise boxes. Used only if NCHORD, field 6 is zero or blank. (Integer > 0)
IGID	Interference group identification; aerodynamic elements with different IGIDs are uncoupled. (Integer > 0)
X1, Y1, Z1 X4, Y4, Z4	Location of points 1 and 4, in coordinate system CP. (Real) }
X12, X43	Edge chord lengths in aerodynamic coordinate system. (Real ≥ 0.0 , but not both zero.)

Remarks:

1. The boxes and corner point nodes are numbered sequentially, beginning with EID. The user should be careful to ensure that all box and corner point node numbers are unique. There can be overlapping IDs between the structural and aerodynamic model, but MSC Patran will not then be able to display any results. Also, non-unique corner IDs are allowed, but results cannot be visualized in MSC Patran.
2. The number of division points is one greater than the number of boxes. Thus, if NSPAN=3, the division points are 0.0, 0.333, 0.667, 1.000. If the user supplies division points, the first and last points need not be 0. and 1. (In which case the corners of the panel would not be at the reference points.)
3. A triangular element is formed if X12 or X43=0.0
4. The element coordinate system is right-handed as shown in [Figure 9-6](#).
5. The continuation is required.
6. It is recommended that NCHORD or LCHORD be chosen so that the typical box chord length Δx satisfies the condition $\Delta x < 0.08 V/f$ (recent studies indicate that .02 V/f is needed to get converged stability derivatives) where V is the minimum velocity and f , in hertz, is the maximum frequency to be analyzed (see the [MSC Nastran Aeroelastic Analysis User's Guide](#)).

**Figure 9-6** Element Coordinate System for Aerodynamic Panel

7. This entry can be used for two different aerodynamic theories: Doublet-Lattice for subsonic and ZONA51 for supersonic. The proper theory is selected based on the specification of Mach number on the MKAEROi or TRIM entry.

CAERO2**Aerodynamic Body Connection**

Defines aerodynamic slender body and interference elements for Doublet-Lattice aerodynamics.

Format:

1	2	3	4	5	6	7	8	9	10
CAERO2	EID	PID	CP	NSB	NINT	LSB	LINT	IGID	
	X1	Y1	Z1	X12					

Example:

CAERO2	1500	2	100		4	99		1	
	-1.0	100.	-30.	175.					

Descriptor	Meaning
EID	Element identification number. ($0 < \text{Integer} < 100,000,000$)
PID	Property identification number of a PAERO2 entry. ($\text{Integer} > 0$)
CP	Coordinate system for locating point 1. ($\text{Integer} \geq 0$; Default = 0)
NSB	Number of slender body elements. If NSB > 0, then NSB equal divisions are assumed; if zero or blank, specify a list of divisions in LSB. ($\text{Integer} \geq 0$)
NINT	Number of interference elements. If NINT > 0, then NINT equal divisions are assumed; if zero or blank, specify a list of divisions in LINT. ($\text{Integer} \geq 0$)
LSB	ID of an AEFACT Bulk Data entry for slender body division points; used only if NSB is zero or blank. ($\text{Integer} \geq 0$)
LINT	ID of an AEFACT data entry containing a list of division points for interference elements; used only if NINT is zero or blank. ($\text{Integer} \geq 0$)
IGID	Interference group identification. Aerodynamic elements with different IGIDs are uncoupled. ($\text{Integer} \geq 0$)
X1, Y1, Z1	Location of point 1 in coordinate system CP. (Real)
X12	Length of body in the x-direction of the aerodynamic coordinate system. (Real > 0.0)

Remarks:

1. Point 1 is the leading point of the body.
2. All CAERO1 (panels) and CAERO2 (bodies) in the same group (IGID) will have aerodynamic interaction.
3. At least one interference element is required for the aerodynamic body specified by this entry.

4. The beams and connection points are numbered sequentially beginning with EID. The user should be careful to ensure that all aero elements and connection point IDs are unique. Overlapping IDs between structure and aerodynamic models are allowed, but will prevent results visualization in Patran.

Old rules regarding numbering among Z, ZY, Y bodies and CAERO1 no longer apply: arbitrary ordering is allowed.

5. At least two slender body elements are required for each aerodynamic body.
6. Interference elements are only intended for use with panels.
7. Determining the size of the j-set (i.e., the number of aerodynamic elements) is essential to input D1JE and D2JE matrices. Use the following expressions for locating the proper row in the two matrices:

$$\begin{aligned} J = & \text{Number of boxes} + \text{Number of I-elements, z} \\ & + 2 * (\text{Number of I-elements, zy}) \\ & + \text{Number of I-elements, y} \\ & + \text{Number of S-elements, z} \\ & + 2 * (\text{Number of S-elements, zy}) \\ & + \text{Number of S-elements, y} \end{aligned}$$

where I-elements denote interference and S-elements denote slender body.

CAERO3

Aerodynamic Panel Element Configuration

Defines the aerodynamic edges of a Mach Box lifting surface. If no cranks are present, this entry defines the aerodynamic Mach Box lifting surface.

Format:

1	2	3	4	5	6	7	8	9	10
CAERO3	EID	PID	CP	LISTW	LISTC1	LISTC2			
	X1	Y1	Z1	X12	X4	Y4	Z4	X43	

Example:

CAERO3	2000	2001	0	22	33				
	1.0	0.0	0.0	100.	17.	130.	0.	100.	

Descriptor	Meaning
EID	Element identification number. ($0 < \text{Integer} < 100,000,000$)
PID	Property identification number of a PAERO3 entry. ($\text{Integer} > 0$)
CP	Coordinate system for locating points 1 and 4. ($\text{Integer} \geq 0$; Default = 0)
LISTW	Identification number of an AEFACT entry that lists (x,y) pairs for structural interpolation of the wing. ($\text{Integer} > 0$)
LISTC1, LISTC2	Identification number of AEFACT entries that list (x,y) pairs for control surfaces, if they exist. ($\text{Integer} \geq 0$)
X1, Y1, Z1 X4, Y4, Z4	Location of points 1 and 4 in coordinate system CP. (Real)
X12, X43	Edge chord lengths in the aerodynamic coordinate system. ($\text{Real} \geq 0$, $X12 \neq 0$)

Remarks:

1. EID must be unique with respect to all other element identification numbers.
2. The (x,y) pairs on LISTW, LISTC1 and LISTC2 AEFACT entries are in the aero element coordinate system (see [Figure 9-7](#)). The (x,y) pairs define a set of aerodynamic grid points that are independent of Mach number and are selected by the user to be representative of the planform and motions of interest. The (x,y) pairs must be sufficient in number and distribution such that: the surface spline provides an accurate interpolation between them and the Mach Box centers that are variously located on the planform as a function of Mach number (a complete description of the Mach Box Method is given in the [MSC Nastran Aeroelastic Analysis User's Guide](#)).

3. The (x,y) pairs are numbered sequentially, beginning with EID for LISTW, then LISTC1, and finally for LISTC2. On SPLINEi entries, the box numbers (BOX1 and BOX2 on SPLINE1, ID1 and ID2 on SPLINE2, and UKID on SPLINE3) refer to the (x,y) pair sequence number appropriate for the surface (primary, or first or second control) being splined.
4. If cranks and/or control surfaces exist, their locations are given on the PAERO3 entry.
5. The numbering system and coordinate system are shown below:

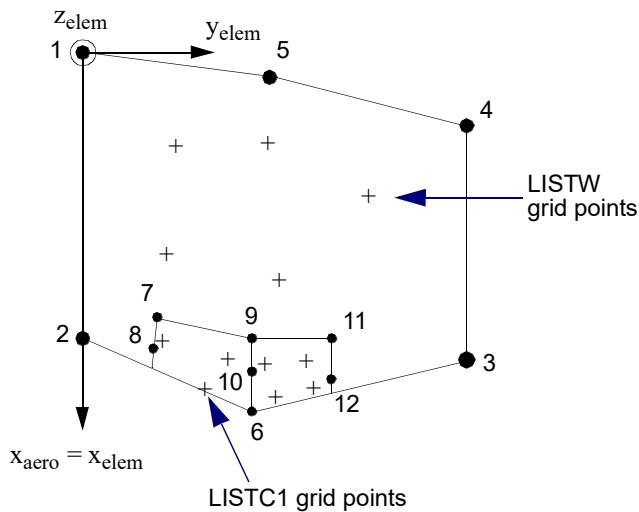


Figure 9-7 CAERO3 Element Configuration

Planform Corners		Control
1	Leading edge, inboard	7 Hinge line, inboard
2	Trailing edge, inboard	8 On inboard edge (usually at trailing edge)
3	Trailing edge, outboard	9 Hinge line, outboard
4	Leading edge, outboard	10 On outboard edge (usually at trailing edge)
Crank		Control (if two)
5	Leading edge	9 Hinge line, inboard
6	Trailing edge	10 On inboard edge (usually at trailing edge)
		11 Hinge line, outboard
		12 On outboard edge (usually at trailing edge)

6. The CAERO3 entry is only supported in SOL 145, SOL 200 with ANALYSIS=FLUT and in SOL 146 with mechanical loads. Gust response in SOL 146 is not supported for the CAERO3.

CAERO4

Aerodynamic Macro-Strip Element Connection

Defines an aerodynamic macro element for Strip theory.

Format:

1	2	3	4	5	6	7	8	9	10
CAERO4	EID	PID	CP	NSPAN	LSPAN				
	X1	Y1	Z1	X12	X4	Y4	Z4	X43	

Example:

CAERO4	6000	6001	100		315				
	0.0	0.0	0.0	1.0	0.2	1.0	0.0	0.8	

Descriptor	Meaning
EID	Element identification number. ($0 < \text{Integer} < 100,000,000$)
PID	Property identification number of a PAERO4 entry. ($\text{Integer} > 0$)
CP	Coordinate system for locating points 1 and 4. ($\text{Integer} \geq 0$; Default = 0)
NSPAN	Number of strips; if a positive value is given, NSPAN equal strips are assumed. If zero or blank, LSPAN must be specified. ($\text{Integer} \geq 0$)
LSPAN	ID of an AEFACT entry containing a list of division points for strips. Used only if NSPAN is zero or blank. ($\text{Integer} > 0$)
X1, Y1, Z1 X4, Y4, Z4	Location of points 1 and 4 in coordinate system CP. (Real) }
X12, X43	Edge chord lengths in aerodynamic coordinate system. (Real ≥ 0.0 , and not both zero.)

Remarks:

1. The strips are numbered sequentially, beginning with EID. The user must ensure that all strip numbers are unique and greater than structural grid, scalar, and extra point IDs.
2. The number of division points is one greater than the number of boxes. Thus, if NSPAN = 3, the division points are 0.0, 0.333, 0.667, and 1.000. If the user supplies division points, the first and last points need not be 0.0 and 1.0 (in which case the corners of the panel would not be at the reference points).
3. A triangular element is formed if X12 or X43 = 0.

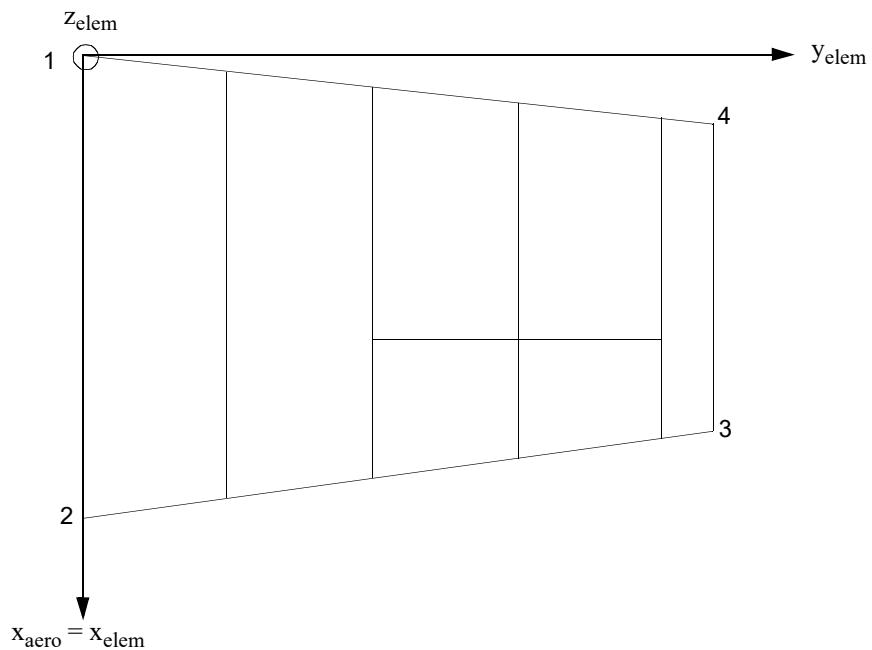


Figure 9-8 CAERO4 Element Connection

4. The CAERO4 entry is only supported in SOL 145, SOL 200 with ANALYSIS=FLUT and in SOL 146 with mechanical loads. Gust response in SOL 146 is not supported for the CAERO4.

CAERO5

Aerodynamic Panel Element Configuration

Defines an aerodynamic macro element for Piston theory.

Format:

1	2	3	4	5	6	7	8	9	10
CAERO5	EID	PID	CP	NSPAN	LSPAN	NTHRY	NTHICK		
	X1	Y1	Z1	X12	X4	Y4	Z4	X43	

Example:

CAERO5	6000	6001	100		315	0	0		
	0.0	0.0	0.0	1.0	0.2	1.0	0.	0.8	

Descriptor	Meaning	
EID	Element identification number. (0 < Integer < 100,000,000)	
PID	Property identification number of a PAERO5 entry. (Integer > 0)	
CP	Coordinate system for locating points 1 and 4. (Integer ≥ 0 ; Default = 0)	
NSPAN	Number of strips. (Integer. If a positive value is given, equal strips are assumed. If zero or blank, then LSPAN must be specified.)	
LSPAN	ID of an AEFACT entry containing a list of division points for strips. Used only if NSPAN is zero or blank. (Integer > 0)	
NTHRY	Parameter to select Piston or van Dyke's theory. (Integer = 0, 1, or 2; Default = 0)	
	Blank or 0	Piston theory is used to compute \bar{C}_1 and \bar{C}_2
	1	van Dyke's theory is used to compute \bar{C}_1 and \bar{C}_2 with no sweep correction ($\sec\Lambda = 1.0$).
	2	van Dyke's theory is used to compute \bar{C}_1 and \bar{C}_2 with a sweep correction based on the actual Λ .
NTHICK	Parameter to select thickness integrals input. (Integer ≥ 0 ; Default = 0)	
	Blank or 0	Thickness integrals are computed internally.
	> 0	Thickness integrals are input directly and are the ID number of an AEFACT entry that lists the I and/or J integrals.
X1,Y1, Z1 X4,Y4, Z4	{}	Location of points 1 and 4 in coordinate system CP. (Real)
X12, X43		Edge chord lengths in aerodynamic coordinate system. (Real ≥ 0 ; X12 and X43 cannot both be zero.)

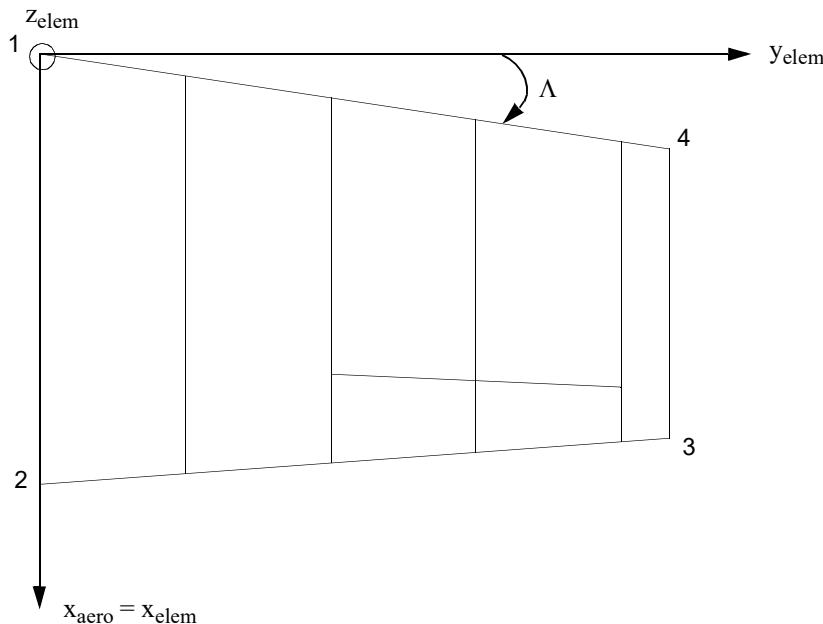


Figure 9-9 CAERO5 Element Configuration

Remarks:

1. The strips are numbered sequentially, beginning with EID. The user must ensure that all strip numbers are unique and different from structural grid IDs.
2. The number of division points is one greater than the number of boxes. Thus, if NSPAN=3, the division points are 0.0, 0.333, 0.667, 1.000. If the user supplies division points, the first and last points need not be 0.0 and 1.0 (in which case the corners of the panel would not be at the reference points).
3. A triangular element is formed if $X12$ or $X43 = 0.0$.

4.
$$\bar{C}_1 = m / (m^2 - \sec^2 \Lambda)^{1/2}$$
- $$\bar{C}_2 = [m^4(\gamma + 1) - 4 \sec^2 \Lambda (m^2 - \sec^2 \Lambda)] / [4(m^2 - \sec^2 \Lambda)^2]$$

where:

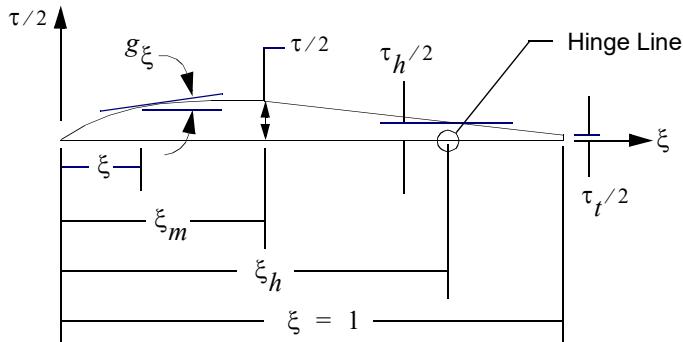
m = Mach number

γ = Specific heat ratio

Λ = Leading edge sweep angle

- When $\sec \Lambda = 0.0$, Piston theory coefficients are obtained (NTHRY = 1)
 When $\sec \Lambda = 1.0$, van Dyke's coefficients are obtained (NTHRY = blank or 0)
 When $\sec \Lambda \neq 0.0$ or $\neq 1.0$, sweep corrections are included (NTHRY = 2)

5. I and J thickness integral definitions:



$$g_\xi \equiv \frac{dg}{d\xi} = \text{slope of airfoil semithickness}$$

$$I_1 = \int_0^1 g_\xi d\xi \quad J_1 = \int_{\xi_h}^1 g_\xi d\xi$$

$$I_2 = \int_0^1 \xi g_\xi d\xi \quad J_2 = \int_{\xi_h}^1 \xi g_\xi d\xi$$

$$I_3 = \int_0^1 \xi^2 g_\xi d\xi \quad J_3 = \int_{\xi_h}^1 \xi^2 g_\xi d\xi$$

$$I_4 = \int_0^1 g_\xi^2 d\xi \quad J_4 = \int_{\xi_h}^1 g_\xi^2 d\xi$$

$$I_5 = \int_0^1 \xi g_\xi^2 d\xi \quad J_5 = \int_{\xi_h}^1 \xi g_\xi^2 d\xi$$

$$I_6 = \int_0^1 \xi^2 g_\xi^2 d\xi \quad J_6 = \int_{\xi_h}^1 \xi^2 g_\xi^2 d\xi$$

Figure 9-10 CAERO5 / and J Thickness Integral Definitions

6. The CAERO5 entry is only supported in SOL 145, SOL 200 with ANALYSIS=FLUT and in SOL 146 with mechanical loads. Gust response in SOL 146 is not supported for the CAERO5.

CAMPBLL**Campbell Diagram Parameters**

Specifies the parameters for Campbell Diagram generation and mode tracking analysis.

Formats:

1	2	3	4	5	6	7	8	9	10
CAMPBLL	CID	VPARM	DDVALID	TYPE					
	MODTRK	CORU	SWITR	NUMMOD	PRTCOR				

Examples:

CAMPBLL	15	SPEED	22	RPM					
	1	0.75	1		1				

Descriptor	Meaning
CID	Identification number of entry (Integer > 0; Required).
VPARM	Variable parameter, allowable entry is: 'SPEED'. 'SPEED', reference rotor speed will be varied (rotordynamic option only).
DDVALID	Identification number of DDVAL entry that specifies the values for the variable parameter (Integer > 0; Required).
TYPE	Allowable entries are: 'FREQ' and 'RPM'.
MODTRK	Perform mode tracking using either numerical approach (default) or eigenvector based approach or both. (Default = 0). See remark 4.
CORU	Threshold for mode correlation.
SWITR	Option to use updated mode for mode tracking in case of failure. See remark 7.
NUMMODE	Number of modes to track (Should be less than 0.5*Neig, where Neig is the number of eigenvalues extracted in EIGC). See remark 4.
PRTCOR	Option to print correlation matrix at each rotor speed (Default = 0). See remark 9.

Remarks:

1. CAMPBLL option is supported for both SOL 107 and SOL 110 analysis.
2. CAMPBLL option is not supported in SOL 200 and SOL 400 for ANALYSIS = DCEIG/MCEIG.
3. Only the modes with positive imaginary part are used for mode tracking (to avoid repeated eigenvalues.)
4. Following methods are available in MSC Nastran for mode tracking:
 - a. Numerical Mode tracking based on second derivative.
Obtain second derivative for the variation of eigenvalue with rotor speed and determine the eigenfrequency with minimum slope.

- b. Mode tracking based on orthogonality of left and right eigenvectors.

This feature is available for analysis in fixed reference frame only.

MODTRK = 0: Perform numerical mode tracking only. (Default)

MODTRK = 1: Perform BOTH numerical and eigenvector based mode tracking.

MODTRK = 2: Perform eigenvector based mode tracking only.

5. The variables specified in the second row are relevant for eigenvector-based mode tracking only.
6. Ideally, for tracked pair of modes, the value of Normalized Cross Complex Orthogonality (NC2O) parameter is very close to 1. However, for complex models involving large stator component and dynamic reduction, this value can be significantly lower. Thus, the default value for CORU is set to 0.7. Based on the model complexity and NC2O matrix obtained for two rotor speeds, user may change this value.
7. In case of mode tracking failure at a particular rotor speed, user has the option to continue mode tracking for rest of the rotors speed by:
 - SWITR = 1: Updating modes corresponding to rotor speed with failed mode tracking, or
 - SWITR = 0: Using the modes from last rotor speed with successful mode tracking. (Default)
 In case the Campbell diagram includes multiple mode switching and there are new modes entering the analysis, use of SWITR=1 is recommended.
8. Only the complex eigenvalues with positive imaginary part are considered for mode tracking analysis. As a result, the number of eigenvalues available for mode tracking may not be same at all the rotor speeds considered in the analysis. This may lead to mode tracking failure. One way to avoid this issue is to pick NUMMODE to be less than half of the number of complex eigenvalues extracted using EIGC bulk data entry. Thus, the analysis uses only the first NUMMODE frequencies with positive imaginary part for mode tracking at each rotor speed.
9. When PRTCOR parameter is turned to 1, the code prints NC2O matrix corresponding to each rotor speed in the F06 file. This information can be very useful in identifying reasons for mode tracking failure.
10. For both approaches, damping ratio is also tracked while tracking eigenfrequencies. The results produced from mode tracking can be extracted in OP4 using following assign statements:

```
$ Extract results for numerical mode tracking
ASSIGN OUTPUT4='freq_nmt.op4',UNIT=71,FORM=FORMATTED,DELETE $ MODE FREQS
ASSIGN OUTPUT4='rot_nmt.op4',UNIT=72,FORM=FORMATTED,DELETE $ ROTOR SPEED
ASSIGN OUTPUT4='dmp_nmt.op4',UNIT=73,FORM=FORMATTED,DELETE $ DAMPING RATIO
$ Extract results for eigenvector based mode tracking
ASSIGN OUTPUT4='freq_evmt.op4',UNIT=81,FORM=FORMATTED,DELETE $ MODE FREQS
ASSIGN OUTPUT4='rot_evmt.op4',UNIT=82,FORM=FORMATTED,DELETE $ ROTOR SPEED
ASSIGN OUTPUT4='dmp_evmt.op4',UNIT=83,FORM=FORMATTED,DELETE $ DAMPING RATIO
```

These OP4 files can be imported in MSC PATRAN to produce Campbell Diagram.

11. DMP>1 is not supported for SOL 110 for generating Campbell diagram when ACMS option is not used.

CAXIFi**Fluid Element Connections**

Defines an axisymmetric fluid element that connects i = 2, 3, or 4 fluid points.

Formats:

1	2	3	4	5	6	7	8	9	10
CAXIF2	EID	IDF1	IDF2			RHO	B		
CAXIF3	EID	IDF1	IDF2	IDF3		RHO	B		
CAXIF4	EID	IDF1	IDF2	IDF3	IDF4	RHO	B		

Examples:

CAXIF2	11	23	25			0.25E-3			
CAXIF3	105	31	32	33		6.47E-3			
CAXIF4	524	421	425	424	422	0.5E-3	2.5+3		

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
IDFi	Identification numbers of connected GRIDF points. (Integer > 0)
RHO	Fluid density in mass units. (Real > 0.0 or blank)
B	Fluid bulk modulus. (Real \geq 0.0 or blank)

Remarks:

1. CAXIFi is allowed only if an AXSLOT entry is also present.
2. The element identification number (EID) must be unique with respect to all other fluid or structural elements.
3. If RHO or B is blank, then the corresponding RHOD and BD fields must be specified on the AXSLOT entry.
4. Plot elements are generated for these elements. Because each plot element connects two points, one is generated for the CAXIF2 element, three are generated for the CAXIF3 element, and four plot elements are generated for the CAXIF4 element. In the last case the elements connect the pairs of points (1-2), (2-3), (3-4), and (4-1).
5. If B = 0.0, the fluid is incompressible.

CAXISYM

Axisymmetric Line Elements Connection - SOL 400

Defines two or three node axisymmetric thick shell elements.

Formats:

1	2	3	4	5	6	7	8	9	10
CAXISYM	EID	PID	G1	G2	G3	NOFF			

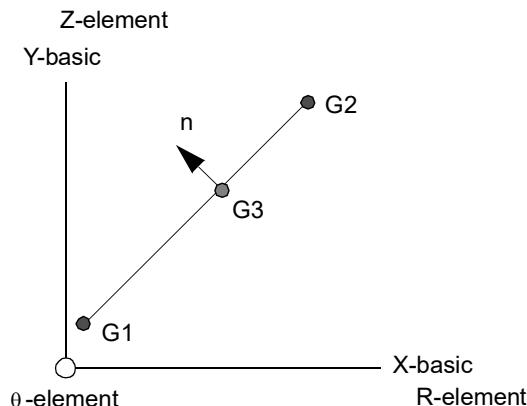
Example:

CAXISYM	22	98	8	16					
---------	----	----	---	----	--	--	--	--	--

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of PAXISYM entry. (Integers > 0)
G1, G2	Identification numbers at the two end grid points. Required data. (Unique; Integers > 0)
G3	Identification number of the one grid in between G1 and G2. See Remark 2. (Unique Integer ≥ 0 or blank). If G3=0; this is a straight two node element.
NOFF	Offset from the surface of the grid points to the element reference plane. (Real)

Remarks:

1. Element identification numbers should be unique with respect to all other identification numbers.
2. It is recommended that G3 be located within the middle third of the element.
3. The element must lie in the x-y plane of the basic system and is oriented as shown below.



CBAR**Simple Beam Element Connection**

Defines a simple beam element.

Format:

1	2	3	4	5	6	7	8	9	10
CBAR	EID	PID	GA	GB	X1	X2	X3	OFFT	
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	

Example:

CBAR	2	39	7	3	0.6	18.	26.	GOG	
		513							

Alternate Format and Example:

CBAR	EID	PID	GA	GB	G0			OFFT	
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	
CBAR	2	39	7	6	105			GOG	
		513							

Descriptor	Meaning
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PBAR, PBARL or PBRSECT entry. (Integer > 0 or blank*; Default = EID unless BAROR entry has nonzero entry in field 3.)
GA, GB	Grid point identification numbers of connection points. (Integer > 0; GA ≠ GB)
X1, X2, X3	Components of orientation vector \vec{v} , from GA, in the displacement coordinate system at GA (Default), or in the basic coordinate system. See Remark 8. (Real)
G0	Alternate method to supply the orientation vector \vec{v} using grid point G0. The direction of \vec{v} is from GA to G0. \vec{v} is then translated to End A. (Integer > 0; G0 ≠ GA or GB)
OFFT	Offset vector interpretation flag. (character or blank) See Remark 8.

Descriptor	Meaning
PA, PB	Pin flags for bar ends A and B, respectively. Used to remove connections between the grid point and selected degrees-of-freedom of the bar. The degrees-of-freedom are defined in the element's coordinate system (see Figure 9-11). The bar must have stiffness associated with the PA and PB degrees-of-freedom to be released by the pin flags. For example, if PA = 4 is specified, the PBAR entry must have a value for J, the torsional stiffness. (Up to 5 of the unique Integers 1 through 6 anywhere in the field with no embedded blanks; Integer > 0.) Pin flags combined with offsets are not allowed for SOL 600. Pin flags are not allowed in SOL 700.
W1A, W2A, W3A W1B, W2B, W3B	Components of offset vectors \vec{w}_a and \vec{w}_b , respectively (see Figure 9-11) in displacement coordinate systems (or in element system depending upon the content of the OFFT field), at points GA and GB, respectively. See Remark 7 and 8 . (Real; Default = 0.0) Offsets are not allowed in SOL 700.

*See the BAROR entry for default options for field 3 and fields 6 through 9.

Remarks:

- Element identification numbers should be unique with respect to all other element identification numbers.
- [Figure 9-11](#) and [Figure 9-12](#) define bar element geometry with and without offsets:

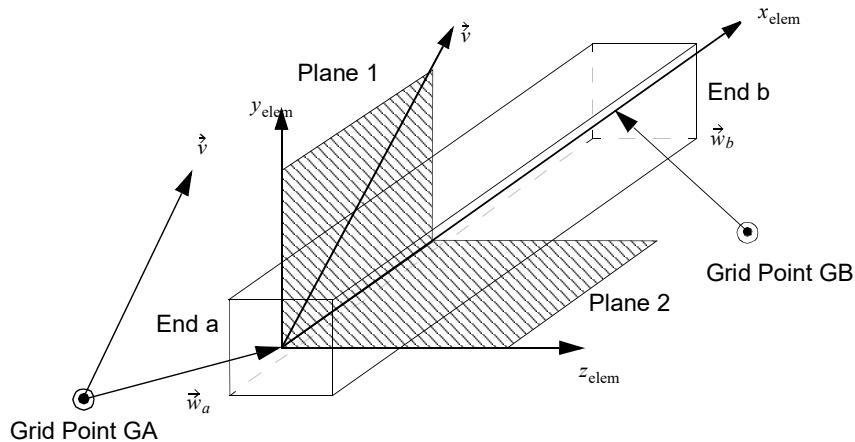


Figure 9-11 CBAR Element Geometry with Offsets

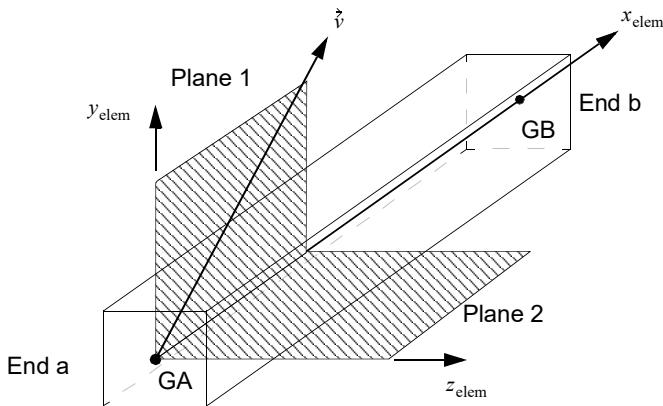


Figure 9-12 CBAR Element Geometry without Offsets

3. [Figure 9-13](#) and [Figure 9-14](#) define the elemental force and moment sign convention.

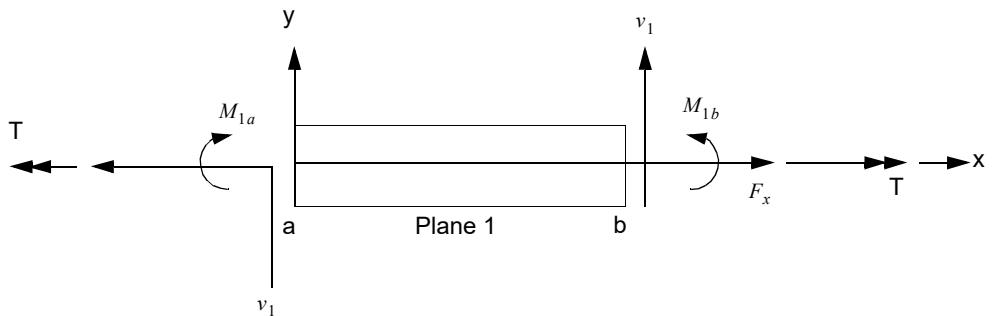


Figure 9-13 CBAR Element Internal Forces and Moments (x-y Plane)

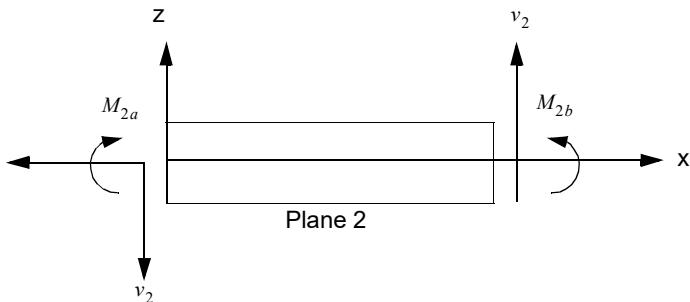


Figure 9-14 CBAR Element Internal Forces and Moments (x-z Plane)

4. The continuation may be omitted if there are no pin flags or offsets.
5. For the case where field 9 is blank and not provided by the BAROR entry, if an integer is specified in field 6, then G0 is used; if field 6 is blank or real, then X1, X2, X3 is used.

6. See [Grid Point and Coordinate System Definition](#) (p. 34) in the *MSC Nastran Reference Guide* for a definition of coordinate system terminology.
7. Offset vectors are treated internally like rigid elements. For solution sequences that require differential stiffness such as SOL103 (with preloading), SOL105, and SOL400, the user is required to use MDLPRM, OFFDEF, LROFF.
 - Two methods are available for the computation of offsets: original and enhanced. The default method is the original method. The enhanced method is requested by the Bulk Data entry MDLPRM,OFFDEF,LROFF. For options of offsets, please refer to the Bulk Data entry MDLPRM,OFFDEF,option.
 - The length of offset vectors is not affected by thermal loads. But the thermal load changes due to location changes by offsets are correctly computed if the enhanced method is used.
 - BAR elements with offsets will give correct buckling results if the enhanced method is used.
 - Masses are correctly transformed for offset effects if MDLPRM, OFFDEF, LROFF is used. In SOL103, if the case control ADAMSMNF FLEXBODY=YES or AVLEXB EXBBODY=YES is present, then MDLPRM, OFFDEF, LROFF must be replaced by MDLPRM, OFFDEF, NOMASS.
 - In nonlinear solution sequences, such as SOL 106 or SOL 400, BAR is treated as linear element. If geometric nonlinear effects are required, please use Bulk Data entry MDLPRM,BRTOBM,1 to convert BAR to BEAM.
8. OFFT is a character string code that describes how the offset and orientation vector components are to be interpreted. By default (string input is GGG or blank), the offset vectors are measured in the displacement coordinate systems at grid points A and B and the orientation vector is measured in the displacement coordinate system of grid point A. At user option, the offset vectors can be measured in an offset coordinate system relative to grid points A and B, and the orientation vector can be measured in the basic system as indicated in the following table:

String	Orientation Vector	End A Offset	End B Offset
GGG	Global	Global	Global
BGG	Basic	Global	Global
GGO	Global	Global	Offset
BGO	Basic	Global	Offset
GOG	Global	Offset	Global
BOG	Basic	Offset	Global
GOO	Global	Offset	Offset
BOO	Basic	Offset	Offset

Any attempt to specify invalid combinations results in a bulk data entry input error message. For example, a value of OOO (indicating offset and orientation vectors are specified in an offset reference system) results in a fatal error since the orientation vector cannot be specified in an offset system. The offset system x-axis is defined from GA to GB. The orientation vector \hat{v} and the offset system x-axis are then used to define the z and y axes of the offset system. A vector is formed from a cross product of a vector going from Grid A to Grid B and the orientation vector to create the offset coordinate z-direction. To obtain a nonzero cross product the orientation vector must not be parallel to both vectors from Grid A to Grid B for the offset coordinate system and End A and End B for the element coordinate system. (Note: The character "O" in the table replaces the obsolete character "E".)

9. For SOL 600, the BIT field is ignored unless param,MAROFSET is 1 or 2. An extra flag
10. For SOL 700, the BIT field is ignored and a warning is issued.
11. For RC network solver in thermal analysis, the G0, OFFT, PA, PB, W1A, W2A, W3A, W1B, W2B and W3B are ignored.

CBARAO**Auxiliary Output Points Along Bar Element Axis (CBAR Entry)**

Defines a series of points along the axis of a bar element (CBAR entry) for stress and force recovery output. This entry is applicable in static and normal modes analysis only.

Format:

1	2	3	4	5	6	7	8	9	10
CBARAO	EID	SCALE	X1	X2	X3	X4	X5	X6	

Example:

CBARAO	1065	FR	0.2	0.4	0.6	0.8			
--------	------	----	-----	-----	-----	-----	--	--	--

Alternate Format and Example:

CBARAO	EID	SCALE	NPTS	X1	DELTAX				
CBARAO	1065	FR	4	0.2	0.2				

Descriptor	Meaning
EID	Element identification of a CBAR entry. (0 < Integer < 100,000,000)
SCALE	Defines scale of Xi values. (Character = "LE" or "FR")
Xi	Series of locations along element axis for stress and force data recovery. (Real > 0.0)
DELTAX	Incremental distance along element axis. (Real)
NPTS	Number of stress recovery points, not including the end points. (Integer > 0)

Remarks:

1. This entry defines intermediate locations on the axis of selected CBAR elements for additional data recovery. The values of Xi are actual distances along the length if SCALE = "LE". If SCALE = "FR", the values of Xi are ratios of actual distances to the bar length. A PLOAD1 Bulk Data entry for the CBAR element in question must be present to obtain intermediate data recovery.
2. When the alternate format is used, a series of locations $Xi = X[i-1] + DELTAX$, $i = 1, 2, \dots, NPTS$ is generated.
3. If a CBARAO or PLOAD1 entry is specified and stress and/or force output is requested, then the stresses and/or forces will be calculated at each location Xi and output as a separate line. The force and stress values at the end points of the beam will always be output. This output format will be used for all beam and bar elements.
4. Intermediate loads on the element defined by the PLOAD1 entry will be accounted for in the calculation of element stresses and forces. If no PLOAD1 entry is defined for the element, the shear forces are constant, the moments are linear, and it is not necessary that the user define additional points.

5. For each bar element, either the basic format or the alternate format, but not both, may be used. A maximum of six internal points can be specified with the basic form. The end points must not be listed because data will be generated for them, as explained in Remark 3. If more than six unequally spaced internal points are desired, it is advisable to subdivide the bar into two or more elements.

CBEAM**Beam Element Connection**

Defines a beam element.

Format:

1	2	3	4	5	6	7	8	9	10
CBEAM	EID	PID	GA	GB	X1	X2	X3	OFFT/BIT	
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	
	SA	SB							

Example:

CBEAM	2	39	7	13	8.2	6.1	-5.6	GOG	
		513		3.0					
	8	5							

Alternate Format and Example:

CBEAM	EID	PID	GA	GB	G0			OFFT/BIT	
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	
	SA	SB							
CBEAM	2	39	7	13	105			GOG	
		513							

Descriptor	Meaning
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of PBEAM, PBCOMP, PBEAML or PBMSECT entry. (Integer > 0; Default = EID)*
GA, GB	Grid point identification numbers of connection points. (Integer > 0; GA ≠ GB)
X1, X2, X3	Components of orientation vector \vec{v} , from GA, in the displacement coordinate system at GA (Default), or in the basic coordinate system. See Remark 9. (Real)
G0	Alternate method to supply the orientation vector \vec{v} using grid point G0. Direction of \vec{v} is from GA to G0. \vec{v} is then transferred to End A. (Integer > 0; G0 ≠ GA or GB)
OFFT	Offset vector interpretation flag. See Remark 9. (Character or blank)
BIT	Built-in twist of the cross-sectional axes about the beam axis at end B relative to end A. For beam p-elements only. (Real; Default = 0.0)

Descriptor	Meaning
PA, PB	Pin flags for beam ends A and B, respectively; used to remove connections between the grid point and selected degrees-of-freedom of the beam. The degrees-of-freedom are defined in the element's coordinate system and the pin flags are applied at the offset ends of the beam (see Figure 9-15). The beam must have stiffness associated with the PA and PB degrees-of-freedom to be released by the pin flags. For example, if PA = 4, the PBEAM entry must have a nonzero value for J, the torsional stiffness. (Up to five of the unique Integers 1 through 6 with no embedded blanks.) Pin flags are not allowed for beam p-elements. Pin flags combined with offsets are not allowed for SOL 600. Pin flags are not presently allowed in SOL 700. Also, Pin flags should not be used in nonlinear analysis when there is large displacement.
W1A, W2A, W3A W1B, W2B, W3B	Components of offset vectors from the grid points to the end points of the axis of the shear center. See Remarks 7 , 8 , and 9 . (Real; Default = 0.0)
SA, SB	Scalar or grid point identification numbers for the ends A and B, respectively. The degrees-of-freedom at these points are the warping variables $d\theta/dx$. SA and SB cannot be specified for beam p-elements. (Integers ≥ 0 or blank)

*See the BEAMOR entry for default options for field 3 and fields 6 through 9.

Remarks:

- Element identification numbers should be unique with respect to all other element identification numbers.
- For an additional explanation of the beam element, see the [Beam Element \(CBEAM\)](#) (p. 55) in the *MSC Nastran Reference Guide*. [Figure 9-15](#) defines beam element geometry:

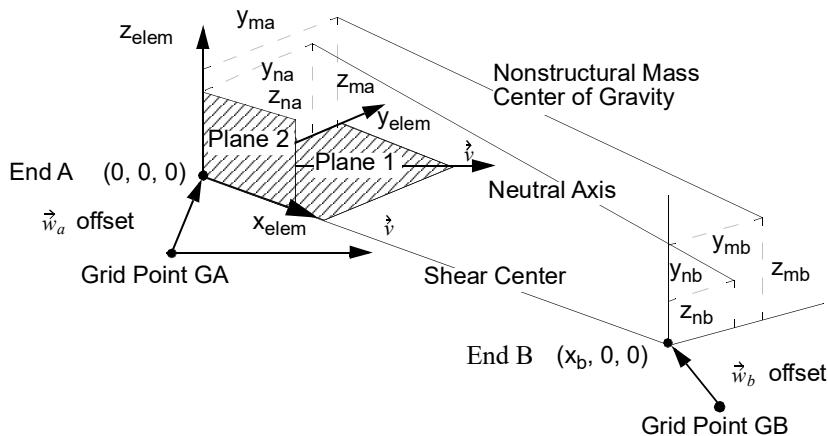


Figure 9-15 CBEAM Element Geometry System (Non p-adaptive)

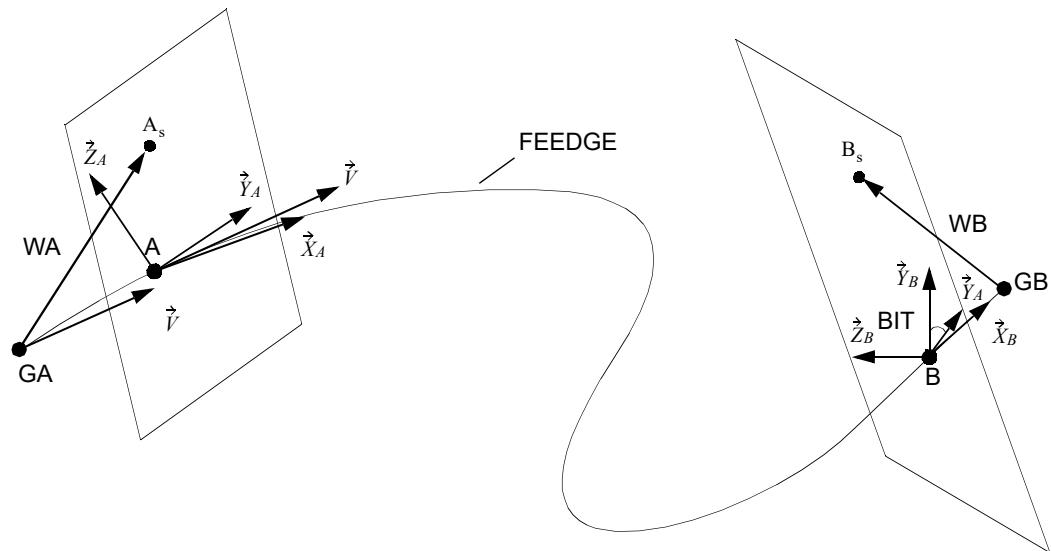


Figure 9-16 CBEAM Element Geometry System (p-adaptive)

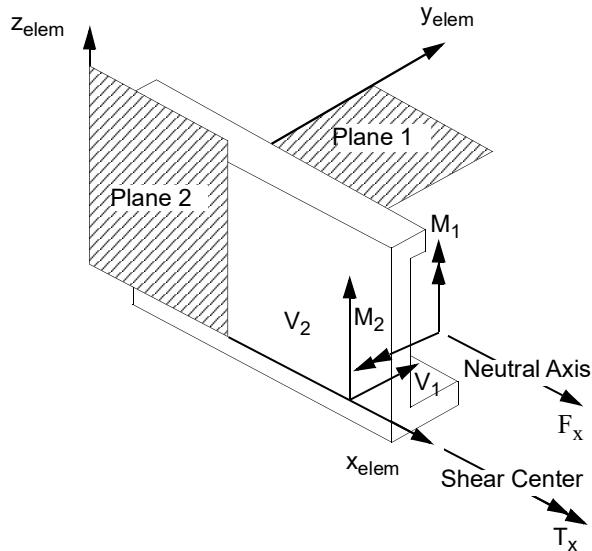


Figure 9-17 CBEAM Internal Element Forces and Moments

3. If field 6 is an integer, then G0 is used. If field 6 is blank or real, then X1, X2, X3 is used.
4. G0 cannot be located at GA or GB.
5. The rules for the continuations entries are:
 - Both continuations may be omitted if there are no pin flags, offsets, or warping variables.

- If the second continuation is used, then the first continuation must be included, even if all fields are blank.
 - If the second continuation is omitted, torsional stiffness due to warping of the cross section will not be considered.
6. If warping is allowed (SA and $SB > 0$), then SA and SB must be defined with SPOINT or GRID entries. If GRID entries are used, the warping degree-of-freedom is attached to the first (T1) component. In addition, SPOINT ID is recommended for SA and SB if GROUNDCHECK is requested in case control.
7. Offset vectors are treated internally like rigid elements. Two methods are available for the computation of offsets: original and enhanced. The default method is the original method. The enhanced method is requested by the Bulk Data entry MDLPRM, OFFDEF, LROFF. For options of offsets, please refer to the Bulk Data entry MDLPRM, OFFDEF, option.
- For solution sequences that require differential stiffness such as SOL103 (with preloading), SOL105, and SOL400, the user is required to use MDLPRM, OFFDEF, LROFF. In SOL103, if the case control ADAMSMNF FLEXBODY=YES or AVLEXB EXBBODY=YES is present, then MDLPRM, OFFDEF, LROFF must be replaced by MDLPRM, OFFDEF, NOMASS.
- For SOLs 106, 129, 153, and 159 the differential stiffness for offset vectors will give incorrect results with PARAM, LGDISP, 1.
8. If the CBEAM element is referenced by a PSET or PVAL entry, then a p-version formulation is used and the element can have curved edges.
- By default, the edge of the element is considered straight unless the element is a p-element and the edge is associated to curved geometry with a FEEDGE entry.
 - If a curved edge of a p-element is shared by an h-element without midside nodes, the geometry of the edge is ignored and considered to be straight. Edges with midside nodes cannot be shared by p-elements.
 - For the beam p-element, components of the offset vectors parallel to the beam axis (FEEDGE) will be ignored.
 - For the beam p-element, offset vectors can only be specified in the displacement coordinate systems at the grid points.
9. If the element is a p-version element, BIT in field 9 contains the value of the built-in-twist measured in radians. Otherwise, OFFT in field 9 is a character string code that describes how the offset and orientation vector components are to be interpreted. By default (string input is GGG or blank), the offset vectors are measured in the displacement coordinate systems at grid points A and B and the orientation vector is measured in the displacement coordinate system of grid point A. At user option, the offset vectors can be measured in an offset system relative to grid points A and B, and the orientation vector can be measured in the basic system as indicated in the following table:

String	Orientation Vector	End A Offset	End B Offset
GGG	Global	Global	Global
BGG	Basic	Global	Global
GGO	Global	Global	Offset
BGO	Basic	Global	Offset
GOG	Global	Offset	Global
BOG	Basic	Offset	Global
GOO	Global	Offset	Offset
BOO	Basic	Offset	Offset

Any attempt to specify invalid combinations results in a bulk data entry input error message. For example, a value of OOO (indicating offset and orientation vectors are specified in an offset reference system) results in a fatal error since the orientation vector cannot be specified in an offset system. The offset system x-axis is defined from GA to GB. The orientation vector \hat{v} and the offset system x-axis are then used to define the z and y axes of the offset system. A vector is formed from a cross product of a vector going from Grid A to Grid B and the orientation vector to create the offset coordinate z-direction. To obtain a nonzero cross product the orientation vector must not be parallel to both vectors from Grid A to Grid B for the offset coordinate system and End A and End B for the element coordinate system. (Note: The character "O" in the table replaces the obsolete character "E".)

10. For RC network solver in thermal analysis, the X1, X2, X3, OFFT, PA, PB, W1A, W2A, W3A, W1B, W2B, W3B, SA and SB are ignored.

CBEAM3**Three-Node Beam Element Connection**

Defines a three-node beam element.

Format:

1	2	3	4	5	6	7	8	9	10
CBEAM3	EID	PID	GA	GB	GC	X1	X2	X3	
	W1A	W2A	W3A	W1B	W2B	W3B	W1C	W2C	
	W3C	TWA	TWB	TWC	SA	SB	SC		

Example:

CBEAM3	101	2	201	332	1000	1.0	3.5	-2.0	
		3.0		3.0	2.2	-1.0			
	2.5	10.	15.	20.0	206	301	312		

Alternate Format and Example:

CBEAM3	EID	PID	GA	GB	GC	G0			
	W1A	W2A	W3A	W1B	W2B	W3B	W1C	W2C	
	W3C	TWA	TWB	TWC	SA	SB	SC		

CBEAM3	101	2	201	332	1000	105			
		3.0			2.2	1.0			
	2.5	10.	15.	20.0	206	301	312		

Descriptor	Meaning
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of PBEAM3 or PBMSECT entries. See Remark 8. (Integer > 0; Required)
GA, GB, GC	Grid point identification numbers of connection points. GA and GB are grid point identification numbers at the two ends of the beam element while GC is the one at the grid point in between. (Integer > 0 or blank; GA, GB and GC must be distinct from each other. See Remark 6.)
X1, X2, X3	Components of orientation vector \vec{v} , from GA, in the displacement coordinate system at GA. (Real)
G0	Alternate method to supply the orientation vector \vec{v} using grid point G0. The direction of \vec{v} is from GA to G0. \vec{v} is then transferred to End A. (Integer > 0; G0 ≠ GA or GB or GC)

Descriptor	Meaning
WiA, WiB, WiC	Components of offsets vectors, measured in the displacement coordinate systems at grid points A, B, and C, from the grid points to the points on the axis of shear center. See Remark 9. (Real; Default = 0.0)
TWA, TWB, TWC	Pretwist angles in degrees at A, B, and C, respectively. (Real; Default = 0.0)
SA, SB, SC	Scalar or grid point identification numbers for A, B, and C, respectively. The degrees of freedom at these points are warping variables. (Integer > 0 or blank)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. If field 7 is an integer, then G0 is used. If field 7 is blank or real, then X1, X2, X3 are used.
3. G0 cannot be located at GA or GB or GC.
4. If warping effect is included in the analysis (SA, SB and SC > 0), then SA, SB, and SC must be defined with either SPOINT or GRID entries. If GRID entries are used, the warping degree of freedom is attached to the first (T1) component. In addition, SPOINT ID is recommended for SA, SB and SC if GROUNDCHECK is requested in case control.
5. BEAMOR cannot be used to set up default options for field 3 and fields 6 through 8 for CBEAM3 entries.
6. If GC is left blank, then the element degenerates to a formulation similar to the two-node straight beam element. The two-node straight beam formulation is not the CBEAM formulation and will result in overly stiff results with warping and shear center offset. This is because the CBEAM3 is a Variational Asymptotic Beam Section that generalizes Timoshenko beam theory and requires the mid node for accuracy.
7. This entry is not available in SOL 600.
8. For CBEAM3 referencing PBMSECT, the CORE/LAYER keywords are required or a Fatal message is issued. For this composite case, it is recommended that CBEAM3 has 3 nodes and 3 warping DOFs.

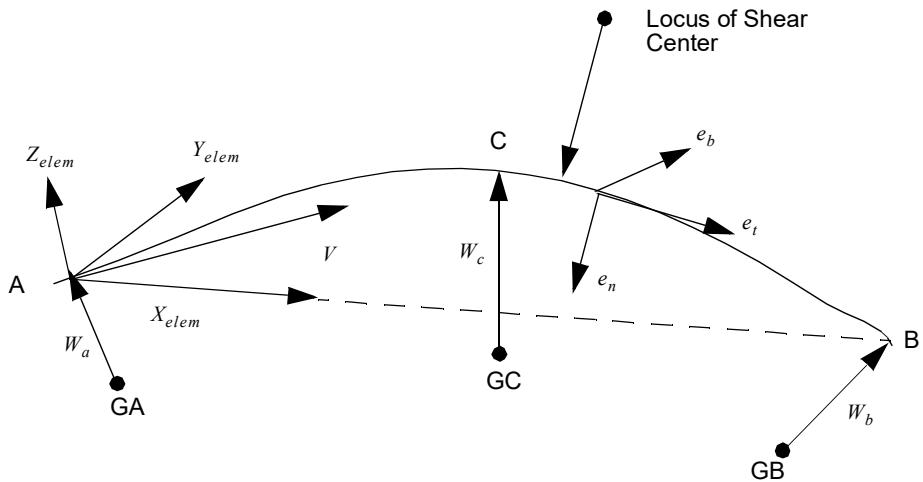


Figure 9-18 CBEAM3 Element Geometry System

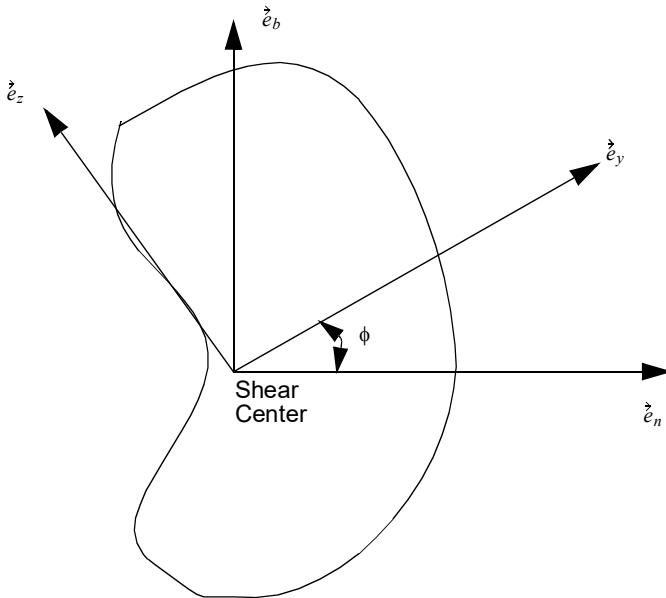


Figure 9-19 Local Coordinate System on Beam Cross-Section

- Offset vectors are treated internally like rigid elements. Two methods are available for the computation of offsets: original and enhanced. The default method is the original method. The enhanced method is requested by the Bulk Data entry MDLPRM, OFFDEF, LROFF. For options of offsets, please refer to the Bulk Data entry MDLPRM, OFFDEF, option.

For solution sequences that require differential stiffness such as SOL103 (with preloading), SOL105, and SOL400, the user is required to use MDLPRM, OFFDEF, LROFF. In SOL103, if the case control ADAMSMNF FLEXBODY=YES or AVLEXB EXBBODY=YES is present, then MDLPRM, OFFDEF, LROFF must be replaced by **MDLPRM, OFFDEF, NOMASS**.

CBEND**Curved Beam or Pipe Element Connection**

Defines a curved beam, curved pipe, or elbow element.

Format:

1	2	3	4	5	6	7	8	9	10
CBEND	EID	PID	GA	GB	X1	X2	X3	GEOM	

Example:

CBEND	32	39	17	19	6.2	5.1	-1.2	3	
-------	----	----	----	----	-----	-----	------	---	--

Alternate Format and Example:

CBEND	EID	PID	GA	GB	G0			GEOM	
CBEND	32	39	17	19	106			3	

Descriptor	Meaning
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PBEND entry. (Integer > 0; Default = EID)
GA, GB	Grid point identification numbers of connection points. (Integer > 0; GA ≠ GB)
X1, X2, X3	Components of orientation vector \vec{v} , from GA, in the displacement coordinate system at GA. (Real)
G0	Alternate method to supply the orientation vector \vec{v} using grid point G0. Direction of \vec{v} is from GA to G0. \vec{v} is then translated to End A. (Integer > 0; G0 ≠ GA or GB)
GEOM	Flag to select specification of the bend element. See Remark 3. (1 ≤ Integer ≤ 4; No Default)

Remarks:

- Element identification numbers must be unique with respect to all other element identification numbers.

2. For an additional explanation of the CBEND element, see the PBEND entry description. [Figure 9-20](#) and [Figure 9-21](#) define the element coordinate system and internal forces and moments.

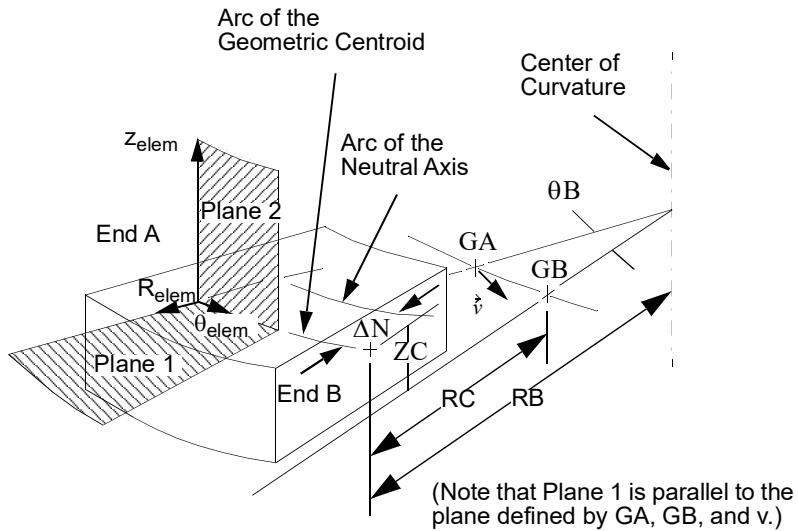


Figure 9-20 CBEND Element Coordinate System

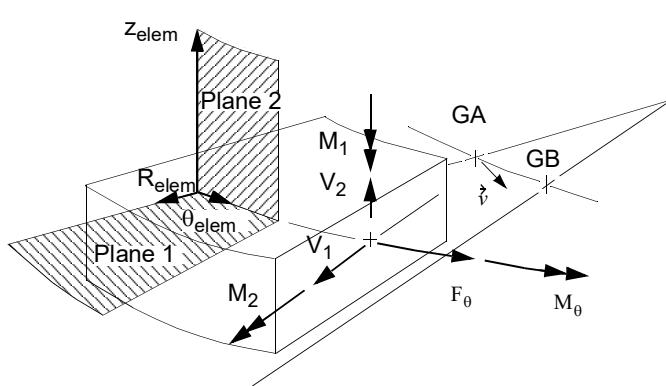
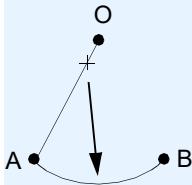
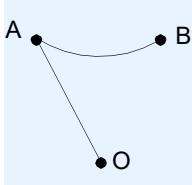
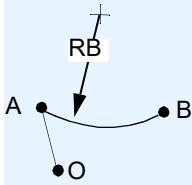
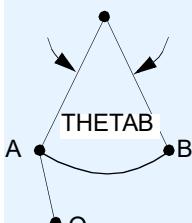


Figure 9-21 CBEND Element Internal Forces and Moments

3. The options for element connection to GA, GB using GEOM are the following.

Table 2 GEOM Options

Configuration	GEOM	Description
	1	The center of curvature lies on the line AO (or its extension) or vector \vec{v} .
	2	The tangent of centroid arc at end A is parallel to line AO or vector \vec{v} . Point O (or vector \vec{v}) and the arc \overline{AB} must be on the same side of the chord \overline{AB} .
	3	The bend radius (RB) is specified on the PBEND entry: Points A, B, and O (or vector \vec{v}) define a plane parallel or coincident with the plane of the element arc. Point O (or vector \vec{v}) lies on the opposite side of line AB from the center of curvature.
	4	THETAB is specified on the PBEND entry. Points A, B, and O (or vector \vec{v}) define a plane parallel or coincident with the plane of the element arc. Point O (or vector \vec{v}) lies on the opposite side of line AB from the center of curvature.

4. For RC network solver in thermal analysis, the X1, X2, X3 and GEOM are ignored.

CBUSH

Generalized Spring-and-Damper Connection

Defines a generalized spring-and-damper structural element that may be nonlinear or frequency dependent.

Format:

1	2	3	4	5	6	7	8	9	10
CBUSH	EID	PID	GA	GB	GO/X1	X2	X3	CID	
	S	OCID	S1	S2	S3				

Example 1: Noncoincident grid points.

CBUSH	39	6	1	100	75				
-------	----	---	---	-----	----	--	--	--	--

Example 2: GB not specified.

CBUSH	39	6	1					0	
-------	----	---	---	--	--	--	--	---	--

Example 3: Coincident grid points (GA ≠ GB).

CBUSH	39	6	1					6	
-------	----	---	---	--	--	--	--	---	--

Example 4: Noncoincident grid points with fields 6 through 9 blank and a spring-damper offset.

CBUSH	39	6	1	600					
	0.25	10	0.	10.	10.				

Descriptor	Meaning
EID	Element identification number. ($0 < \text{Integer} < 100,000,000$)
PID	Property identification number of a PBUSH entry. ($\text{Integer} > 0$; Default = EID)
GA, GB	Grid point identification number of connection points. See Remark 6. ($GA > 0$, $GB \geq 0$ or blank)
Xi	Components of orientation vector \vec{v} , from GA, in the displacement coordinate system at GA. (Real)
GO	Alternate method to supply vector \vec{v} using grid point GO. Direction of \vec{v} is from GA to GO. \vec{v} is then transferred to End A. See Remark 3. (Integer > 0)
CID	Element coordinate system identification. A 0 value means the basic coordinate system will be used. If CID is blank, then the element coordinate system is determined from GO or Xi. See Figure 9-22 and Remark 3. (Integer ≥ 0 or blank)
S	Location of spring damper. See Figure 9-21. (Real; Default = 0.5)

Descriptor	Meaning
OCID	Coordinate system identification of spring-damper offset. See Remark 9. (Integer ≥ -1 ; Default = -1, which means the offset point lies on the line between GA and GB according to Figure 9-22)
S1, S2, S3	Components of spring-damper offset in the OCID coordinate system if OCID ≥ 0 . See Figure 9-23 and Remark 9. (Real)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. [Figure 9-22](#) shows the bush element geometry.
3. CID ≥ 0 overrides GO and Xi. Then the element x-axis is along T1, the element y-axis is along T2, and the element z-axis is along T3 of the CID coordinate system. If the CID refers to a cylindrical coordinate system or a spherical coordinate system, then grid GA is used to locate the system. If for cylindrical or spherical coordinate, GA falls on the z-axis used to define them, it is recommended that another CID be selected to define the element x-axis. Nastran does not convect user specified coordinate systems, thus with this option, in nonlinear analysis the element stiffness direction does not change with deformation.
4. For noncoincident grids (GA \neq GB), when GO or (X1, X2, X3) is given and no CID is specified, the line AB is the element x-axis and the orientation vector \vec{v} lies in the x-y plane (similar to the CBEAM element).
5. For noncoincident grids (GA \neq GB), if neither GO or (X1, X2, X3) is specified and no CID is specified, then the line AB is the element x-axis. This option is valid only when K1 (or B1) or K4 (or B4) or both on the PBUSH entry are specified (but K2, K3, K5, K6 or B2, B3, B5, B6 are not specified). If K2, K3, K5, or K6 (or B2, B3, B5, or B6) are specified, a fatal message will be issued.
6. If the distance between GA and GB is less than .0001, or if GB is blank, then CID must be specified. GB blank implies that B is a grounded terminal, a grounded terminal is a point with a displacement that is constrained to zero.
7. If PID references a PBUSHT entry, then the CBUSH element may only be defined in the residual structure and cannot be attached to any omitted degrees-of-freedom.
8. Element impedance output is computed in the CID coordinate system. The impedances in this system are uncoupled.
9. If OCID = -1 or blank (default) then S is used and S1, S2, S3 are ignored. If OCID ≥ 0 , then S is ignored and S1, S2, S3 are used.

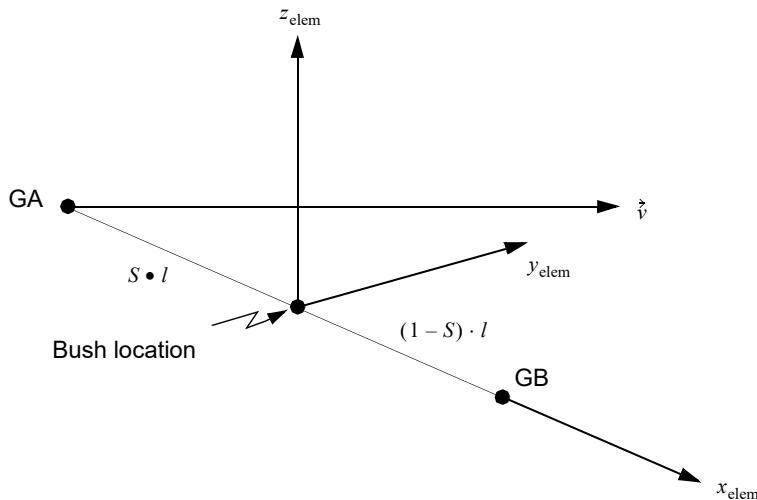
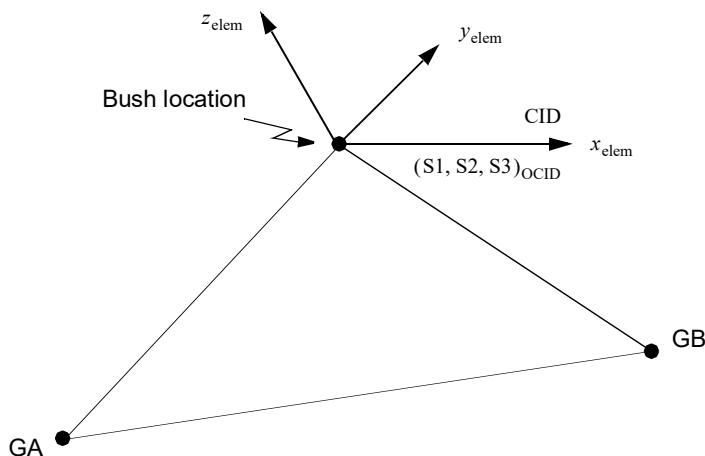


Figure 9-22 CBUSH Element



The material stiffness and damping properties of the elastomer are located at $(S1, S2, S3)$.

Figure 9-23 Definition of Offset S1, S2, S3

10. When $\text{CID} \geq 0$, the element x-axis is set as in Remark 3. This means that the element force is always computed as $K_e \cdot (U_B - U_A)$; if $U_A > U_B$, a compressive force will result. This is unlike the GO or Xi options, where relative positive elongation in tension and relative negative elongation is compression.
11. The CBUSH element is designed to satisfy rigid body equilibrium requirements. For noncoincident grids, internal rigid links connect the bush location to the grid locations. This results in coupling between translational and rotational degrees-of-freedom at the grids even when no rotational springs or dampers are specified on the PBUSH.

12. For SOL 600, if G0, X1, X2, X3, CID or OCID are entered, a Severe Warning will be issued and Marc will not run. SOL 600 translates the spring and damping terms in global coordinates of GA and GB and will use K1 to K6 and B1 to B6 whether or not GA and GB are coincident or not. S, S1, S2, S3 are ignored. CID is ignored unless it is zero, in which case K1 and B1 are along the axis of GA to GB and K2-K6 and B2-B6 are ignored if entered.
13. CBUSH elements are not supported in thermal analysis.

CBUSH1D

Rod Type Spring-and-Damper Connection

Defines the connectivity of a one-dimensional spring and viscous damper element.

Format:

1	2	3	4	5	6	7	8	9	10
CBUSH1D	EID	PID	GA	GB	CID				

Example:

CBUSH1D	35	102	108	112					
---------	----	-----	-----	-----	--	--	--	--	--

Descriptor	Meaning	Default Values
EID	Element identification number. ($0 < \text{Integer} < 100,000,000$)	Required
PID	Property identification number of a PBUSH1D entry. ($\text{Integer} > 0$)	EID
GA	Grid point id of first grid.	Required
GB	Grid point id of second grid	blank
CID	Coordinate system id. ($\text{Integer} \geq 0$)	blank

Remarks:

1. For noncoincident grids $GA \neq GB$ and if CID is blank, the line GA to GB is the element axis. In geometric nonlinear analysis, the element axis (line GA to GB) follows the deformation of grids GA and GB. See [Figure 9-24](#).
2. If $CID \geq 0$ is specified, the x-axis of the CID coordinate system is the element axis. In geometric nonlinear analysis, the element axis (x-axis of CID) remains fixed.
3. If GA and GB are coincident or if GB is blank, then $CID \geq 0$ must be specified and the element axis is the x-axis of CID.

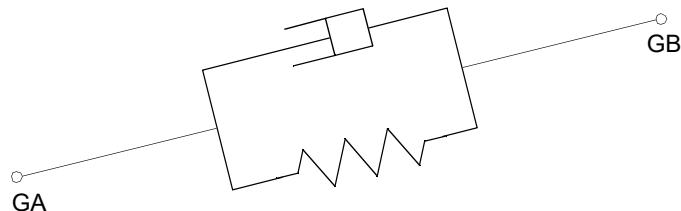


Figure 9-24 Spring and Damper Element

CBUSH2D**2-D Linear-Nonlinear Connection**

Defines the connectivity of a two-dimensional Linear-Nonlinear element.

Format:

1	2	3	4	5	6	7	8	9	10
CBUSH2D	EID	PID	GA	GB	CID	PLANE			

Example:

CBUSH2D	100	101	1001	2001	0	XY			
---------	-----	-----	------	------	---	----	--	--	--

Descriptor	Meaning
EID	Element identification number. ($0 < \text{Integer} < 100,000,000$; Required)
PID	Property identification number of a PBUSH2D. ($\text{Integer} > 0$; Required)
GA	Inner grid. ($\text{Integer} > 0$; Required)
GB	Outer grid. ($\text{Integer} > 0$; Required)
CID	Coordinate system used to define 2-D plane. ($\text{Integer} \geq 0$; Default = 0)
PLANE	Orientation plane in CID: XY, YZ, ZX, see Remark 1. (Character; Default = 'XY')

Remarks:

1. The XY, YZ, and ZX planes are relative to the displacements coordinates of GA and GB. The planes correspond to directions 1 and 2. GA and GB should be coincident grids with parallel displacement coordinate systems. The coordinate systems are not checked. Wrong answers will be produced if this rule is not followed.
2. The behavior is different if there is no rotor in the model and a CBUSH2D is used. When this happens, for most cases, if there is no rotor the nominal stiffness values from the PBUSH2D will be used, with no frequency-dependence. However, if an ELEMUDS is used, the CBUSH2D will be ignored if there is no rotor in the model.

CCONEAX

Axisymmetric Shell Element Connection

Defines a conical shell element.

Format:

1	2	3	4	5	6	7	8	9	10
CCONEAX	EID	PID	RA	RB					

Example:

CCONEAX	1	2	3	4					
---------	---	---	---	---	--	--	--	--	--

Descriptor	Meaning
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PCONEAX entry. (Integer > 0; Default = EID)
RA	Identification number of a RINGAX entry. (Integer > 0; RA ≠ RB)
RB	Identification number of a RINGAX entry. (Integer > 0; RA ≠ RB)

Remarks:

1. This element has limited capabilities. See the *MSC Nastran Reference Guide*, Section 5.3.3.
2. This entry is allowed only if an AXIC entry is also present.
3. In order to reference this entry on a SET Case Control command, the ID must be modified by

$$\text{ID}_n = \text{ID} \cdot 1000 + n$$

where n is the harmonic number plus one and ID_n is the value specified on the SET entry.

CDAMP1

Scalar Damper Connection

Defines a scalar damper element.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP1	EID	PID	G1	C1	G2	C2			

Example:

CDAMP1	19	6	0		23	2			
--------	----	---	---	--	----	---	--	--	--

Descriptor	Meaning
EID	Unique element identification number. ($0 < \text{Integer} < 100,000,000$)
PID	Property identification number of a PDAMP property entry. ($\text{Integer} > 0$; Default = EID)
G1, G2	Geometric grid point identification number. ($\text{Integer} \geq 0$)
C1, C2	Component number. ($0 \leq \text{Integer} \leq 6$; blank or zero if scalar point.)

Remarks:

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2), must be distinct.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASI, CMASSI, CDAMPI\)](#) (p. 174) in the *MSC Nastran Reference Guide*.
5. When CDAMP1 is used in heat transfer analysis, it generates a lumped heat capacity.
6. A scalar point specified on this entry need not be defined on an SPOINT entry.
7. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.

CDAMP1D

Scalar Damper Connection for SOL 700 Only

Defines a scalar damper connection for use in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP1D	EID	PID	G1	C1	G2	C2			
	CORD	FOLLOW							

Example:

CDAMP1D	1001	101	55	1					
---------	------	-----	----	---	--	--	--	--	--

Descriptor	Meaning
EID	Unique element identification number. ($0 < \text{Integer} < 100,000,000$)
PID	Property identification number of a PDAMPn entry. ($\text{Integer} > 0$; Default = EID)
G1, G2	Geometric grid point identification number. ($\text{Integer} \geq 0$)
C1, C2	Component number. ($0 \leq \text{Integer} \leq 6$; 0 or up to six unique integers, 1 through 6 may be specified in the field with no embedded blanks. 0 applies to scalar points and 1 through 6 applies to grid points.)
CORD	Number of a coordinate system in which the degree-of-freedom (C1,C2) is defined. ($\text{Integer} \geq 0$)
FOLLOW	Method to update the direction vector in which the damper acts: FOLLOW=CORD: direction vector follows the motion of the coordinate system as specified under CORD.

Remark:

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2), must be distinct.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASI, CMASSI, CDAMPI\)](#) (p. 174) in the *MSC Nastran Reference Guide*.
5. When CDAMP1 is used in heat transfer analysis, it generates a lumped heat capacity.
6. A scalar point specified on this entry need not be defined on an SPOINT entry.
7. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry. Available in SOL 700 only.

CDAMP2

Scalar Damper Property and Connection

Defines a scalar damper element without reference to a material or property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP2	EID	B	G1	C1	G2	C2			

Example:

CDAMP2	16	2.98	32	1					
--------	----	------	----	---	--	--	--	--	--

Descriptor	Meaning
EID	Unique element identification number. ($0 < \text{Integer} < 100,000,000$)
B	Value of the scalar damper. (Real)
G1, G2	Geometric grid point identification number. ($\text{Integer} \geq 0$)
C1, C2	Component number. ($0 \leq \text{Integer} \leq 6$; blank or zero if scalar point.)

Remarks:

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2), must be distinct.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASI, CMASSi, CDAMPi\)](#) (p. 174) in the *MSC Nastran Reference Guide*.
5. When CDAMP2 is used in heat transfer analysis, it generates a lumped heat capacity.
6. A scalar point specified on this entry need not be defined on an SPOINT entry.
7. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.
8. RC network solver does not support CDAMP2 for thermal analysis.

CDAMP2D

Scalar Damper Connection for SOL 700 Only

Defines a scalar damper connection for use in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP2D	EID	B	G1	C1	G2	C2			
	CORD	FOLLOW							

Example:

CDAMP2D	1001	101	55	1					
---------	------	-----	----	---	--	--	--	--	--

Descriptor	Meaning
EID	Unique element identification number. ($0 < \text{Integer} < 100,000,000$)
B	Value of the scalar damper. (Real)
G1, G2	Geometric grid point identification number. ($\text{Integer} \geq 0$)
C1, C2	Component number. ($0 \leq \text{Integer} \leq 6$; 0 or up to six unique integers, 1 through 6 may be specified in the field with no embedded blanks. 0 applies to scalar points and 1 through 6 applies to grid points.)
CORD	Number of a coordinate system in which the degree-of-freedom (C1,C2) is defined. ($\text{Integer} \geq 0$)
FOLLOW	Method to update the direction vector in which the damper acts: FOLLOW=CORD: direction vector follows the motion of the coordinate system as specified under CORD.

Remark:

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2), must be distinct.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASI, CMASSI, CDAMPI\)](#) (p. 174) in the *MSC Nastran Reference Guide*.
5. When CDAMP2 is used in heat transfer analysis, it generates a lumped heat capacity.
6. A scalar point specified on this entry need not be defined on an SPOINT entry.
7. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry. Available in SOL 700 only.

CDAMP3

Scalar Damper Connection to Scalar Points Only

Defines a scalar damper element that is connected only to scalar points.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP3	EID	PID	S1	S2					

Example:

CDAMP3	16	978	24	36					
--------	----	-----	----	----	--	--	--	--	--

Descriptor	Meaning
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PDAMP entry. (Integer > 0; Default = EID)
S1, S2	Scalar point identification numbers. (Integer ≥ 0 ; S1 \neq S2)

Remarks:

1. S1 or S2 may be blank or zero, indicating a constrained coordinate.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one scalar damper element may be defined on a single entry.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASI, CMASSI, CDAMPI\)](#) in the *MSC Nastran Reference Guide*.
5. When CDAMP3 is used in heat transfer analysis, it generates a lumped heat capacity.
6. A scalar point specified on this entry need not be defined on an SPOINT entry.
7. RC network solver does not support CDAMP3 for thermal analysis.

CDAMP4**Scalar Damper Property and Connection to Scalar Points Only**

Defines a scalar damper element that connected only to scalar points and without reference to a material or property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP4	EID	B	S1	S2					

Example:

CDAMP4	16	-2.6	4	9					
--------	----	------	---	---	--	--	--	--	--

Descriptor	Meaning
EID	Unique element identification number. ($0 < \text{Integer} < 100,000,000$)
B	Scalar damper value. (Real)
S1, S2	Scalar point identification numbers. ($\text{Integer} \geq 0$; $S1 \neq S2$)

Remarks:

1. S1 or S2 may be blank or zero, indicating a constrained coordinate.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one scalar damper element may be defined on a single entry.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASI, CMASSI, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
5. If this entry is used in heat transfer analysis, it generates a lumped heat capacity.
6. A scalar point specified on this entry need not be defined on an SPOINT entry.
7. RC network solver does not support CDAMP4 for thermal analysis.

CDAMP5**Scalar Damper with Material Property**

Defines a damping element that refers to a material property entry and connection to grid or scalar points. This element is intended for heat transfer analysis only.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP5	EID	PID	G1	G2					

Example:

CDAMP5	1	4	10	20					
--------	---	---	----	----	--	--	--	--	--

Descriptor	Meaning
EID	Unique element identification number. ($0 < \text{Integer} < 100,000,000$)
PID	Identification number of a PDAMP5 property entry. ($\text{Integer} > 0$; Default = EID)
G1, G2	Grid or scalar point identification numbers. ($\text{Integer} \geq 0$ and $\text{G1} \neq \text{G2}$)

Remarks:

1. G1 or G2 may be blank or zero indicating a constraint.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. CDAMP5 generates a lumped heat capacity in heat transfer analysis.
4. A scalar point specified on CDAMP5 need not be defined on an SPOINT entry.
5. This entry is not supported in SOL 600.
6. RC network solver does not support CDAMP5 for thermal analysis.

CDUMi**Dummy Element Connection**

Defines a dummy element ($3 \leq i \leq 7$).

Format:

1	2	3	4	5	6	7	8	9	10
CDUMi	EID	PID	G1	G2	G3	G4	-etc.-		
	A1	A2	-etc.-						

Example:

CDUM3	114	108	2	5	6	8	11		
	2.4		3.E4	2		50			

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PDUMi entry. See Remark 2. (Integer > 0)
Gi	Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2 ... ≠ GN)
Ai	Additional fields. (Real or Integer)

Remarks:

1. The user must write the associated element subroutines for matrix generation, stress recovery, etc., and perform a link edit to replace the dummy routines. See the *MSC Nastran Programmer's Manual*.
2. If no property entry is required, PID may contain the material identification number.
3. Additional entries are defined in the user-written element routines.
4. The fields on this entry are required to be defined on the corresponding ADUMi entry. This entry requires a license for "USER MODIFIABLE Nastran". Other than the EID field, all field checking is the responsibility of the user supplied code.

CELAS1

Scalar Spring Connection

Defines a scalar spring element.

Format:

1	2	3	4	5	6	7	8	9	10
CELAS1	EID	PID	G1	C1	G2	C2			

Example:

CELAS1	2	6			8	1			
--------	---	---	--	--	---	---	--	--	--

Descriptor	Meaning
EID	Unique element identification number. ($0 < \text{Integer} < 100,000,000$)
PID	Property identification number of a PELAS entry. ($\text{Integer} > 0$; Default = EID)
G1, G2	Geometric grid point identification number. ($\text{Integer} \geq 0$)
C1, C2	Component number. ($0 \leq \text{Integer} \leq 6$; blank or zero if scalar point.)

Remarks:

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero. If only scalar points and/or ground are involved, it is more efficient to use the CELAS3 entry.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2) must be distinct.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASI, CMASSI, CDAMPI\)](#) (p. 174) in the *MSC Nastran Reference Guide*.
5. A scalar point specified on this entry need not be defined on an SPOINT entry.
6. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.
7. It is recommended that for most applications the grids connected by CELAS1 entries be coincident and the displacement coordinate systems of each grid be congruent. Connecting non-coincident grids and/or non-congruent coordinate systems can lead to models that do not pass strain energy checks (see the [GROUNDCHECK \(Case\)](#) Case Control command).

CELAS1D

Scalar Spring Connection for SOL 700 Only

Defines a scalar spring connection for use in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
CELAS1D	EID	PID	G1	C1	G2	C2			
	CORD	FOLLOW							

Example:

CELAS1D	1001	101	55	1	8	1			
---------	------	-----	----	---	---	---	--	--	--

Descriptor	Meaning
EID	Unique element identification number. ($0 < \text{Integer} < 100,000,000$)
PID	Property identification number of a PELAS entry. ($\text{Integer} > 0$; Default = EID)
G1, G2	Geometric grid point identification number. ($\text{Integer} \geq 0$)
C1, C2	Component number. ($0 \leq \text{Integer} \leq 6$; 0 or up to six unique, 1 through 6 may be specified in the field with no embedded blanks. 0 applies to scalar points and 1 through 6 applies to grid points.)
CORD	Number of a coordinate system in which the degree-of-freedom (C1,C2) is defined. ($\text{Integer} \geq 0$) (SOL 700 only)
FOLLOW	Method to update the direction vector in which the spring acts: For SOL 700: FOLLOW=CORD: Direction vector follows the motion of the coordinate system as specified under CORD.

Remarks:

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero. If only scalar points and/or ground are involved, it is more efficient to use the CELAS3 entry.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2) must be distinct.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASI, CMASSI, CDAMPI\)](#) in the *MSC Nastran Reference Guide*.
5. A scalar point specified on this entry need not be defined on an SPOINT entry.

6. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.
7. Available in SOL 700 only.
8. It is recommended that for most applications the grids connected by CELAS1D entries be coincident and the displacement coordinate systems of each grid be congruent. Connecting non-coincident grids and/or non-congruent coordinate systems can lead to models that do not pass strain energy checks (see the [GROUNDCHECK \(Case\)](#) Case Control command).

CELAS2**Scalar Spring Property and Connection**

Defines a scalar spring element without reference to a property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CELAS2	EID	K	G1	C1	G2	C2	GE	S	

Example:

CELAS2	28	6.2+3	32		19	4			
--------	----	-------	----	--	----	---	--	--	--

Descriptor	Meaning
EID	Unique element identification number. ($0 < \text{Integer} < 100,000,000$)
K	Stiffness of the scalar spring. (Real)
G1, G2	Geometric grid point or scalar identification number. ($\text{Integer} \geq 0$)
C1, C2	Component number. ($0 \leq \text{Integer} \leq 6$; 0 blank or zero if scalar point.)
GE	Damping coefficient. See Remarks 6. and 8. (Real)
S	Stress coefficient. (Real)

Remarks:

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero. If only scalar points and/or ground are involved, it is more efficient to use the CELAS4 entry.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2) must be distinct.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASI, CMASSI, CDAMPI\)](#) in the *MSC Nastran Reference Guide*.
5. A scalar point specified on this entry need not be defined on an SPOINT entry.
6. If PARAM,W4 is not specified, GE is ignored in transient analysis. See [Parameters, 783](#).
7. If Gi refers to a grid point then Ci refers to degrees-of-freedom in the displacement coordinate system specified by CD on the GRID entry.
8. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0 by 2.0.

CELAS2D**Scalar Spring Connection for SOL 700 Only**

Defines a scalar spring connection for use in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
CELAS2D	EID	K	G1	C1	G2	C2			
	CORD	FOLLOW							

Example:

CELASD2	1001	101	55	1	8	1			
---------	------	-----	----	---	---	---	--	--	--

Descriptor	Meaning
EID	Unique element identification number. ($0 < \text{Integer} < 100,000,000$)
K	Stiffness of the scalar spring. (Real)
G1, G2	Geometric grid point identification number. ($\text{Integer} \geq 0$)
C1, C2	Component number. ($0 \leq \text{Integer} \leq 6$; 0 or up to six unique, 1 through 6 may be specified in the field with no embedded blanks. 0 applies to scalar points and 1 through 6 applies to grid points.)
CORD	Number of a coordinate system in which the degree-of-freedom (C1,C2) is defined. ($\text{Integer} \geq 0$)
FOLLOW	Method to update the direction vector in which the spring acts: FOLLOW=CORD: Direction vector follows the motion of the coordinate system as specified under CORD.

Remarks:

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero. If only scalar points and/or ground are involved, it is more efficient to use the CELAS4 entry.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2) must be distinct.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASI, CMASSi, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
5. A scalar point specified on this entry need not be defined on an SPOINT entry.
6. If Gi refers to a grid point then Ci refers to degrees-of-freedom in the displacement coordinate system specified by CD on the GRID entry.

7. It is recommended that for most applications the grids connected by CELAS2D entries be coincident and the displacement coordinate systems of each grid be congruent. Connecting non-coincident grids and/or non-congruent coordinate systems can lead to models that do not pass strain energy checks (see the [GROUNDCHECK \(Case\)](#) Case Control command).

CELAS3**Scalar Spring Connection to Scalar Points Only**

Defines a scalar spring element that connects only to scalar points.

Format:

1	2	3	4	5	6	7	8	9	10
CELAS3	EID	PID	S1	S2					

Example:

CELAS3	19	2	14	15					
--------	----	---	----	----	--	--	--	--	--

Descriptor	Meaning
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PELAS entry. (Integer > 0; Default = EID)
S1, S2	Scalar point identification numbers. (Integer ≥ 0 ; S1 \neq S2)

Remarks:

1. S1 or S2 may be blank or zero, indicating a constrained coordinate.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one scalar spring element may be defined on a single entry.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASi, CMASSi, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
5. A scalar point specified on this entry need not be defined on an SPOINT entry.

CELAS4

Scalar Spring Property and Connection to Scalar Points Only

Defines a scalar spring element that is connected only to scalar points, without reference to a property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CELAS4	EID	K	S1	S2					

Example:

CELAS4	42	6.2-3	2						
--------	----	-------	---	--	--	--	--	--	--

Descriptor	Meaning
EID	Unique element identification number. (0 < Integer < 100,000,000)
K	Stiffness of the scalar spring. (Real)
S1, S2	Scalar point identification numbers. (Integer ≥ 0 ; S1 \neq S2)

Remarks:

1. S1 or S2, but not both, may be blank or zero indicating a constrained coordinate.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. A structural damping coefficient is not available with CELAS4. The value of g is assumed to be 0.0.
4. No stress coefficient is available with CELAS4.
5. Only one scalar spring element may be defined on a single entry.
6. For a discussion of the scalar elements, see [Scalar Elements \(CELASI, CMASSI, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
7. A scalar point specified on this entry need not be defined on an SPOINT entry.

CFAST**A Shell Patch Fastener Connection**

Defines a fastener with material orientation connecting two surface patches. Large displacement and large rotational effects are supported when in SOL 600 and SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
CFAST	EID	PID	TYPE	IDA	IDB	GS	GA	GB	
	XS	YS	ZS						

Example using PROP:

CFAST	3	20	PROP	21	24	206			
-------	---	----	------	----	----	-----	--	--	--

Example using ELEM:

CFAST	7	70	ELEM	27	74	707			
-------	---	----	------	----	----	-----	--	--	--

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PFAST entry. (Integer > 0; Default = EID)
TYPE	Specifies the surface patch definition: (Character) If TYPE = 'PROP', the surface patch connectivity between patch A and patch B is defined with two PSHELL (or PCOMP) properties with property ids given by IDA and IDB. See Remark 1. and Figure 9-25 . If TYPE = 'ELEM', the surface patch connectivity between patch A and patch B is defined with two shell element ids given by IDA and IDB. See Remark 1. and Figure 9-25 .
IDA, IDB	Property id (for PROP option) or Element id (for ELEM option) defining patches A and B. IDA ≠ IDB (Integer > 0)
GS	Grid point defining the location of the fastener. See Remark 2. (Integer > 0 or blank)
GA, GB	Grid ids of piercing points on patches A and B. See Remark 2. (Integer > 0 or blank)
XS, YS, ZS	Location of the fastener in basic. Required if neither GS nor GA is defined. See Remark 2. (Real or blank)

Remarks:

1. The CFAST defines a flexible connection between two surface patches. The number of unique physical grids connected depends on the location for the piercing points GA and GB and the size of the diameter D (see [PFAST, 2829](#)).

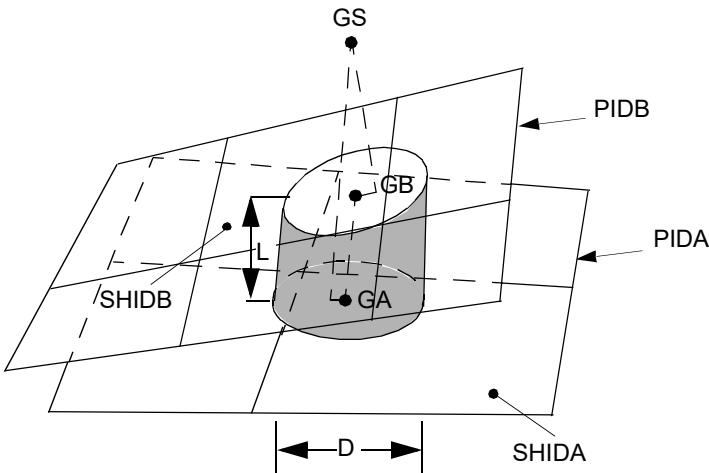


Figure 9-25 Patches Defined with TYPE= 'PROP' or TYPE= 'ELEM'

2. GS defines the approximate location of the fastener in space. GS is projected onto the surface patches A and B. The resulting piercing points GA and GB define the axis of the fastener. GS does not have to lie on the surfaces of the patches. GS must be able to project normals to the two patches. GA can be specified in lieu of GS, in which case GS will be ignored. If neither GS nor GA is specified, then (XS, YS, ZS) in basic must be specified.

If both GA and GB are specified, they must lie on or at least have projections onto surface patches A and B respectively. If GA and GB are both specified, GS is ignored. By default, the locations of user specified GA and GB will not be changed. If the user specifies "SWLDP RM, MOVGAB, 1," then the locations will be corrected so that they lie on the surface patches A and B within machine precision. The length of the fastener is the final distance between GA and GB. If the length is zero, the normal to patch A is used to define the axis of the fastener.

Diagnostic printouts, checkout runs and control of search and projection parameters are requested on the SWLDP RM Bulk Data entry.

3. The use of param,cfdiagp,yes and param,cfranel,real_fraction_value allows for the random removal of a percentage of CFAST elements for failure studies.
4. This entry is not supported in SOL 700.
5. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 9 for that entry.
6. In SOL400, the behavior of this element in regard to large rotation is affected by the Case Control Command Rigid.
7. If partitioned superelements are present, then CFAST is supported in the main Bulk Data section only.

CFLUIDi**Fluid Element Connections**

Defines three types of fluid elements for an axisymmetric fluid model.

Formats:

1	2	3	4	5	6	7	8	9	10
CFLUID2	EID	IDF1	IDF2			RHO	B		
CFLUID3	EID	IDF1	IDF2	IDF3		RHO	B		
CFLUID4	EID	IDF1	IDF2	IDF3	IDF4	RHO	B		

Examples:

CFLUID2	100	11	14			.025	0.0		
CFLUID3	110	15	13	12		1.2			
CFLUID4	120	11	15	12	14				

Descriptor	Meaning
EID	Unique element identification number. (0 < Integer < 100,000,000)
IDFi	Identification number of a RINGFL entry. (Integer > 0; IDF1 ≠ IDF2 ≠ IDF3 ≠ IDF4 ; all IDFi < 500000)
RHO	Mass density. (Real > 0.0; Default is the value of DRHO on the AXIF entry)
B	Bulk modulus, pressure per volume ratio. (Real; Default is the value of DB on the AXIF entry)

Remarks:

1. CFLUIDi is allowed only if an AXIF entry is also present.
2. Element identification number must be unique with respect to all other fluid, scalar, and structural elements.

3. The volume defined by IDF_i is a body of revolution about the polar axis of the fluid coordinate system defined by AXIF. CFLUID2 defines a thick disk with IDF1 and IDF2 defining the outer corners as shown in [Figure 9-26](#):

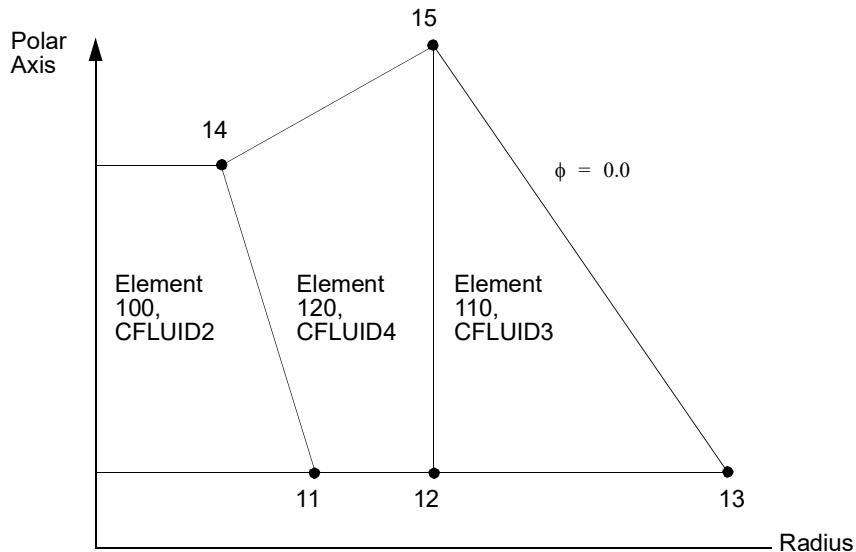


Figure 9-26 CFLUIDi Examples

4. All interior angles must be less than 180°.
5. The order of connected RINGFL points is arbitrary.
6. If B = 0.0, the fluid is incompressible.

CGAP**Gap Element Connection**

Defines a gap or friction element.

Format:

1	2	3	4	5	6	7	8	9	10
CGAP	EID	PID	GA	GB	X1	X2	X3	CID	

Example:

CGAP	17	2	110	112	5.2	0.3	-6.1		
------	----	---	-----	-----	-----	-----	------	--	--

Alternate Format and Example:

CGAP	EID	PID	GA	GB	GO			CID	
CGAP	17	2	110	112	13				

Descriptor	Meaning
EID	Element identification number. ($0 < \text{Integer} < 100,000,000$)
PID	Property identification number of a PGAP entry. ($\text{Integer} > 0$; Default = EID)
GA, GB	Connected grid points at ends A and B. ($\text{Integers} > 0$; $\text{GA} \neq \text{GB}$)
X1, X2, X3	Components of the orientation vector \vec{v} , from GA, in the displacement coordinate system at GA. (Real)
GO	Alternate method to supply the orientation vector \vec{v} using grid point GO. Direction of \vec{v} is from GA to GO. (Integer > 0)
CID	Element coordinate system identification number. CID must be specified if GA and GB are coincident ($\text{distance from GA to GB} < 10^{-4}$). See Remark 6. (Integer ≥ 0 or blank)

Remarks:

1. The CGAP element is intended for the nonlinear solution sequences 106, 129, 153, 159, and 400. However, it will produce a linear stiffness matrix for the other solutions, but remains linear with the initial stiffness. The stiffness used depends on the value for the initial gap opening (U0 field in the PGAP entry).
2. The gap element coordinate system is defined by one of two following methods:
 - If the coordinate system (CID field) is specified, the element coordinate system is established using that coordinate system, in which the element x-axis is in the T1 direction and the y-axis in the T2 direction. The orientation vector \vec{v} will be ignored in this case.
 - If the CID field is blank and the grid points GA and GB are not coincident (distance from A to B $\geq 10^{-4}$), then the line AB is the element x-axis and the orientation vector \vec{v} lies in the x-y plane (like the CBEAM element).

3. The element coordinate system does not rotate as a result of deflections.
4. Initial gap openings are specified on the PGAP entry and not derived from the separation distance between GA and GB.
5. Forces are requested with the FORCE or NLSTRESS Case Control command. The NLSTRESS command is only for nonlinear solutions, and the output also includes the gap STATUS. Forces are output in the element coordinate system. The force F_x is positive for compression.
6. If CID is being used to define the element coordinate system and the CID refers to either a cylindrical or spherical coordinate system then grid GA will be used to locate the system. If grid GA lies on the z-axis of the cylindrical or spherical coordinate system it is recommended that a different coordinate system be used for this element.
7. See PARAM,CDITER for an alternative approach.

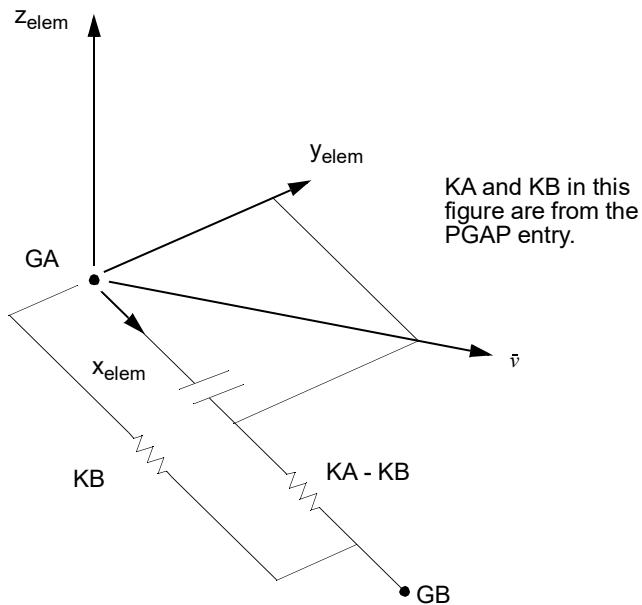


Figure 9-27 CGAP Element Coordinate System

8. Since a large stiffness is used for KA (the closed GAP stiffness), param,g damping should be avoided. Instead damping should be specified on the MATi entries and PARAM,W4 set.

CHACAB**Acoustic Absorber Element Connection**

Defines the acoustic absorber element in coupled fluid-structural analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CHACAB	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12			
			G17	G18	G19	G20			

Example:

CHACAB	95	12	1	2	5	7	8	9	
	24	23							

Descriptor	Meaning
EID	Element identification number. ($0 < \text{Integer} < 100,000,000$)
PID	Property identification number of a PACABS entry. ($\text{Integer} > 0$)
Gi	Grid point identification numbers of connection points. ($\text{Integer} \geq 0$ or blank)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Grid points G1 through G4 must be given in consecutive order about one quadrilateral face. G5 through G8 must be on the opposite face with G5 opposite G1, G6 opposite G2, etc.
3. The edge points, G9 to G20, are optional. Any or all of them may be deleted. If the ID of any edge connection point is left blank or set to zero (as for G9 and G10 in the example), the equations of the element are adjusted to give correct results for the reduced number of connections. Corner grid points cannot be deleted.
4. It is recommended that the edge points be located within the middle third of the edge.
5. The face consisting of grid points G1 through G4 and G9 through G12 is assumed to be in contact with the structure.

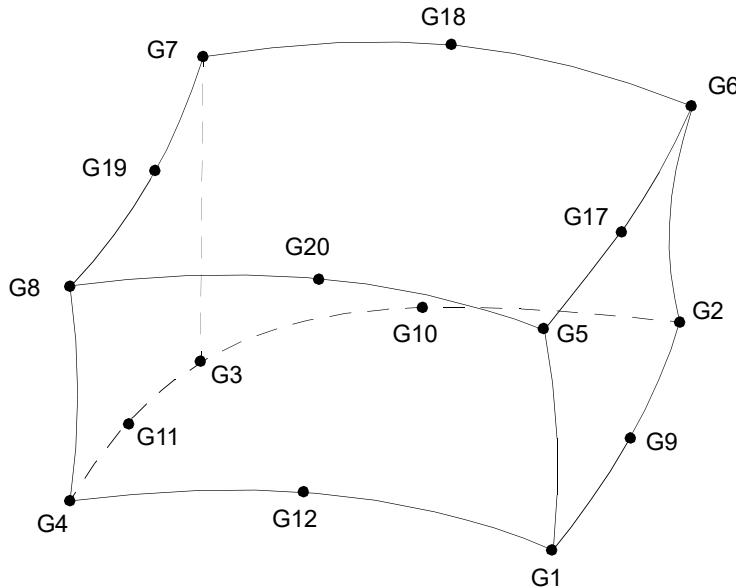


Figure 9-28 CHACAB Element Connection

6. The mass is lumped to the face formed by grid points G5 through G8 and G17 through G20 and defined to be in contact with the fluid. The opposite face has no mass contribution due to the absorber element. Also, the face in contact with the fluid has only translational stiffness in the direction normal to the face.

CHACBR**Acoustic Barrier Element Connection**

Defines the acoustic barrier element.

Format:

1	2	3	4	5	6	7	8	9	10
CHACBR	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12			
			G17	G18	G19	G20			

Example:

CHACBR	95	12	1	2	5	7	8	9	
	24	23							

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PACBAR entry. (Integer > 0)
Gi	Grid point identification numbers of connection points. (Integer > 0)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Grid points G1 through G4 must be given in consecutive order about one quadrilateral face. G5 through G8 must be on the opposite face with G5 opposite G1, G6 opposite G2, etc.
3. The edge points, G9 to G20, are optional. Any or all of them may be deleted. If the ID of any edge connection point is left blank or set to zero (as for G9 and G10 in the example), the equations of the element are adjusted to give correct results for the reduced number of connections. Corner grid points cannot be deleted.
4. It is recommended that the edge points be located within the middle third of the edge.
5. The face consisting of grids G1 through G4 and G9 through G12 is assumed to be the backing that corresponds to MBACK on the PACBAR entry.
6. The face consisting of grid points G5 through G8 and G17 through G20 is assumed to be the septum that corresponds to MSEPTM on the PACBAR entry.

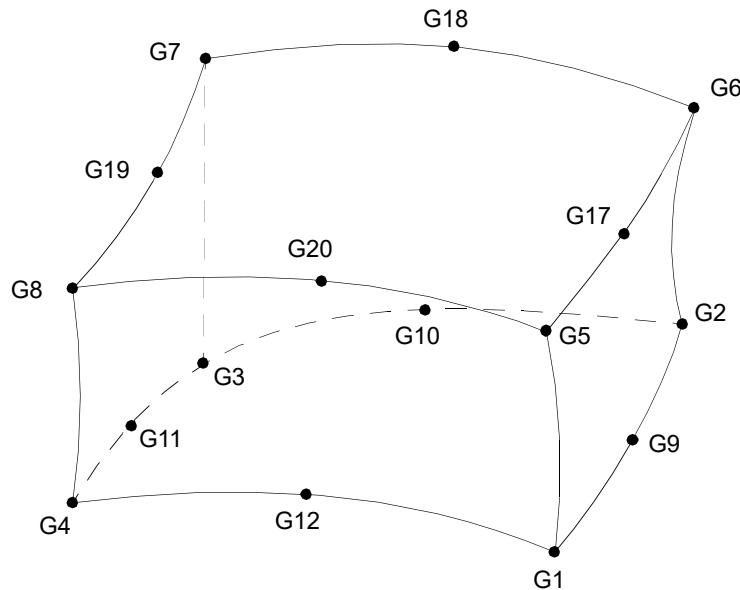


Figure 9-29 CHACBR Element Connection

7. The face in contact with the fluid is defined to be the face formed by grid points G5 through G8 and G17 through G20 and has only translational stiffness in the direction normal to the face.

CHBDYE

Geometric Surface Element Definition (Element Form)

Defines a boundary condition surface element with reference to a heat conduction element.

Format:

1	2	3	4	5	6	7	8	9	10
CHBDYE	EID	EID2	SIDE	IVIEWF	IVIEWB	RADMIDF	RADMIDB		

Example:

CHBDYE	2	10	1	3	3	2	2		
--------	---	----	---	---	---	---	---	--	--

Descriptor	Meaning
EID	Surface element identification number for a specific side of a particular element. See Remarks 1. and 9. (Unique ($0 < \text{Integer} < 100,000,000$) among all elements.)
EID2	A heat conduction element identification number. ($\text{Integer} > 0$)
SIDE	A consistent element side identification number. See Remark 6. ($1 \leq \text{Integer} \leq 6$)
IVIEWF	A VIEW entry identification number for the front face of surface element. See Remark 2. for default. ($\text{Integer} \geq 0$)
IVIEWB	A VIEW entry identification number for the back face of surface element. See Remark 2. for default. ($\text{Integer} \geq 0$)
RADMIDF	RADM identification number for front face of surface element. See Remark 2. for default. ($\text{Integer} \geq 0$)
RADMIDB	RADM identification number for back face of surface element. See Remark 2. for default. ($\text{Integer} \geq 0$)

Remarks:

1. EID is a unique elemental ID associated with a particular surface element. EID2 identifies the general heat conduction element being considered for this surface element.
2. The defaults for IVIEWF, IVIEWB, RADMIDF, and RADMIDB may be specified on the BDYOR entry. If a particular field is blank both on the CHBDYE entry and the BDYOR entry, then the default is zero.
3. For the front face of shell elements, the right-hand rule is used as one progresses around the element surface from G1 to G2 to ... Gn. For the edges of shell elements or the ends of line elements, an outward normal is used to define the front surface.
4. If the surface element is to be used in the calculation of view factors, it must have an associated VIEW entry.
5. All conduction elements to which any boundary condition is to be applied must be individually identified with the application of one of the surface element entries: CHBDYE, CHBDYG, or CHBDYP.

6. Side conventions for solid elements.

The sides of the solid elements are numbered consecutively according to the order of the grid point numbers on the solid element entry. The sides of solid elements are either quadrilaterals or triangles. For each element type, tabulate the grid points (gp) at the corners of each side.

8-node or 20-node CHEXA

side	gp	gp	gp	gp
1	4	3	2	1
2	1	2	6	5
3	2	3	7	6
4	3	4	8	7
5	4	1	5	8
6	5	6	7	8

CPENTA

side	gp	gp	gp	gp
1	3	2	1	
2	1	2	5	4
3	2	3	6	5
4	3	1	4	6
5	4	5	6	

CTETRA

side	gp	gp	gp
1	3	2	1
2	1	2	4
3	2	3	4
4	3	1	4

CPYRAM

side	gp	gp	gp	gp
1	3	2	5	
2	1	4	5	
3	2	1	5	
4	3	5	4	
5	4	1	2	3

7. Side conventions for shell elements.

Side 1 of shell elements (top) are of an AREA type, and additional sides (2 through a maximum of 5 for a QUAD) are of LINE type. (See [CHBDYG, 1525](#) for surface type definition.)

Area Type Sides -- The first side is that given by the right-hand rule on the shell elements grid points.

Line Type Sides -- The second side (first line) proceeds from grid point 1 to grid point 2 of the shell element, and the remaining lines are numbered consecutively. The thickness of the line is that of the shell element, and the normal to the line is outward from the shell element in the plane of the shell. Note that any midside nodes are ignored in this specification.

For 3-D heat shell elements when used in combination with linear or quadratic nodal temperature (see option TEMPP of the NLMOPTS entry), SIDE=6 refers to BOT surface. SIDE=1 refers to TOP surface. The IVIEW and RADM should be filled in consistently in relation with the SIDE entry. E.g., when SIDE=1 or 6 then only IVIEWF makes sense.

8. Side conventions for line elements.

LINE elements have one linear side (side 1) with geometry that is the same as that of the element and two POINT-type sides corresponding to the two points bounding the linear element (first grid point-side 2; second grid point-side 3).

The TUBE-type element has two linear sides of type TUBE. The first side represents the outside with diameters equal to that of the outside of the tube. The second side represents the inside with diameters equal to that of the inside of the tube.

Point Sides -- Point sides may be used with any linear element. The direction of the outward normals of these points is in line with the element axis, but pointing away from the element. The area assigned to these POINT-type sides is consistent with the element geometry.

Rev Sides -- The CTRIA6 element has associated with it three REV sides. The first side is associated with Grid Points G1, G2, and G3. The positive face identification normals point away from the element.

9. Application of boundary conditions to CHBDYE is referenced through the EID. Boundary conditions can reference either the front or back face of the CHBDYE by specifying +EID or -EID respectively. Correspondingly, the back face is minus the normal vector of the front face. Similarly, IVIEWF and RADMIDF are associated with +EID and IVIEWB and RADMIDB with -EID. For radiation problems, if the RADMIDF or RADMIDB is zero, default radiant properties assume perfect black body behavior.
10. Starting with MSC Nastran 2004, axisymmetric view factors are supported CHBDYG of TYPE=REV, but not supported CHBDYE. If CHBDYE is used for this, axisymmetric view factors are not calculated.

CHBDYG

Geometric Surface Element Definition (Grid Form)

Defines a boundary condition surface element without reference to a property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CHBDYG	EID		TYPE	IVIEWF	IVIEWB	RADMIDF	RADMIDB		
	G1	G2	G3	G4	G5	G6	G7	G8	

Example:

CHBDYG	2		AREA4	3	3	2	2		
	100	103	102	101					

Descriptor	Meaning
EID	Surface element identification number. (Unique ($0 < \text{Integer} < 100,000,000$) among all elemental entries)
TYPE	Surface type. See Remark 3. (Character)
IVIEWF	A VIEW entry identification number for the front face. See Remark 2. for default. (Integer ≥ 0)
IVIEWB	A VIEW entry identification number for the back face. See Remark 2. for default. (Integer ≥ 0)
RADMIDF	RADM identification number for front face of surface element. See Remark 2. for default. (Integer ≥ 0)
RADMIDB	RADM identification number for back face of surface element. See Remark 2. for default. (Integer ≥ 0)
Gi	Grid point IDs of grids bounding the surface. (Integer > 0)

Remarks:

1. EID is a unique ID associated with a particular surface element as defined by the grid points.
2. The defaults for TYPE, IVIEWF, IVIEWB, RADMIDF, and RADMIDB may be specified on the BDYOR entry. If a particular field is blank on both the CHBDYG entry and the BDYOR entry, then the default is zero.
3. TYPE specifies the kind of element surface; allowed types are: REV, REV1, AREA3, AREA4, AREA6, and AREA8. See [Figure 9-30](#), [Figure 9-31](#), and [Figure 9-32](#).
 - TYPE = REV

The “REV” type has two primary grid points that must lie in the x-z plane of the basic coordinate system with $x > 0$. A midside grid point G3 is optional and supports convection or heat flux from the edge of the six-noded CTRIA6 element. The defined area is a conical section with z as the axis of symmetry. A property entry is required for convection, radiation, or thermal vector flux.

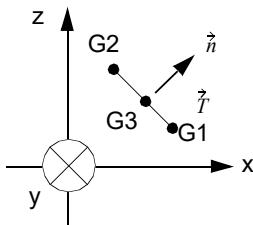


Figure 9-30 Normal Vector for CHBDYG Element of Type “REV”

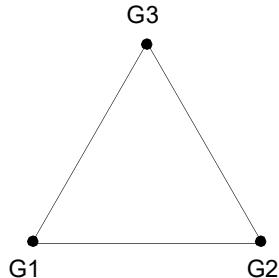
The unit normal lies in the x-z plane, and is given by

$$\hat{n} = (\hat{e}_y \times \vec{T}) / |\hat{e}_y \times \vec{T}| .$$

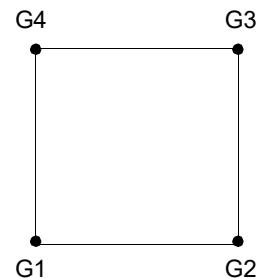
\hat{e}_y is the unit vector in the y direction.

- TYPE = REV1. The “REV1” type has two primary grid points that must lie in the x-y plane of the basic coordinate system with $x > 0$. A midside grid point G3 is optional and supports convection or heat flux from the edge of the CQUADX (4 or 8 node) and CTRIA6 (3 or 6 node) element. The defined area is a conical section with y as the axis of symmetry. A property entry is required for convection, radiation, or thermal vector flux.
- TYPE = AREA3, AREA4, AREA6, or AREA8

These types have three and four primary grid points, respectively, that define a triangular or quadrilateral surface and must be ordered to go around the boundary. A property entry is required for convection, radiation, or thermal vector flux.



AREA3



AREA4

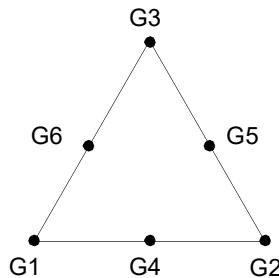
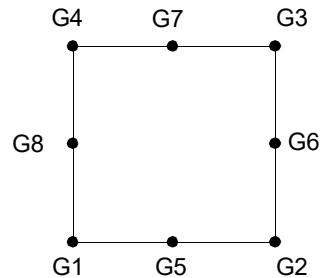
AREA6
(Grid points G4 through G6 optional)AREA8
(Grid points G5 through G8 optional)

Figure 9-31 TYPE Examples

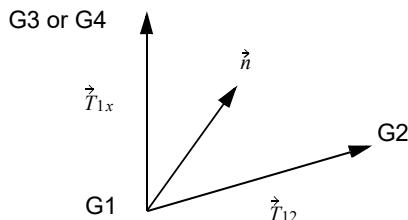


Figure 9-32 Normal Vector for CHBDYG Element of Types "AREAI"

The unit normal vector is given by

$$\hat{n} = \frac{(\vec{T}_{12} \times \vec{T}_{1x})}{|\vec{T}_{12} \times \vec{T}_{1x}|}$$

(G3 is used for triangles, and G4 is used for quadrilaterals.)

4. For defining the front face, the right-hand rule is used on the sequence G1 to G2 to ... Gn of grid points.

5. If the surface element is to be used in the calculation of view factors, it must have an associated VIEW entry.
6. All conduction elements to which any boundary condition is to be applied must be individually identified with one of the surface element entries: CHBDYE, CHBDYG, or CHBDYP.
7. See Remark 9. of CHBDYE for application of boundary conditions using CHBDYG entries and a discussion of front and back faces.

CHBDYP

Geometric Surface Element Definition (Property Form)

Defines a boundary condition surface element with reference to a PHBDY entry.

Format:

1	2	3	4	5	6	7	8	9	10
CHBDYP	EID	PID	TYPE	IVIEWF	IVIEWB	G1	G2	G0	
	RADMIDF	RADMIDB	GMID	CE	E1	E2	E3		

Example:

CHBDYP	2	5	POINT	2	2	101		500	
	3	3			0.0	0.0	1.0		

Descriptor	Meaning
EID	Surface element identification number. (Unique $0 < \text{Integer} < 100,000,000$ among all element identification numbers.)
PID	PHBDY property entry identification numbers. ($\text{Integer} > 0$)
TYPE	Surface type. See Remark 3. (Character)
IVIEWF	VIEW entry identification number for the front face. ($\text{Integer} \geq 0$ or blank)
IVIEWB	VIEW entry identification number for the back face. ($\text{Integer} \geq 0$ or blank)
G1, G2	Grid point identification numbers of grids bounding the surface. ($\text{Integer} > 0$)
GO	Orientation grid point. ($\text{Integer} \geq 0$; Default = 0)
RADMIDF	RADM entry identification number for front face. ($\text{Integer} \geq 0$ or blank)
RADMIDB	RADM entry identification number for back face. ($\text{Integer} \geq 0$ or blank)
GMID	Grid point identification number of a midside node if it is used with the line type surface element.
CE	Coordinate system for defining orientation vector. ($\text{Integer} \geq 0$; Default = 0)
Ei	Components of the orientation vector in coordinate system CE. The origin of the orientation vector is grid point G1. (Real or blank)

Remarks:

1. EID is a unique ID associated with a particular surface element as defined by the grid point(s).
2. The defaults for PID, TYPE, IVIEWF, IVIEWB, GO, RADMIDF, RADMIDB, CE, and Ei may be specified on the BDYOR entry. If a particular field is blank on both the CHBDYP entry and the BDYOR entry, then the default is zero.

3. TYPE specifies the kind of element surface; the allowed types are: "POINT," "LINE," "ELCYL," "FTUBE," and "TUBE." For TYPE = "FTUBE" and TYPE = "TUBE," the geometric orientation is completely determined by G1 and G2; the GO, CE, E1, E2, and E3 fields are ignored.

- TYPE = "POINT"

TYPE = "POINT" has one primary grid point, requires a property entry, and the normal vector \vec{V} must be specified if thermal flux is to be used.

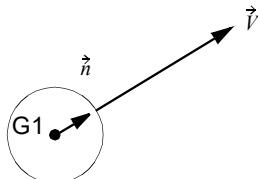


Figure 9-33 Normal Vector for CHBDYP Element of Type "POINT"
(See Remarks 4. and 5.)

The unit normal vector is given by $\hat{n} = \vec{V}/|\vec{V}|$ where \vec{V} is specified in the Ei field and given in the basic system at the referenced grid point. See Remarks 4. and 5. for the determination of \vec{V} .

- TYPE = "LINE," "FTUBE," or "TUBE"

The TYPE = "LINE" type has two primary grid points, requires a property entry, and the vector is required. TYPE = "FTUBE" and TYPE = "TUBE" are similar to TYPE = "LINE" except they can have linear taper with no automatic view factor calculations. GMID is an option for the TYPE = "LINE" surface element only and is ignored for TYPE = "FTUBE" and "TUBE".

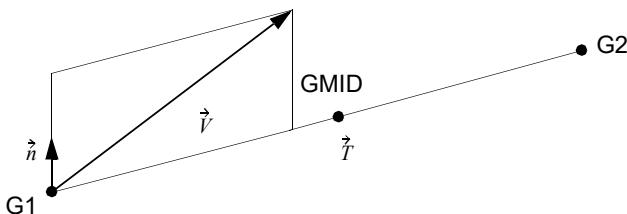


Figure 9-34 Normal Vector for CHBDYP Element with TYPE="LINE",
TYPE="FTUBE", or TYPE="TUBE" (See Remarks 4. and 5.)

The unit normal lies in the plane \vec{V} and \vec{T} , is perpendicular to \vec{T} , and is given by:

$$\hat{n} = \frac{\vec{T} \times (\vec{V} \times \vec{T})}{|\vec{T} \times (\vec{V} \times \vec{T})|}$$

- TYPE = "ELCYL"

TYPE = "ELCYL" (elliptic cylinder) has two connected primary grid points and requires a property entry. The vector must be nonzero.

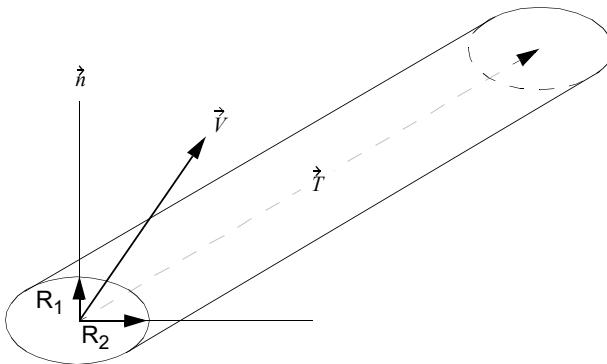


Figure 9-35 Normal Vector for CHBDYP Element of TYPE="ELCYL"
(See Remarks 4. and 5.)

The same logic is used to determine \hat{n} as for TYPE = LINE. The "radius" R_1 is in the \hat{n} direction, and R_2 is the perpendicular to \hat{n} and \hat{T} (see fields 7 and 8 of PHBDY entry).

4. For TYPE = "POINT," TYPE = "LINE," and TYPE = "ELCYL," geometric orientation is required. The required information is sought in the following order:
 - If GO > 0 is found on the CHBDYP entry, it is used.
 - Otherwise, if a nonblank CE is found on the CHBDYP continuation entry, this CE and the corresponding vectors E1, E2, and E3 are used.
 - If neither of the above, the same information is sought in the same way from the BDYOR entry.
 - If none of the above apply, a warning message is issued.
5. The geometric orientation can be defined by either GO or the vector E1, E2, E3.
 - If GO > zero:

For a TYPE = "POINT" surface, the normal to the front face is the vector from G1 to GO. For the TYPE = "LINE" surface, the plane passes through G1, G2, GO and the right-hand rule is used on this sequence to get the normal to the front face. For TYPE = "ELCYL" surface the first axis of the ellipse lies on the G1, G2, GO plane, and the second axis is normal to this plane. For TYPE = "FTUBE" or "TUBE" surface, no orientation is required, and GO is superfluous.
 - If GO is zero:

For a TYPE = "POINT" surface, the normal to the front face is the orientation vector. For the TYPE = "LINE" surface, the plane passes through G1, G2, and the orientation vector; the front face is based on the right-hand rule for the vectors G2-G1 and the orientation vector. For TYPE = "ELCYL" surface, the first axis of the ellipse lies on the G1, G2, orientation vector plane, and the second axis is normal to this plane.
6. The continuation entry is optional.
7. If the surface element is to be used in the calculation of view factors, it must have an associated VIEW entry.

8. All conduction elements to which any boundary condition is to be applied must be individually identified with the application of one of the surface element entries: CHBDYE, CHBDYG, or CHBDYP entries.
9. For RC network solver in thermal analysis, the G0, GMID, CE, E1, C2 and E3 are ignored.

CHEXA**Six-Sided Solid Element Connection**

Defines the connections of the six-sided solid element with eight to twenty grid points or the six-sided solid shell element with eight grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CHEXA	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15	G16	G17	G18	G19	G20			

Example:

CHEXA	71	4	3	4	5	6	7	8	
	9	10	0	0	30	31	53	54	
	55	56	57	58	59	60			

Descriptor	Meaning	Type	Default
EID	Element identification number. ($0 < \text{Integer} < 100,000,000$)	$\text{Integer} > 0$	Required
PID	Property identification number of a PSOLID, PLSOLID or PCOMPLS entry.	$\text{Integer} > 0$	Required
Gi	Grid point identification numbers of connection points. Integer ≥ 0 or blank	Integer ≥ 0 or blank	Required

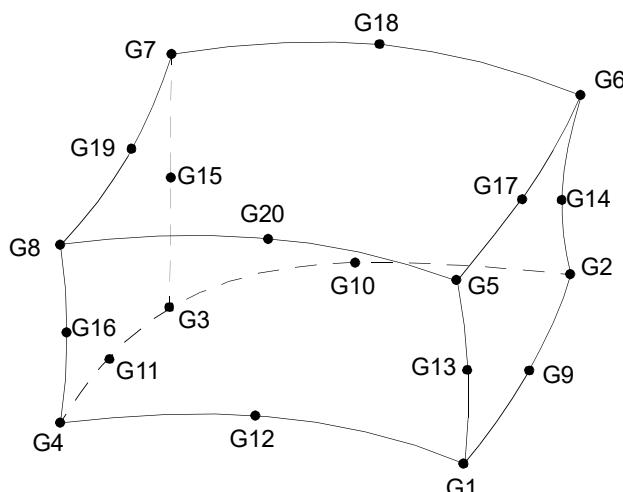


Figure 9-36 CHEXA Element Connection

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Grid points G1 through G4 must be given in consecutive order about one quadrilateral face. G5 through G8 must be on the opposite face with G5 opposite G1, G6 opposite G2, etc.
3. For Nastran conventional element, the edge points, G9 to G20, are optional. Any or all of them may be deleted. If the ID of any edge connection point is left blank or set to zero (as for G9 and G10 in the input example), the equations of the element are adjusted to give correct results for the reduced number of connections. Corner grid points cannot be deleted. The element is an isoparametric element (with shear correction) in all cases.
Note that for advanced nonlinear elements, partial specification of mid-points is not allowed. i.e. Either none of the mid-points should be specified or all of the mid points should be specified.
4. Components of stress are output in the material coordinate system except for hyperelastic elements, which are output in the basic coordinate system. The material coordinate system is defined on the PSOLID entry.
5. The second continuation is optional.
6. For nonhyperelastic and non-composite elements, the element coordinate system for the CHEXA element is defined in terms of the three vectors R, S, and T, which join the centroids of opposite faces.
R vector joins the centroids of faces G4-G1-G5-G8 and G3-G2-G6-G7.
S vector joins the centroids of faces G1-G2-G6-G5 and G4-G3-G7-G8.
T vector joins the centroids of faces G1-G2-G3-G4 and G5-G6-G7-G8.

The origin of the coordinate system is located at the intersection of these vectors. The X, Y, and Z axes of the element coordinate system are chosen as close as possible to the R, S, and T vectors and point in the same general direction. (Mathematically speaking, the coordinate system is computed in such a way that if the R, S, and T vectors are described in the element coordinate system a 3×3 positive-definite symmetric matrix would be produced.)

Solid elements have both a material and an element coordinate system. Both systems are defined for the initial geometry, and for geometric nonlinear analysis they will rotate with the element. The material coordinate system is used to input anisotropic material properties and for stress output. The material coordinate system is defined by the CORDM field of the PSOLID entry. The element coordinate system is used for element stiffness integration (reduced shear for example) and optionally to define the material coordinate system (only if PSOLID,CORDM=-1).

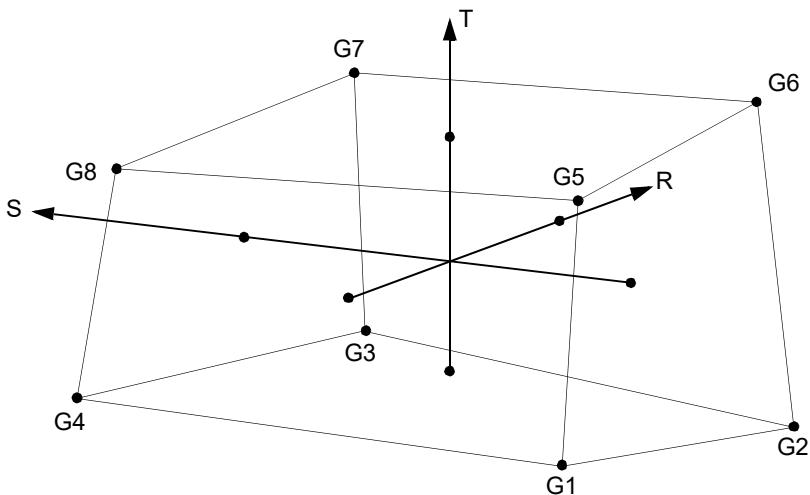


Figure 9-37 CHEXA Element R, S, and T Vectors

7. It is recommended that the edge points be located within the middle third of the edge.
8. For hyperelastic elements, the plot codes are specified under the CHEXA FD element name in [Item Codes, 1045](#).
9. If a CHEXA element is referenced by a PSET or PVAL entry, then a p-version formulation is used and the element can have curved edges.
 - If a curved edge of a p-element is shared by an h-element without midside nodes, the geometry of the edge is ignored and set straight.
 - Elements with midside nodes cannot be p-elements and edges with midside nodes cannot be shared by p-elements.
10. By default, all of the twelve edges of the element are considered straight unless:
 - For p-elements there is an FEEDGE or FEFACE entry that contains the two grids of any edge of this element. In this case, the geometry of the edge is used in the element.
 - For h-elements, any of G9 through G20 are specified.
11. This element may be used:
 - a. with BEH8=SLCOMP and INT8=ASTN, as a three-dimensional eight-node solid shell element in conjunction with PSLDN1 entry (SOL400 only),
 - b. with BEH8=SLCOMP and INT8=ASTN, as a three-dimensional eight-node composite solid shell element in conjunction with PCOMPLS entry with BEH8 = SLCOMP (INT8 = ASTN) (SOL 400 and all linear solution sequences between SOL101 and SOL112), or
 - c. with BEH8=SLCOMP and INT8=L or Q, or BEH20=SLCOMP and INT8=Q.
12. When this element is used as a three-dimensional eight-node solid shell element or three-dimensional eight-node composite solid shell element, the user should keep in mind that the layer orientation is required to be in the element T-direction, when specifying grid order.

13. The internal coordinate system of the element is used internally and is based on eigenvalue techniques to insure non bias in the element formulation. For stress/strain output this internal coordinate system (CORDM=-1 on PSOLID entry) is hard to visualize. Thus a CORDM=-2 on the PSOLID is available as shown in [Figure 9-38](#).

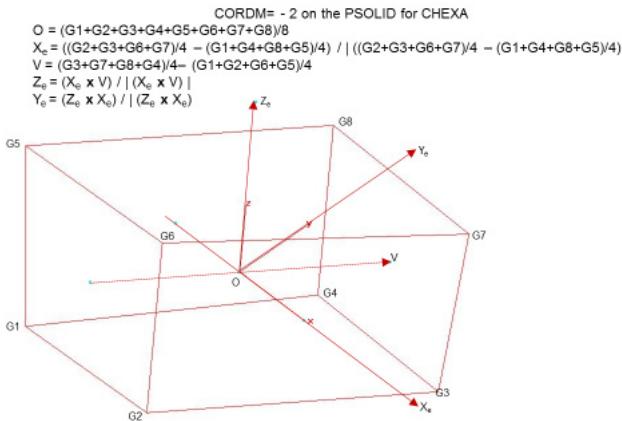


Figure 9-38 PSOLID for CHEXA

CIFHEX**Solid InterFace Cohesive Zone Modeling Element**

Linear/Quadratic, twenty-node, three-dimensional interface element used to simulate the onset and progress of delamination in SOL 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
CIFHEX	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15	G16	G17	G18	G19	G20			

Example:

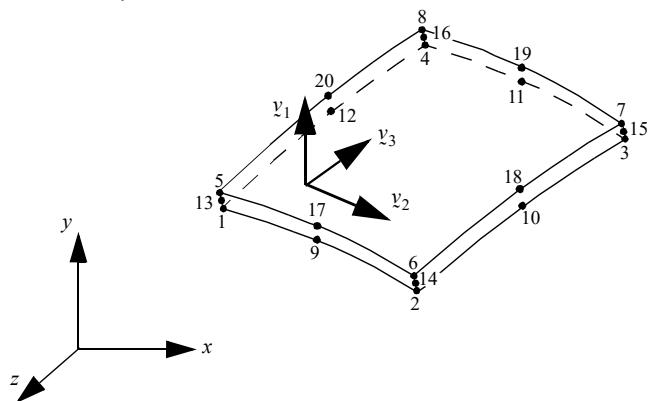
CIFHEX	700	701	456	357	882	889	443	447	
	162	911							

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property number of a PCOHE entry. (Integer > 0)
G1-G8	Identification number of connected corner grid points. Required data for all eight corner grid points. (Unique Integer > 0)
G9-G12	Identification number of connected edge grid points. Optional data for bottom and top edge grid points. (Unique Integer > 0)
G17-G20	Identification number of connected midside grid points. Optional data for midside grid points used only to make the element compatible with twenty-noded hexahedral elements. (Unique Integer > 0)
G13-G16	

Remarks:

1. Element identification numbers should be unique with respect to all other element ID's of any kind.
2. Grid points Gi must be numbered as shown in the following figure. Specify either G1-G8, or all G1-G20.
3. The element is typically used to model the interface between different materials, where G1, G9, G2, G10, G3, G11, G4 and G12 correspond to one side (called the bottom) and G5, G17, G6, G18, G7, G19, and G20 correspond to the other side (called the top). The stress components are one normal and two shear tractions. When only G1-G8 are specified, the element is linear. When in addition to G1-G8, G9-G12, G17-G20 are specified, the element is quadratic.
4. The corresponding deformations are relative displacements between the top and bottom edge of the element.

5. The element is allowed to be infinitesimally thin; in which case edges defined by grids G1-G4 and G5-G8 may coincide.



6. This element does not support thermal load. No thermal strain will be generated.

CIFPENT**Solid InterFace Cohesive Zone Modeling Element**

Linear/Quadratic, fifteen-node, three-dimensional interface element used to simulate the onset and progress of delamination in SOL 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
CIFPENT	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15								

Example:

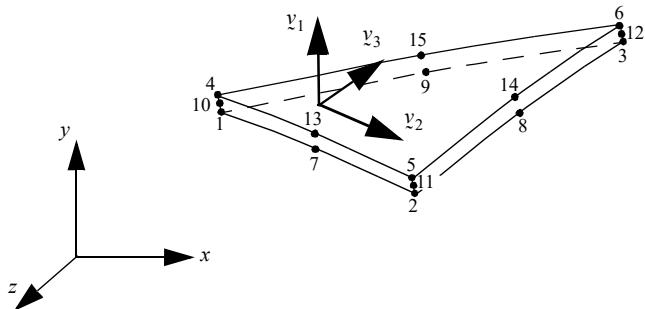
CIFPENT	700	701	456	357	882	889	443	447	
---------	-----	-----	-----	-----	-----	-----	-----	-----	--

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property number of a PCOHE entry. (Integer > 0)
G1-G6	Identification number of connected corner grid points. Required data for all four corner grid points. (Unique Integer > 0)
G7-G9	Identification number of connected edge grid points. Optional data for bottom and top edge grid points. (Unique Integer > 0)
G13-G15	
G10-G12	Identification number of connected midside grid points. Optional data for midside grid points used only to make the element compatible with fifteen-noded pentahedral elements. (Unique Integer > 0)

Remarks:

1. Element identification numbers should be unique with respect to all other element ID's of any kind.
2. Grid points Gi must be numbered as shown in the following figure. Specify either G1-G6 or all G1-G15.
3. The element is typically used to model the interface between different materials, where G1, G7, G2, G8, G3 and G9 correspond to one side (called the bottom) and G4, G13, G5, G14, G6, and G15 correspond to the other side (called the top). The stress components are one normal and one shear tractions. When only G1-G6 are specified, the element is linear. When in addition to G1-G6, G7-G9, G13-G15 are specified, the element is quadratic.
4. The corresponding deformations are relative displacements between the top and bottom edge of the element.

5. The element is allowed to be infinitesimally thin; in which case edges defined by grids G1-G3 and G4-G6 may coincide.



6. This element does not support thermal load. No thermal strain will be generated.

CIFQDX**Axisymmetric InterFace Cohesive Zone Modeling Element**

Linear/Quadratic, eight-node, axisymmetric interface element used to simulate the onset and progress of delamination in SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
CIFQDX	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8							

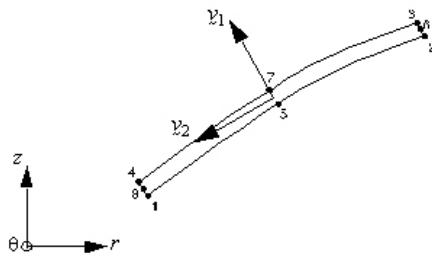
Example:

CIFQDX	700	701	456	357	882	889	443	447	
	1612	911							

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property number of a PCOHE entry. (Integer > 0)
G1-G4	Identification number of connected corner grid points. Required data for all four corner grid points. (Unique Integer > 0)
G5, G7	Identification number of connected edge grid points. Optional data for bottom and top edge grid points. (Unique Integer > 0)
G6, G8	Identification number of connected edge grid points. Optional data for side grid points used only to make the element compatible with eight-noded quadrilateral axisymmetric elements. (Unique Integer > 0)

Remarks:

1. Element identification numbers should be unique with respect to all other element ID's of any kind.
2. Grid points Gi must be numbered as shown in the following figure.
3. The element is typically used to model the interface between different materials, where G1, G5, and G2 correspond to one side (called the bottom) and G3, G7, and G4 correspond to the other side (called the top). The stress components are one normal and one shear traction. Then only G1-G4 are specified, the element is linear. When in addition to G1-G4, G5 and G7 are specified, the element is quadratic.
4. The corresponding deformation are relative displacements between the top and bottom edge of the element.
5. The element is allowed to be infinitesimally thin; in which case edges G1-G5-G2 and G3-G7-G4 may coincide.
6. The element must lie in the x-y plane of the basic system. Coordinate r is parallel to the x-basic and coordinate z is parallel to y-basic.



7. This element does not support thermal load. No thermal strain will be generated.

CIFQUAD**Planar InterFace Cohesive Zone Modeling Element**

Linear/Quadratic, eight-noded planar interface element used to simulate the onset and progress of delamination in SOL 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
CIFQUAD	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8							

Example:

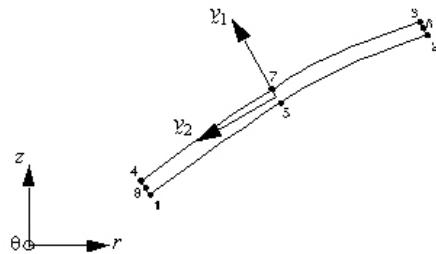
CIFQUAD	700	701	456	357	882	889	443	447	
	1612	911							

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property number of a PCOHE entry. (Integer > 0)
G1-G4	Identification number of connected corner grid points. Required data for all four corner grid points. (Unique Integer > 0)
G5, G7	Identification number of connected edge grid points. Optional data for bottom and top edge grid points. (Unique Integer > 0)
G6, G8	Identification number of connected edge grid points. Optional data for side grid points used only to make the element compatible with eight-noded quadrilateral elements. (Unique Integer > 0)

Remarks:

1. Element identification numbers should be unique with respect to all other element ID's of any kind.
2. Grid points Gi must be numbered as shown in the following figure.
3. The element is typically used to model the interface between different materials, where G1, G5, and G2 correspond to one side (called the bottom) and G3, G7, and G4 correspond to the other side (called the top). The stress components are one normal and one shear traction. Then only G1-G4 are specified, the element is linear. When in addition to G1-G4, G5 and G7 are specified, the element is quadratic.
4. The corresponding deformation are relative displacements between the top and bottom edge of the element.
5. The element is allowed to be infinitesimally thin; in which case edges G1-G5-G2 and G3-G7-G4 may coincide.

6. The element must lie in the x-y plane of the basic system.



7. This element does not support thermal load. No thermal strain will be generated.

CINTC**Line Interface Element Connection**

Defines a line interface element with specified boundaries.

Format:

1	2	3	4	5	6	7	8	9	10
CINTC	EID	TYPE							
LIST = (BID1(INTP1), BID2(INTP2),...,BIDn(INTPn))									

Example:

CINTC	1001	GRDLIS T							
LIST=(101,102(Q),-103(Q),104(L))									

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
TYPE	Connectivity. If TYPE = "GRDLIST" or blank (Default), the user will specify the boundaries via Bulk Data entry, GMBNDC. See Remark 2. (Character; Default = "GRDLIST")
BIDI	Boundary curve identification number, referenced to Bulk Data entry, GMBNDC. See Remark 2. (Integer ≠ 0)
INTPi	Interpolation scheme. (Character; Default = "L") INTP = "L": Linear interpolation; INTP = "Q": Quadratic interpolation.

Remarks:

1. Line interface element identification numbers must be unique with respect to all other line interface elements.
2. There must be at least two BIDI specified. If all BIDI are positive, by default, the degrees of freedom associated with the grids on the boundary represented by the first BID will be taken as the independent (n-set), and the degrees of freedom with the grids on the rest of boundaries are taken as the dependent (m-set). If there is a single negative BID, the degrees of freedom associated with the grids on the boundary represented by this BID will be taken as the independent (n-set), and the rest of the degrees of freedom with other boundaries are used as the dependent (m-set). If there are two or more negative BIDs, the degrees of freedom with the first negative one will be taken as the independent.
3. Forces of multipoint constraints may be recovered with the MPCFORCE Case Control command.
4. The m-set degrees of freedom specified on the boundary grids by this entry may not be specified by other entries that define mutually exclusive sets.

CLOAD

Static Load Combination for Superelement Loads (Superposition)

Defines a static load as a linear combination of previously calculated superelement loads defined by the LSEQ entry in nonlinear static analysis (SOLs 106 or 153).

Format:

1	2	3	4	5	6	7	8	9	10
CLOAD	CID	S	S1	IDV1	S2	IDV2	S3	IDV3	
	S4	IDV4	-etc.-						

Example:

CLOAD	25	1.0	25.0	10	-1.0	101	2.2-1	604	
	-62.0	62							

Descriptor	Meaning
CID	Combination identification number. (Integer > 0)
S	Scale factor. (Real)
Si	Scale factors. (Real)
IDVi	Identification numbers of load vectors (EXCITEID of a selected LSEQ entry) calculated for a superelement loads entry. (Integer > 0)

Remarks:

1. The CLOAD entry must be selected in the residual solution subcases of the Case Control with CLOAD = CID and must be used if loads are applied to upstream superelements in SOL 106 or 153.
2. The load vector defined is given by $\{P\} = S \sum_i S_i \{P_{IDVi}\}$
3. The IDVi field refers to a previously calculated load vector for the superelement via the LSEQ approach. That is, a LOADSET keyword must have been selected in Case Control that in turn refers to one or more LSEQ entries in the Bulk Data Section. The IDVi refers to the EXCITEID of such LSEQ entries. For more details, see the Case Control commands [LSEQ, 2190](#) Bulk Data entry and the [LOADSET \(Case\), 439](#).
4. In the CID or IDV fields, a CLOAD entry may not reference an identification number defined by another CLOAD entry.

CMARKB2**Two-Noded Marker Connectivity Definition**

Defines a 2-noded marker beam element by means of connecting two grid points. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
CMARKB2	ID	PID	G1	G2					

Example:

CMARKB2	7	1	9	10					
---------	---	---	---	----	--	--	--	--	--

Descriptor	Meaning
ID	Unique element number. (Integer > 0; Required)
PID	Property ID referring to a PMARKER entry. (Integer > 0; Required)
G1	Grid point number connectivity 1. (Integer > 0; Required)
G2	Grid point number connectivity 2. (Integer > 0; Required)

Remarks:

1. A CMARKB2 element may refer to two types of grid points:
 - a. Structural grid points, thus grid points that are part of the connectivity of an element
 - b. Free grid points in space. These grid points do not have mass associated with them. The motion of these grid points is specified by the PMARKER property
2. The ID must be unique in the model and may not be used as structural element ID.

CMARKN1

One-Noded Marker Connectivity Definition

Defines a 1-noded marker element on a grid point. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
CMARKN1	ID	PID	G1						

Example:

CMARKN1	7	1	9						
---------	---	---	---	--	--	--	--	--	--

Descriptor	Meaning
ID	Unique element number. (Integer > 0; Required)
PID	Property ID referring to a PMARKER entry. (Integer > 0; Required)
G	Grid point number. (Integer > 0; Required)

Remarks:

1. A CMARKB2 element may refer to two types of grid points:
 - a. Structural grid points, thus grid points that are part of the connectivity of an element
 - b. Free grid points in space. These grid points do not have mass associated with them. The motion of these grid points is specified by the PMARKER property
2. The ID must be unique in the model and may not be used as structural element ID.

CMASS1

Scalar Mass Connection

Defines a scalar mass element.

Format:

1	2	3	4	5	6	7	8	9	10
CMASS1	EID	PID	G1	C1	G2	C2			

Example:

CMASS1	32	6	2	1					
--------	----	---	---	---	--	--	--	--	--

Descriptor	Meaning
EID	Unique element identification number. ($0 < \text{Integer} < 100,000,000$)
PID	Property identification number of a PMASS entry. ($\text{Integer} > 0$; Default = EID)
G1, G2	Geometric grid or scalar point identification number. ($\text{Integer} \geq 0$)
C1, C2	Component number. ($0 \leq \text{Integer} \leq 6$; blank or zero if scalar point)

Remarks:

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero. If only scalar points and/or ground are involved, it is more efficient to use the CMASS3 entry.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2) must not be coincident.
4. For a discussion of the scalar elements, see the *MSC Nastran Reference Guide*, Section 5.6.
5. A scalar point specified on this entry need not be defined on an SPOINT entry.
6. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.
7. Scalar elements input coupled mass matrices when the second pair of fields is entered. When uncoupled point masses are desired input only the first pair of fields. When a coupled mass matrix is requested the submatrix added has M on the diagonal, and -M on the off-diagonal. The element is not checked for internal constraints, which is the user's responsibility if desired. There are instances where elements with internal constraints are desired, although not frequently. To identify the presence of internal constraints caused by coupled mass, inspect GPWG output, OLOAD output due to GRAV loads, and rigid body modes of free structures. Some forms of coupled mass will cause coupling of rigid body translational mass terms in GPWG output, and poor rigid body modes in modal analysis.

CMASS2

Scalar Mass Property and Connection

Defines a scalar mass element without reference to a property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CMASS2	EID	M	G1	C1	G2	C2			

Example:

CMASS2	32	9.25	6	1					
--------	----	------	---	---	--	--	--	--	--

Descriptor	Meaning
EID	Unique element identification number. (0 < Integer < 100,000,000)
M	Value of the scalar mass. (Real)
G1, G2	Geometric grid or scalar point identification number. (Integer ≥ 0)
C1, C2	Component number. (0 \leq Integer ≤ 6 ; blank or zero if scalar point)

Remarks:

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero. If only scalar points and/or ground are involved, it is more efficient to use the CMASS4 entry.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2) must be distinct. Except in unusual circumstances, one of them will be a grounded terminal with blank entries for Gi and Ci.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASI, CMASSI, CDAMPI\)](#) in the *MSC Nastran Reference Guide*.
5. A scalar point specified on this entry need not be defined on an SPOINT entry.
6. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.
7. See Remark 7 for [CMASS1, 1549](#).

CMASS3**Scalar Mass Connection to Scalar Points Only**

Defines a scalar mass element that is connected only to scalar points.

Format:

1	2	3	4	5	6	7	8	9	10
CMASS3	EID	PID	S1	S2					

Example:

CMASS3	13	42	62						
--------	----	----	----	--	--	--	--	--	--

Descriptor	Meaning
EID	Unique element identification number. ($0 < \text{Integer} < 100,000,000$)
PID	Property identification number of a PMASS entry. ($\text{Integer} > 0$; Default = EID)
S1, S2	Scalar point identification numbers. ($\text{Integer} \geq 0$; $S1 \neq S2$)

Remarks:

1. S1 or S2 may be blank or zero, indicating a constrained coordinate.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one scalar mass element may be defined on a single entry.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASI, CMASSi, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
5. A scalar point specified on this entry need not be defined on an SPOINT entry.

CMASS4**Scalar Mass Property and Connection to Scalar Points Only**

Defines a scalar mass element that is connected only to scalar points, without reference to a property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CMASS4	EID	M	S1	S2					

Example:

CMASS4	23	14.92		23					
--------	----	-------	--	----	--	--	--	--	--

Descriptor	Meaning
EID	Unique element identification number. (0 < Integer < 100,000,000)
M	Scalar mass value. (Real)
S1, S2	Scalar point identification numbers. (Integer ≥ 0 ; S1 \neq S2)

Remarks:

1. S1 or S2 may be blank or zero, indicating a constrained coordinate. This is the usual case.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one scalar mass element may be defined on a single entry.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASi, CMASSI, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
5. A scalar point specified on this entry need not be defined on an SPOINT entry.

CMREBAI

Defines Rebar Elements and Matching "Matrix" Solid Elements using the Marc REBAR with INSERT Method in SOL 600

In some cases, particularly for modeling of concrete or tires, it is beneficial to add rebar or cord material to a matrix. The resulting combined material is similar to a composite but it is sometimes easier to postprocess the stresses of the rebar and matrix separately to determine failure conditions. Unlike the CMREBAR element, this element can span multiple matrix CHEXA elements (for example, could be allied to all elements on the bottom of a flat surface modeled with several layers of CHEXA matrix elements through the thickness).

Format:

1	2	3	4	5	6	7	8	9	10
CMREBAI	ID	IP	G1	G2	G3	G4	G5	G6	
	G7	G8							

Example:

CMREBAI	100	2	31	32	88	87			
CMREBAI	250	8	1	2	3	4	5	6	
	7	8							

Descriptor	Meaning
ID	Rebar element ID should be distinct between all element ID's. (Integer; Required; no Default)
IP	Property identification of a matching PMRBAR entry. (Integer; Required; no Default)
G1-G4	Grid point identification numbers of the four corner points. (Integer > 0; all unique)
G5-G8	Grid point identification numbers of the four mid-side nodes similar to G5-G8 for the CQUAD8 element. (Integer > 0, or blank. If any of G5-G8 are not blank, all nodes in the range G5 to G8 must be defined and must be unique among the range G1-G8.)

Remarks:

1. This entry makes use of Marc's REBAR and INSERT capabilities for membrane element types 147 and 148. If G5-G8 are blank, Marc element 147 with 4 nodes is used. If G5-G8 are not blank, Marc element 148 with 4 corner nodes and 4 mid-side nodes is used. Entry CMREBAR makes use of Marc's REBAR capability (without INSERT) and uses rebar elements 23 and 146.
2. The grid ID's do not have to correspond to those of any matrix CHEXA element. The PMREABI entry is used to describe the matrix CHEXA elements that these rebar elements will be inserted into.
3. Cord-reinforced composites are characterized by a group of reinforcing cords with arbitrary spatial orientations embedded in various matrix materials. The different constituents may have different mechanical properties. Two typical examples of the cord-reinforces composites are tires and cord-reinforced concretes. In modeling such materials, the rebar technique is very useful. The basic idea

of rebar layer concept contains (1) the reinforcing cords and the matrix materials of the composites are represented independently by different types of elements along with different constitutive models, (2) the reinforcing cords within the elements modeling these cords (the so-called rebar elements) are assumed to be in the form of layers, and (3) the rebar elements are then embedded into the matrix elements. The compatibility between the cord elements and the matrix elements is enforced by embedding membrane rebar elements into solid matrix elements using Marc's INSERT option. Membrane rebar elements types 147 and 148 are available with this option. They are empty 4-node or 8-node quadrilaterals. You can place reinforcing cord layers within these empty elements. These elements are then embedded into their corresponding solid elements representing the matrix materials. Independent meshes can be used for the rebar membrane elements and the matrix elements. Marc's INSERT option is automatically invoked by the CMREBAI elements and used to enforce the compatibility between two different meshes.

4. The major difference between the CMREBAR and CMREBAI elements is that CMREBAR elements share the same grids as the matrix CHEXA elements while CMREBAI elements typically have different grid ID's than the matrix CHEXA elements. Marc's INSERT option automatically adds tyings (MPC's) between the CMREBAI grids and the CHEXA grids.
5. CMREBAI elements are preferred over the CMREBAR elements when re-meshing is involved.
6. See MPREBAI for additional information and figures defining these rebar elements.
7. Only CHEXA elements may be used for the matrix elements.

CMREBAR

Defines Rebar Elements and Matching "Matrix" Solid Elements using the Marc REBAR without INSERT Method in SOL 600

In some cases, particularly for modeling of concrete or tires, it is beneficial to add rebar or cord material to a matrix. The resulting combined material is similar to a composite but it is sometimes easier to postprocess the stresses of the rebar and matrix separately to determine failure conditions. CMREBAR elements require that the rebar be placed in matching CHEXA matrix elements on a one-to-one basis. For a similar capability where the rebar can span multiple CEHXA matrix elements, see the CMREBAI entry.

Format:

1	2	3	4	5	6	7	8	9	10
CMREBAR	ID	IP	ID2	IDD	ID22				

Example:

CMREBAR	100	2	1						
CMREBAR	1001	50	101	1100	200				

Descriptor	Meaning
EID	Rebar element ID should be distinct between all element ID's. (Integer; Required; no Default)
IP	Property identification of a matching PMRBAR entry. (Integer; Required; no Default)
ID2	CHEXA (8 node or 20 node) "matrix" element that the rebar will be added to. (Integer; Required; no Default)
IDD	If more than one rebar element in a continuous range is to be added to a continuous range of rebar elements, IDD represents the first rebar element identification number in the range. (Integer; Required; Default = 0, IDD must be larger than ID)
ID22	If more than one rebar element in a continuous range is to be added to a continuous range of "matrix" elements, ID22 represents the last CHEXA matrix element identification number in the range. (Integer; Required; Default = 0, ID22 must be larger than ID2)

Remarks:

1. This entry makes use of Marc's REBAR capability for element types 23 and 146. Entry CMREBAI makes use of Marc's REBAR and INSERT capabilities and uses membrane rebar elements 147 and 148.
2. If IDD is entered, ID22 must also be entered and the difference IDD-ID must match the difference ID22-ID2.

3. The same grid ID's are used to define element ID and ID2. Similarly the same grid ID's define the other elements if a range of elements are used. Element ID is an "empty shell element" and up to 5 rebar layers (each containing multiple rebar) are placed in the empty shell as specified by the PMREBAR property entries.
4. Cord-reinforced composites are characterized by a group of reinforcing cords with arbitrary spatial orientations embedded in various matrix materials. The different constituents may have different mechanical properties. Two typical examples of the cord-reinforces composites are tires and cord-reinforced concretes. In modeling such materials, the rebar technique is very useful. The basic idea of rebar layer concept contains that (1) the reinforcing cords and the matrix materials of the composites are represented independently by different types of elements along with different constitutive models, (2) the reinforcing cords within the elements modeling these cords (the so-called rebar elements) are assumed to be in the form of layers, and (3) the rebar elements are then embedded into the matrix elements. The compatibility between the cord elements and the matrix elements is enforced by superimposing solid rebar elements on corresponding solid matrix elements using the same element connectivity. The rebar elements are empty 8-node or 20-node CHEXA elements derived from the matching matrix elements. The reinforcing cord layers are placed within the elements. Each solid rebar element is then superimposed on a solid matrix element. The two elements share the same space with the same element connectivity (therefore, the same element nodes). The compatibility condition between the reinforcements and the matrix materials is then automatically enforced.
5. The major difference between the CMREBAR and CMREBAI elements is that CMREBAR elements share the same grids as the matrix CHEXA elements while CMREBAI elements typically have different grid ID's than the matrix CHEXA elements. Marc's INSER T option automatically adds tyings (MPC's) between the CMREBAI grids and the CHEXA grids.
6. CMREBAI elements are preferred over the CMREBAR elements when re-meshing is involved.
7. See PMREBAR for additional information and figures defining these rebar elements.
8. Only CHEXA elements may be used for the matrix elements.

Entries CO - CY

COHESIV

Defines Data for Cohesive Materials in SOL 600

This option allows you to define material properties for interface elements, that may be used to simulate the onset or progress of delamination, and to associate these material properties with a list of element numbers. The cohesive material is defined using the cohesive energy (also called critical energy release rate), that equals the area below the equivalent traction versus equivalent relative displacement curve. The shape of this curve can be bilinear, exponential, or combined linear-exponential. Mixed mode delamination is incorporated by converting the normal and shear components of the relative displacements into an equivalent using the normal shear weighting factor.

As an alternative to the standard linear, exponential, and liner-exponential model, the user can also utilize this option to trigger the call to the UCOHESIVE user-subroutine. Used in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
COHESIV	MID	ITYPE	IACT	NAME					
	EN	COD	MOD	NSW	SNW	DECAY		'	
	VISC	RATE	STIFF						
	ISET								

Example:

COHESIV	101	1	1	material	number10	1			
	2000.	.01	.015	23.	24.	1.			
	.05	0.0	1.0						
	101								

Descriptor	Meaning
MID (3,1)	Material ID - Must match a MATXXX entry. (Integer; no Default)
ITYPE (3,2)	Type of cohesive model. (Integer > 0; Default = 1) <ul style="list-style-type: none"> 1 Bilinear model (Default) 2 Exponential modes 3 Combined linear-exponential model -1 User-defined using user-subroutine UCOHESIVE
IACT (3,3)	Option to deactivate elements and output to t16 file. (Integer; Default = 0) <ul style="list-style-type: none"> 0 Elements remain active regardless of the damage level. 1 Deactivate the elements if the maximum damage in all the element integration points has been reached. Do not remove the elements from the t16 file.

Descriptor	Meaning
2	Deactivate the elements if the maximum damage in all the element integration points has been reached. Remove the elements from the t16 file.
NAME (3,5)	Name of the material. (Character up to 40 characters; no Default; optional entry)
EN (4,1)	Cohesive energy. (Real; no Default; Required value)
COD (4,2)	Critical opening displacement. (Real; no Default; Required value)
MOD (4,3)	Maximum opening displacement (Bilinear model only). (Real; no Default)
NSW (4,4)	Normal-shear weighting factor, beta. (Real; no Default; Required value)
SNW (4,5)	Shear-normal weighting factor. (Real; Default = 1.0)
DECAY (4,6)	Exponential decay factor. (Linear-Exponential model only).(Real; Default = 1.0)
VISC (4,7)	Viscous energy dissipation factor (zero implies no viscous energy dissipation). (Real; Default = 0.0)
RATE (4,8)	Relative displacement reference rate. Only used if viscous energy dissipation is by setting VISC to a non-zero value. (Real; Default = 0.0)
STIFF (4,9)	Stiffening factor in compression. (Real; Default = 1.0)
ISET (5)	ID of a SET3 entry defining the elements associated with this cohesive material. (Integer, no Default, Required value)

Remarks:

1. Values in parenthesis (i,j) refer to Marc's COHESIVE entry. (Datablock, field)
2. For solid elements, this entry may be used to add a "layer" or interface between the solid elements (CHEXA, CTETRA or CPENTA). This interface can fail or delaminate depending on the properties entered. It is used in conjunction with Marc element types 188, 192 and 193 and provides the material properties for these element types.
3. For shell elements (CQUADi, CTRIAi) this entry may be used to add a layer between the edges of adjacent shells. It is used in conjunction with Marc element types 186 and 187 and provides the material properties for these element types.
4. Solid or shell elements with a COHESIV MID will automatically be assigned Marc element types 186, 187, 188, 189, 192, 193 as appropriate to the type and number of grids defined for that element. These solid elements must either only have corner nodes or have the full parabolic number of nodes (for example, CHEXA must either have 8 nodes or 20 nodes).
5. For SOL 600, cohesive behavior is not available for axisymmetric or plane strain analyses.
6. MID must not be used by any other material such as MAT1, MAT2, etc.
7. All continuation lines are required.

COHFRIC**Cohesive Friction**

Allows friction and sticking during tensile conditions at the coupling surface. Use SOL700 only.

Format:

COHFRIC	CID	MAXSTRS	FRIC	REFVEL				
---------	-----	---------	------	--------	--	--	--	--

Example:

COHFRIC	112	8e+10	8e+5	2				
---------	-----	-------	------	---	--	--	--	--

Descriptor	Meaning
CID	Unique number of a COHFRIC entry. (Integer>0; required)
MAXSTRS	Maximal normal stress. Allows tensile stresses at the coupling surface as long as the normal stress does not exceed MAXSTRS. (Real ≥ 0.0 ; default=0.0)
FRIC	Friction stress under tensile conditions. (Real ≥ 0.0 ; default=0.0)
REFVEL	Reference value for velocity. (Real ≥ 0.0 ; default=0.0)

Remarks:

1. If the cohesive friction parameters is uniform across the coupling surface then DYPARAM, COHESION can be used instead.
2. During tension any relative tangential velocity between coupling surface and Eulerian material will yield a shear stress whose magnitude equals $Fric \times \min\left(1, \frac{V_{REL, \text{tangential}}}{REFVEL}\right)$. This is a viscous-like friction law.
3. This shear force opposes the relative tangential movement along the coupling surface.

COMPUDS

Orthotropic Failure Model that allows to model Property Degradation

Defines an orthotropic failure model for shell composites specified by a user subroutine. Used in SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
COMPUDS	MID	GROUP	UNAME						+
+	E1	E2	N12	G12	G1,Z	G2,Z	RHO		+
+	NV	S	ALPHA	TRSFAIL	F12				+
+	XT	XT	YT	YC	PFD	VALUE	PFDST		+
+	FBTEN	FBCOM	MXTEN	MXCOM	MXSHR				+
+	PRDFT	PRDFC	PRDMT	PRDMC	PRDSH				+

Example:

In FMS Section of the MSC Nastran input stream:

```
CONNECT SERVICE excomp1 'SCA.MDSolver.Obj.Uds.Dytran.Materials'
```

In Bulk Data:

COMPUDS	1	EXCOMP1							+
+	30E6	1.0E6	0.3	2.0E6	3.0E6	1.5E6	0.056		+
+	5	100							
+	200	150	100	110					

Field	Contents
MID	Unique material identification number. (Integer > 0; required)
GROUP	The group name used for the FMS section CONNECT SERVICE statement. (Character; required).
UNAME	User subroutine name associated with the entry. (Character; default=blank).
E1	Modulus of elasticity in longitudinal direction (also defined as fiber direction or one-direction). (Real > 0.0; required)
E2	Modulus of elasticity in lateral direction (also defined as matrix direction or two-direction). (Real > 0.0; required)

Field	Contents
N12	Poisson's ratio (2/ 1 for uniaxial loading in one-direction). Note that $21 = 1/ 2$ for uniaxial loading in two-direction is related to $12, E1, E2$ by the relation $12 E2 = 21 E1$. (Real > 0.0; required)
G12	In-plane shear modulus R. (Real > 0.0; required)
G1,Z	Transverse shear modulus for shear in 1-Z plane. (Real > 0.0; default= G12)
RHO	Mass density. (R > 0.0; required)
NV	Number of additional history variables for a user model. See Remark 8. (0 < Integer < 10; default=0)
S	Failure stress for in-plane shear. (Real \geq 0.0; default=0.0)
ALPHA	Nonlinear shear coefficient. (Real \geq 0.0; default=0.0)
TRSFAIL	Transverse shear failure. (Character; default=SUBL) ELEM: Failure if element fails SUBL: Failure if sublayer fails
F12	Interaction term in Tsai-Wu theory. (Real; default=0.0)
XT, XC	Tensile compressive failure stress in the large structural direction. (Real > 0.0; required)
YT, YC	Tensile compressive failure stress in the lateral direction. (Real > 0.0; required)
PFD	Post-failure degradation model. (Character; default=STEPS) STEPS: Degrade stresses by time steps TIME: Degrade stresses by time VELOC: Degrade stresses by velocity
VALUE	Depending on PFD, VALUE gives the number of time steps, time interval, or propagation velocity. (Integer or Real; default=100)
PFDS	Post-failure degradation start. (Character; default=INDV) INDV Stresses are degraded per distinct failure mode. ALL Stresses are degraded if all elastic constants are zero.
FBTEN,FBCO	Failure modes in fiber, matrix direction, and theory failure. (Character; default=blank)
N,MXTEN,M	
XCOM,MXS	
HR	

Field	Contents
PRDFT	Property degradation due to fiber-tension failure. (Integer; default=1111)
PRDFC	Property degradation due to fiber-compression failure. (Integer; default=1010)
PRDMT	Property degradation due to matrix-tension failure. (Integer; default=0110)
PRDMC	Property degradation due to matrix-compression failure. (Integer; default=0110)
PRDSH	Property degradation due to in-plane shear failure. (Integer; default=0001)

Remarks:

11. Please check MAT8 and MAT8A cards about the details of each field. The field definition is identical as MAT8 and MAT8A.
12. UNAME can be:

Subroutine Name	Function
EXCOMP	User defined orthotropic material models for shell composite

CONCTL

Parameter SWLDPRM override for CFAST, CSEAM, and CWELD Connector Elements

This entry provides local connector search algorithm control to override SWLDPRM values.

Format:

1	2	3	4	5	6	7	8	9	10
CONCTL	SET3ID		PARAM1	VALUE1	PARAM2	VALUE2	PARAM3	VALUE3	
	PARAM4	VALUE5	PARAM4	VALUE5	-etc.-				

Example:

CONCTL	75		PROJTOL	0.2	PRTSW	1	GSMOVE	3	
	NREDIA	2							

CONCTL	83		MOVGAB	1					
--------	----	--	--------	---	--	--	--	--	--

Descriptor	Meaning
SET3ID	The ID of a SET3 entry using (DES= ELEM) to identify a group of connector elements. (Integer>0)
PARAMi	Name of the connector parameter. Allowable names are listed in Table 8-62 of SWLDPRM. (Character)
VALUEi	Value of the parameter. See Table 8-62 of SWLDPRM (Real or Integer)

Remarks:

1. Multiple CONCTL entries are allowed and ALL appearing will be used. If the user supplies a PARAMi, VALUEi that refers to specific Element on one CONCTL entry and a same PARAMi with VALUEj that refers to the same specific Element on another CONCTL, a fatal message will be issued.
2. For a given connector element ID, this entry will override the global default set by the SWLDPRM entry.
3. The parameter CHKRUN can only be set on SWLDPRM and will be ignored on CONCTL entries.
4. A user fatal will be issued if DES on the SET3 entry is not "ELEM".

CONM1

Concentrated Mass Element Connection, General Form

Defines a 6 x 6 symmetric mass matrix at a geometric grid point.

Format:

1	2	3	4	5	6	7	8	9	10
CONM1	EID	G	CID	M11	M21	M22	M31	M32	
	M33	M41	M42	M43	M44	M51	M52	M53	
	M54	M55	M61	M62	M63	M64	M65	M66	

Example:

CONM1	2	22	2	2.9	6.3				
	4.8	28.6							
		28.6							28.6

Descriptor	Meaning
EID	Unique element identification number. ($0 < \text{Integer} < 100,000,000$). See Remark 2.
G	Grid point identification number. ($\text{Integer} > 0$). See Remark 3.
CID	Coordinate system identification number for the mass matrix. ($\text{Integer} \geq 0$)
Mij	Mass matrix values. (Real)

Remarks:

- For a less general means of defining concentrated mass at grid points, see the CONM2 entry description.
- Element identification numbers should be unique with respect to all other element identification numbers.
- Unlike the CONM2 entry, the CONM1 entry does not allow for the specification of concentrated mass at harmonic structural grid points. Therefore, grid point G must necessarily be a non-harmonic structural grid point.

CONM2

Concentrated Mass Element Connection, Rigid Body Form

Defines a concentrated mass at a grid point.

Format:

1	2	3	4	5	6	7	8	9	10
CONM2	EID	G	CID	M	X1	X2	X3		
	I11	I21	I22	I31	I32	I33			

Example:

CONM2	2	15	6	49.7					
	16.2		16.2			7.8			

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000). See Remark 1.
G	Grid point identification number. (Integer > 0). See Remark 7.
CID	Coordinate system identification number. For CID of -1; see X1, X2, X3 below. (Integer \geq -1; Default = 0). See Remarks 4., 6. and 7.
M	Mass value. (Real). See Remarks 5. and 7.
X1, X2, X3	Offset distances from the grid point to the center of gravity of the mass in the coordinate system defined in field 4, unless CID = -1, in which case X1, X2, X3 are the coordinates, not offsets, of the center of gravity of the mass in the basic coordinate system. (Real). See Remarks 4., 5., 6. and 7.
Iij	Mass moments of inertia measured at the mass center of gravity in the coordinate system defined by field 4. If CID = -1, the basic coordinate system is implied. (Real). See Remarks 4., 5. and 7.

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. For a more general means of defining concentrated mass at grid points, see the CONM1 entry description.
3. The continuation is optional.
4. If CID = -1, offsets are internally computed as the difference between the grid point location and X1, X2, X3. The grid point locations may be defined in a nonbasic coordinate system. In this case, the values of Iij must be in a coordinate system that parallels the basic coordinate system.

5. The form of the inertia matrix about its center of gravity is taken as:

$$\begin{bmatrix} M & & & \\ M & M & & \text{symmetric} \\ M & & M & \\ & I_{11} & & \\ & -I_{21} & I_{22} & \\ & -I_{31} & -I_{32} & I_{33} \end{bmatrix}$$

where

$$\begin{aligned} M &= \int \rho dV \\ I_{11} &= \int \rho(x_2^2 + x_3^2) dV \\ I_{22} &= \int \rho(x_1^2 + x_3^2) dV \\ I_{33} &= \int \rho(x_1^2 + x_2^2) dV \\ I_{21} &= \int \rho x_1 x_2 dV \\ I_{31} &= \int \rho x_1 x_3 dV \\ I_{32} &= \int \rho x_2 x_3 dV \end{aligned}$$

and x_1, x_2, x_3 are components of distance from the center of gravity in the coordinate system defined in field 4. The negative signs for the off-diagonal terms are supplied automatically. A warning message is issued if the inertia matrix is nonpositive definite, since this may cause fatal errors in dynamic analysis modules.

6. If CID ≥ 0 , then X1, X2, and X3 are defined by a local Cartesian system, even if CID references a spherical or cylindrical coordinate system. This is similar to the manner in which displacement coordinate systems are defined.
7. MSC Nastran contains two kinds of structural grid points, namely, non-harmonic structural grid points and harmonic structural grid points. The former have the standard three translational and three rotational degrees of freedom while the latter have three symmetric components and three anti-symmetric components dependent on the harmonic value.

Harmonic grid points are identified automatically as the grid points listed on CQUADX and CTRIAX element entries that are associated with PAXSYMH entries.

For a harmonic grid point, the mass value M is the total mass. This value is not to be multiplied by 2π . The mass matrix computed for harmonic grids is dependent on the harmonic value associated with the PAXSYMH entry and will be automatically determined by the code. Only harmonic values of 0 and 1 have contributions to grid point weight generator type calculations.

Any values specified in the CID, X1, X2, X3, I11, I21, I22, I31, I32, or I33 fields are ignored for a harmonic grid.

CONROD

Rod Element Property and Connection

Defines a rod element without reference to a property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CONROD	EID	G1	G2	MID	A	J	C	NSM	

Example:

CONROD	2	16	17	4	2.69				
--------	---	----	----	---	------	--	--	--	--

Descriptor	Meaning
EID	Unique element identification number. (0 < Integer < 100,000,000)
G1, G2	Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2)
MID	Material identification number. (Integer > 0)
A	Area of the rod. (Real)
J	Torsional constant. (Real)
C	Coefficient for torsional stress determination. (Real)
NSM	Nonstructural mass per unit length. (Real)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. For structural problems, MID must reference a MAT1 material entry.
3. For heat transfer problems, MID must reference a MAT4 or MAT5 material entry.
4. For RC network solver in thermal analysis, the J, C and NSM are ignored.

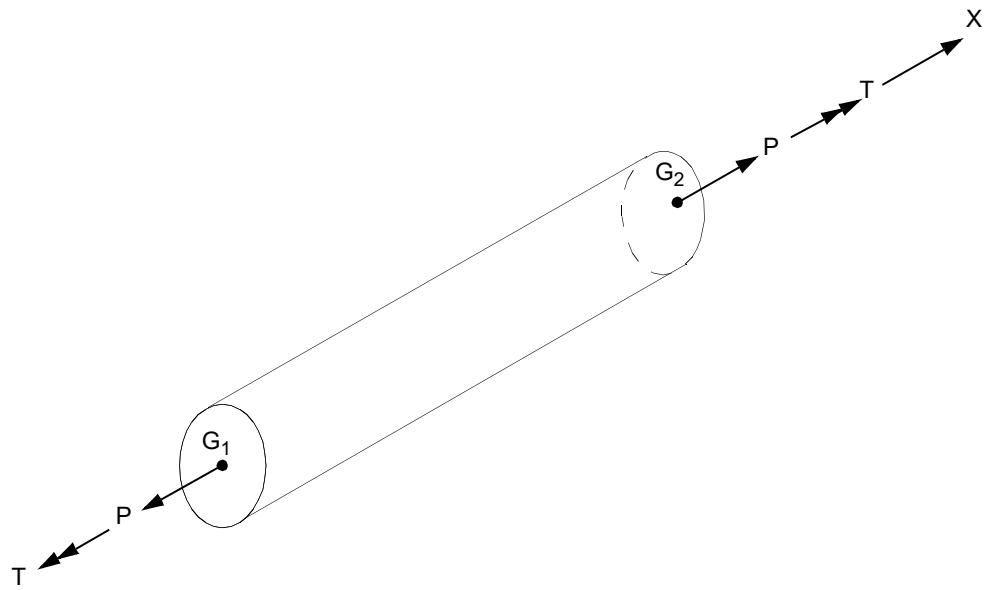


Figure 9-39 CONROD Element Forces and Moments

CONTRLT

Thermal Control Element for Heat Transfer Analysis

Defines the control mechanism for QVECT, QVOL, QBDY3, RADBC, CONV, and CONVM in heat transfer analysis (SOL 159 and SOL 400 with analysis=htran).

Format:

1	2	3	4	5	6	7	8	9	10
CONTRLT	ID	SENSOR	SFORM	CTYPE	PL	PH	PTYPE	PZERO	
	DT	DELAY	TAUC				FCTMIN	FCTMAX	
	GAIN1	GAIN2	GAIN3						

Example:

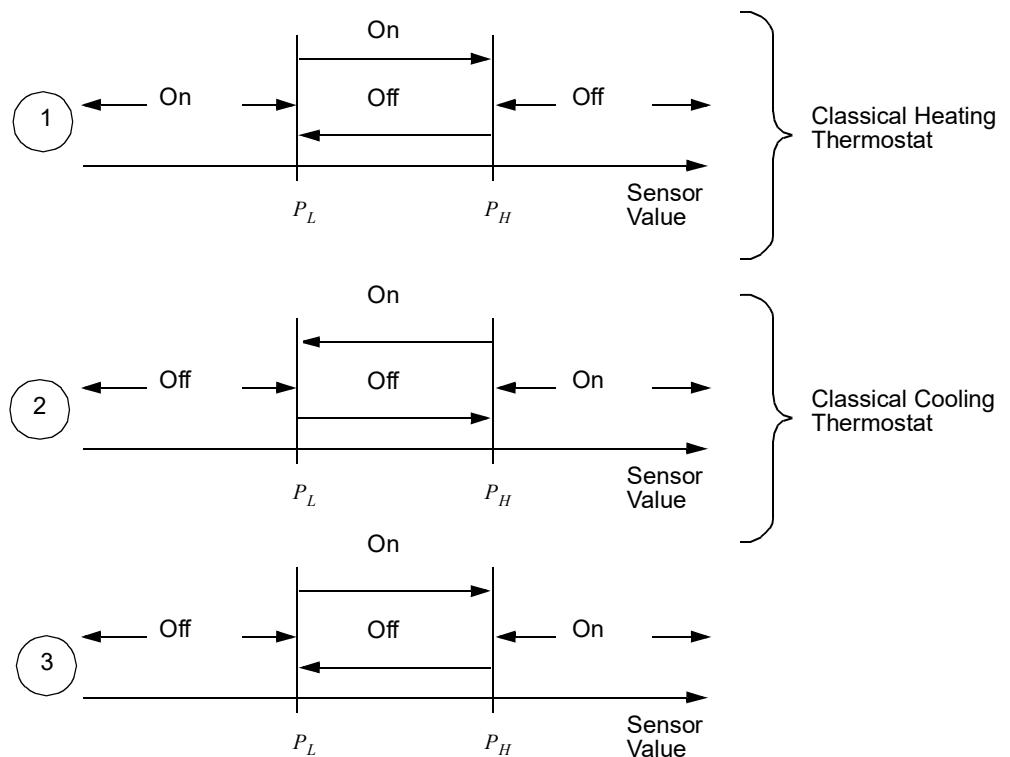
CONTRLT	100	20	T	TSTAT	68.	73.	7	1.	
	0.1	0.01	0.				0.	1.	
	0.	0.	0.						

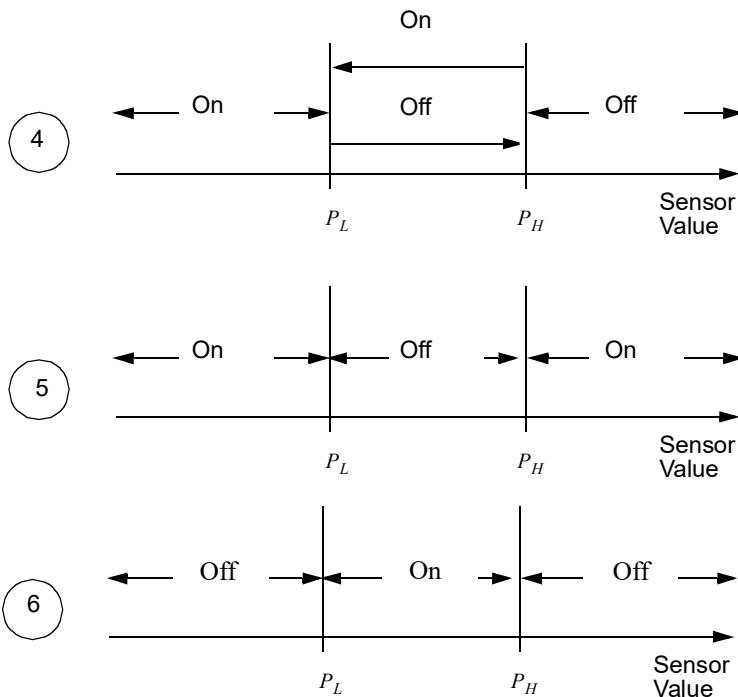
Descriptor	Meaning	
ID	Controller identification node. See Remark 1. (Integer > 0; Required)	
SENSOR	Grid or scalar point ID of sensor. See Remark 2. (Integer > 0; Required)	
SFORM	Sensor output form. See Remark 3. (Character, T; Default = T)	
CTYPE	Control type. See Remark 4. (Character, Default TSTAT)	
PL	Minimum temperature. For PTYPE 9 through 11, set temperature. See Remark 5. (Real; Required)	
PH	Maximum temperature. For PTYPE 9 through 11, unused. See Remark 5. (Real; Required)	
PTYPE	Process type. See Remark 5. (Integer 1 through 11, 1 through 6 already defined; no Default)	
	7	Bang-bang (a thermostat that is either on or off, a special case of a bang-bang controller)
	8	Proportional
	9	Steady-state (special routine for steady state models)
	10	Classical PID (proportional-integral-differential) and does not include the time derivative of the set point)
	11	Standard PID (Includes the time derivative of the set point)
PZERO	Initial controller value. See Remark 4. (0. < Real < 1.; Default = 0)	
DT	Monitoring time interval or sampling period. (Real > 0.; Default = 0)	

Descriptor	Meaning
DELAY	Time delay after the switch is triggered or time for delayed control action in PID control. (Real > 0.; Default = 0.0)
TAUC	Decay time constant for actuator response. (Real > 0.; Default = 0.0)
FCTMIN/MAX	Minimum factor/Maximum factor on load to apply. (Real; Default = 0.0)
GAINx	Gain variables for heat controller. Use GAIN1 for P value, GAIN2 for I value, and GAIN3 for D value. (Real; Default = 0.0)

Remarks:

1. The CONTRLT ID is referenced by CNTRLND entry identified on any of the CONV, CONVM, RADBC, QVECT, QVOL, and QBDY3 Bulk Data entries. If any grid or scalar point ID is the same as the CONTRLT ID, then the combined logic associated with the controller and the control node will be in force for the LBC referenced. Any number of CONTRLT statements may exist in a single model.
2. Sensor point, where a feedback temperature or rate of change of temperature is measured. May be a dependent DOF in a MPC relationship.
3. Sensor output may only be temperature (T)
4. Control type can only be TSTAT. The PZERO field cannot have any other value but 0.0 or 1.0.
5. The upper and lower limit values (PL and PH) define a dead band for a thermostat. The available thermostat controller (TSTAT) formats are (PTYPE = 1 through 6).
6. RC network solver only supports PTTYPE=1 and PTTYPE=2 thermostat controller, SFORM, CTYPE, PZERO, DT, DELAY, TAUC and TA8 are ignored.





7. The last 5 parameters are for RC Network solver only.
8. RC Network solver only supports controllers (thermostats) on source data. This entry can be used in either steady-state (static) or transient state (dynamic). For more details about these parameters, please reference the *MSC SINDA User's Guide and Library Reference*.

CONV**Heat Boundary Element Free Convection Entry**

Specifies a free convection boundary condition for heat transfer analysis through connection to a surface element (CHBDYi entry).

Format:

1	2	3	4	5	6	7	8	9	10
CONV	EID	PCONID	FLMND	CNTRLND	TA1	TA2	TA3	TA4	
	TA5	TA6	TA7	TA8					

Example:

CONV	2	101	3	201	301				
------	---	-----	---	-----	-----	--	--	--	--

Descriptor	Meaning
EID	CHBDYG, CHBDYE, or CHBDYP surface element identification number. (0 < Integer < 100,000,000)
PCONID	Convection property identification number of a PCONV entry. (Integer > 0)
FLMND	Point for film convection fluid property temperature. (Integer ≥ 0; Default = 0)
CNTRLND	Control point for free convection boundary condition. (Integer ≥ 0; Default = 0)
TAi	Ambient points used for convection. (Integer > 0 for TA1 and Integer ≥ 0 for TA2 through TA8; Default for TA2 through TA8 is TA1.)

Remarks:

- The basic exchange relationship can be expressed in one of the following forms:
 - $q = H \cdot (T - TAMB)^{EXPF} (T - TAMB)$, CNTRLND = 0
 - $q = (H \cdot u_{CNTRLND})(T - TAMB)^{EXPF} (T - TAMB)$, CNTRLND ≠ 0
 - $q = H(T^{EXPF} - TAMB^{EXPF})$, CNTRLND = 0
 - $q = (H \cdot u_{CNTRLND})(T^{EXPF} - TAMB^{EXPF})$, CNTRLND ≠ 0
 EXPF is specified on the PCONV entry.
(See [PCONV, 2801](#) entry for additional clarification of forms.)
- The continuation entry is not required.
- CONV is used with an CHBDYi (CHBDYG, CHBDYE, or CHBDYP) entry having the same EID.
- The temperature of the film convection point provides the look up temperature to determine the convection film coefficient. If FLMND=0, the reference temperature has several options. It can be the average of surface and ambient temperatures, the surface temperature, or the ambient temperature, as defined in the FORM field of the PCONV Bulk Data entry.

5. If only one ambient point is specified then all the ambient points are assumed to have the same temperature. If midside ambient points are missing, the temperature of these points is assumed to be the average of the connecting corner points.
6. See the Bulk Data entry, [PConv, 2801](#), for an explanation of the mathematical relationships involved in free convection and the reference temperature for convection film coefficient.
7. RC network solver only supports CNTRLND node defined by SPC or SPCD entries, FLMND is ignored.

CONVM**Heat Boundary Element Forced Convection Entry**

Specifies a forced convection boundary condition for heat transfer analysis through connection to a surface element (CHBDYi entry).

Format:

1	2	3	4	5	6	7	8	9	10
CONVM	EID	PCONID	FLMND	CNTMDOT	TA1	TA2	Mdot		

Example:

CONVM	101	1	201	301	20	21			
-------	-----	---	-----	-----	----	----	--	--	--

Descriptor	Meaning
EID	CHBDYP element identification number. (0 < Integer < 100,000,000)
PCONID	Convection property identification number of a PCONVM entry. (Integer > 0)
FLMND	Point used for fluid film temperature. (Integer ≥ 0 ; Default = 0)
CNTMDOT	Control point used for controlling mass flow. (Integer ≥ 0 or Blank). Blank or zero is only allowed when Mdot > 0.0. See Remark 3.
TA1, TA2	Ambient points used for convection. (Integer > 0 for TA1 and Integer ≥ 0 for TA2; Default for TA2 is TA1.)
Mdot	A multiplier for mass flow rate in case there is no point associated with the CNTRLND field. (Real > 0.0; Default = 1.0 if CNTMDOT > 0). See Remark 3.)

Remarks:

1. CONVM is used with an CHBDYP entry of type FTUBE having the same EID.
2. The temperature of the fluid film point may be specified to determine the material properties for the fluid. If FLMND=0, the reference temperature has several options. It can be the average of surface and ambient temperatures, the surface temperatures, or the ambient temperature, as defined in the FORM field of the PCONVM Bulk Data entry.
3. The CNTMDOT has a dual function. It can reference the ID of the CONTRLT Bulk Data entry to activate a thermostat controlled flux transfer. It can also reference a GRID or a scalar point which is set to the desired mass flow rate (mdot) to effect the advection of energy downstream at an $mdot \cdot Cp \cdot T$ rate. If CNTMDOT is zero or is not specified (blank field), then the mass flow rate must be specified on the Mdot field (8th field).

Case 1: Define a thermostat controller using the CONTRLT Bulk Data entry.

Total mass flow rate = $\text{CONTRLT} \cdot \text{Mdot} \cdot \text{CNTMDOT}$

where CONTRLT is the grid ID on the 2nd field of the CONTRLT Bulk Data entry that ties the mass flow rate through this sensor grid.

Mdot is the 8th field of the CONVM Bulk Data entry.

CNTMDOT is the grid ID that has a mass flow rate associated with it.

Example: The thermostat controller controls GRID 5 (sensor grid) from 250 degrees to 300 degrees. If the temperature exceeds 300 degrees, then the mass flow is turned ON. If the temperature is below 250 degrees, then the mass flow is turned OFF.

```
CONTRLT, 5, 2, T, TSTAT, 250.0, 300.0, 1, 1.0 +
+, 0.0, 0.0, 0.0
CONVM, 100001, 1, , 5, 4
```

Note that GRID 5 is also specified on the 5th field of the CONVM Bulk Data entry.

Case 2: There is no CONTRLT Bulk Data entry (more general case).

Total mass flow rate = $\text{Mdot} \cdot \text{CNTMDOT}$

If the CNTMDOT (the 5th field of the CONVM Bulk Data entry) is specified, then the default Mdot value is 1.0 and the mass flow rate is defined through CNTMDOT .

Example: Define a constant mass flow rate of 0.5 through CNTMDOT .

```
CONVM, 100001, 1, , 5, 4
SPC, 1, 5, 1, 0.5
```

To specify a time-varying mass flow rate, replace the SPC Bulk Data entry by TEMPBC,TRAN for SOL 159 or by SPC1 and SPCD for SOL 400.

For SOL 159:

```
TEMPBC, 2, TRAN, 1.0, 5
DLOAD, 700, 1.0, 1.0, 9
TLOAD1, 9, 2, , , 121
TABLED1, 121
, 0.0, 0.3, 100.0, 0.5, endt
```

For SOL 400:

```
SPC = 111 (Case Control command)
SPC1, 111, , 5
SPCD, 2, 5, , 1.0
DLOAD, 700, 1.0, 1.0, 9
TLOAD1, 9, 2, , 1, 121
TABLED1, 121
, 0.0, 0.3, 100.0, 0.5, endt
```

If the CNTMDOT is not specified (zero or blank), then the mass flow rate must be constant and must be defined on Mdot (the 8th field of the CONVM Bulk Data entry).

Example: Define a mass flow rate of 0.5 directly on Mdot without specifying CNTMDOT .

CONVM, 100001, 1,,, 99,, 0.5

4. If only the first ambient point is specified then, the second ambient point is assumed to have the same temperature.
5. See the Bulk Data entry, [PCONVM, 2804](#), for an explanation of the mathematical relationships available for forced convection and the reference temperature for fluid material properties.
6. RC network solver does not support CONVM for thermal analysis, it uses PRJCON for forced convection.

CORD1C**Cylindrical Coordinate System Definition, Form 1**

Defines a cylindrical coordinate system using three grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CORD1C	CIDA	G1A	G2A	G3A	CIDB	G1B	G2B	G3B	

Example:

CORD1C	3	16	32	19					
--------	---	----	----	----	--	--	--	--	--

Descriptor	Meaning
CIDA, CIDB	Coordinate system identification number. (Integer > 0)
GiA, GiB	Grid point identification numbers. (Integer > 0; G1A ≠ G2A ≠ G3A ; G1B ≠ G2B ≠ G3B)

Remarks:

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, CORD2S and CORD3G entries must be unique.
2. One or two coordinate systems may be defined on a single entry.
3. GiA and GiB must be defined in coordinate systems with definitions that do not involve the coordinate system being defined. The first point is the origin, the second lies on the z-axis, and the third lies in the plane of the azimuthal origin. The three grid points GiA (or GiB) must be noncolinear and not coincident.
4. The location of a grid point (P in Figure 9-40) in this coordinate system is given by (R, θ, Z) where θ is measured in degrees.

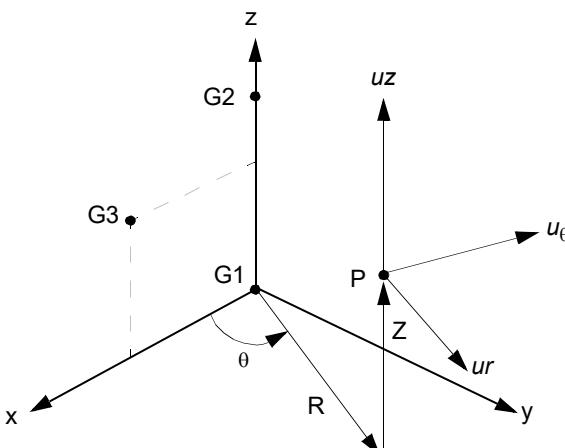


Figure 9-40 CORD1C Definition

5. The displacement coordinate directions at P are dependent on the location of P as shown above by (u_r, u_θ, u_z) .
6. It is recommended that points on the z-axis not have their displacement directions defined in this coordinate system. See the discussion of cylindrical coordinate systems in [Grid Point and Coordinate System Definition](#) (p. 34) in the *MSC Nastran Reference Guide*.

If CYLINDRICAL and point exactly on z-axis ($r = 0.0$ exactly) then R, θ , Z triad coordinates are exactly aligned with the local defining system triad. User caution: if $r \approx 0.0$ but not exactly 0.0 should be avoided as unexpected direction cosine matrix may result.

CORD1R

Rectangular Coordinate System Definition, Form 1

Defines a rectangular coordinate system using three grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CORD1R	CIDA	G1A	G2A	G3A	CIDB	G1B	G2B	G3B	

Example:

CORD1R	3	16	32	19					
--------	---	----	----	----	--	--	--	--	--

Descriptor	Meaning
CIDA, CIDB	Coordinate system identification number. (Integer > 0)
GiA, GiB	Grid point identification numbers. (Integer > 0; G1A ≠ G2A ≠ G3A and G1B ≠ G2B ≠ G3B)

Remarks:

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, CORD2S and CORD3G entries must be unique.
2. One or two coordinate systems may be defined on a single entry.
3. GiA and GiB must be defined in coordinate systems with definitions that do not involve the coordinate system being defined. The first point is the origin, the second lies on the z-axis, and the third lies in the x-z plane. The three grid points GiA (or GiB) must be noncolinear and not coincident.
4. The location of a grid point (P in [Figure 9-41](#)) in this coordinate system is given by (X, Y, Z).

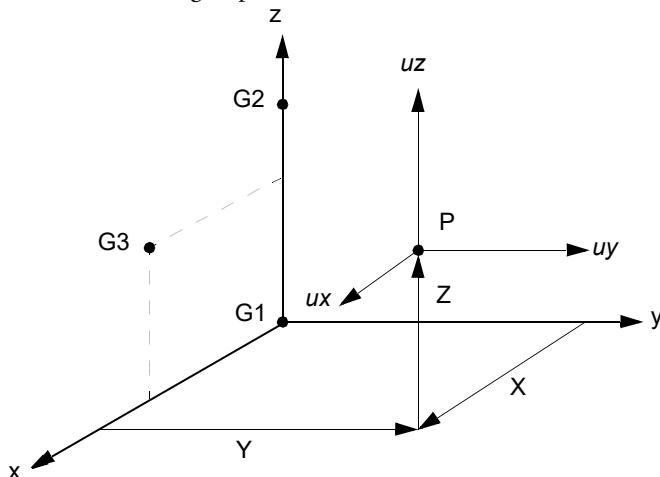


Figure 9-41 CORD1R Definition

5. The displacement coordinate directions at P are shown above by (u_x, u_y, u_z) .

CORD1S

Spherical Coordinate System Definition, Form 1

Defines a spherical coordinate system by reference to three grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CORD1S	CIDA	G1A	G2A	G3A	CIDB	G1B	G2B	G3B	

Example:

CORD1S	3	16	32	19					
--------	---	----	----	----	--	--	--	--	--

Descriptor	Meaning
CIDA, CIDB	Coordinate system identification numbers. (Integer > 0)
GiA, GiB	Grid point identification numbers. (Integer > 0; G1A ≠ G2A ≠ G3A and G1B ≠ G2B ≠ G3B)

Remarks:

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, CORD2S and CORD3G entries must be unique.
2. One or two coordinate systems may be defined on a single entry.
3. GiA and GiB must be defined in coordinate systems with a definition that does not involve the coordinate system being defined. The first point is the origin, the second lies on the z-axis, and the third lies in the plane of the azimuthal origin. The three grid points GiA (or GiB) must be noncolinear and not coincident.
4. The location of a grid point (P in Figure 9-42) in this coordinate system is given by (R, θ, ϕ) where θ and ϕ are measured in degrees.

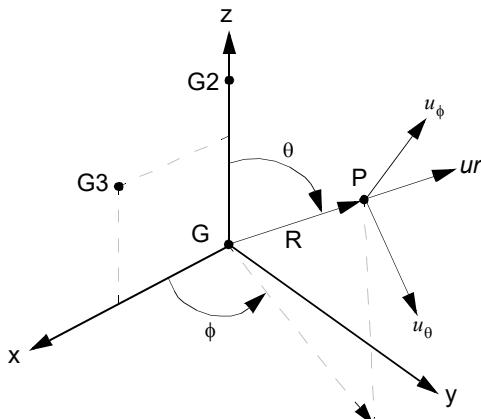


Figure 9-42 CORD1S Definition

5. The displacement coordinate directions at P are dependent on the location of P as shown above by (u_r, u_θ, u_ϕ) .
6. It is recommended that points on the z-axis not have their displacement directions defined in this coordinate system. See the discussion of spherical coordinate systems in [Grid Point and Coordinate System Definition](#) (p. 34) in the *MSC Nastran Reference Guide*.

If SPHERICAL and R = 0.0 (point at origin) then R, θ , ϕ triad coordinates are exactly aligned with the local defining system triad. User caution: if $R \approx 0.0$ but not exactly 0.0 should be avoided as unexpected direction cosine matrix may result.

If SPHERICAL and r in azimuth plane is $r = 0.0$ and $z \neq 0.0$ then the coordinate system is defined as R global along $\pm Z$ - local defining system triad according if $z > 0.$ or $z < 0.$, θ -global along X-local defining system triad and ϕ -global along Y-local defining system triad. User caution: if $r \approx 0.0$ but not exactly 0.0 should be avoided as unexpected direction cosine matrix may result.

CORD2C**Cylindrical Coordinate System Definition, Form 2**

Defines a cylindrical coordinate system using the coordinates of three points.

Format:

1	2	3	4	5	6	7	8	9	10
CORD2C	CID	RID	A1	A2	A3	B1	B2	B3	
	C1	C2	C3						

Example:

CORD2C	3	17	-2.9	1.0	0.0	3.6	0.0	1.0	
	5.2	1.0	-2.9						

Descriptor	Meaning
CID	Coordinate system identification number. (Integer > 0)
RID	Identification number of a coordinate system that is defined independently from this coordinate system. (Integer ≥ 0 ; Default = 0 is the basic coordinate system.)
Ai, Bi, Ci	Coordinates of three points given with respect to the coordinate system defined by RID. (Real)

Remarks:

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, CORD2S and CORD3G entries must be unique.
2. The three points [(A1, A2, A3), (B1, B2, B3), (C1, C2, C3)] must be unique and noncolinear. Noncolinearity is checked by the geometry processor. The first point defines the origin. The second point defines the direction of the z-axis. The third lies in the plane of the azimuthal origin. The reference coordinate system must be independently defined.
3. The continuation entry is required.
4. If RID is zero or blank, the basic coordinate system is used.

5. The location of a grid point (P in [Figure 9-43](#)) in this coordinate system is given by (R, θ, Z), where θ is measured in degrees.

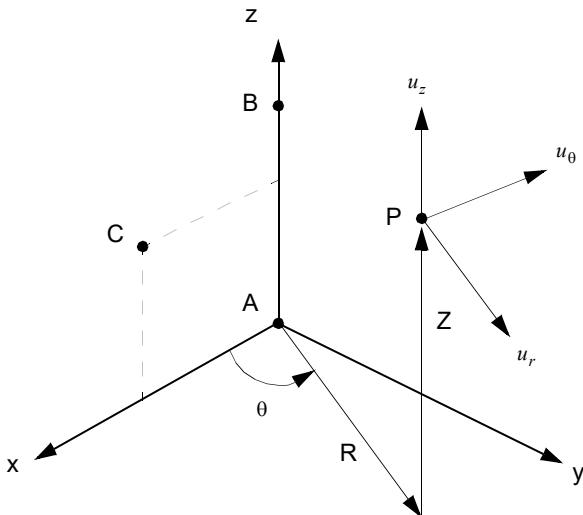


Figure 9-43 CORD2C Definition

6. The displacement coordinate directions at P are dependent on the location of P as shown above by (u_r, u_θ, u_z) .
7. It is recommended that points on the z-axis not have their displacement directions defined in this coordinate system. See the discussion of cylindrical coordinate systems in [Grid Point and Coordinate System Definition](#) (p. 34) in the *MSC Nastran Reference Manual*. If CYLINDRICAL and point exactly on z-axis ($r = 0.0$ exactly) then R, θ, Z triad coordinates are exactly aligned with the local defining system triad. User caution: if $r \approx 0.0$ but not exactly 0.0 should be avoided as unexpected direction cosine matrix may result.
8. If any CORD2C, CORD2R, or CORD2S entry is changed or added on restart, then a complete re-analysis is performed. Therefore, CORD2C, CORD2R, or CORD2S changes or additions are not recommended on restart.
9. An example of defining a coordinate system with respect to another coordinate system (RID) may be found in the Linear Static Analysis manual [Example](#) (Ch. 3).

CORD2R**Rectangular Coordinate System Definition, Form 2**

Defines a rectangular coordinate system using the coordinates of three points.

Format:

1	2	3	4	5	6	7	8	9	10
CORD2R	CID	RID	A1	A2	A3	B1	B2	B3	
	C1	C2	C3						

Example:

CORD2R	3	17	-2.9	1.0	0.0	3.6	0.0	1.0	
	5.2	1.0	-2.9						

Descriptor	Meaning
CID	Coordinate system identification number. (Integer > 0)
RID	Identification number of a coordinate system that is defined independently from this coordinate system. (Integer ≥ 0 ; Default = 0; which is the basic coordinate system.)
Ai, Bi, Ci	Coordinates of three points given with respect to the coordinate system defined by RID. (Real)

Remarks:

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, CORD2S, and CORD3G entries must be unique.
2. The three points [(A1, A2, A3), (B1, B2, B3), (C1, C2, C3)] must be unique and noncolinear. Noncolinearity is checked by the geometry processor. The first point defines the origin. The second defines the direction of the z-axis. The third point defines a vector which, with the z-axis, defines the x-z plane. The reference coordinate system must be independently defined.
3. The continuation entry is required.
4. If RID is zero or blank, the basic coordinate system is used.
5. An example of defining a coordinate system with respect to another coordinate system (RID) may be found in the Linear Static Analysis manual [Example](#) (Ch. 3).

6. The location of a grid point (P in the [Figure 9-44](#)) in this coordinate system is given by (X, Y, Z).

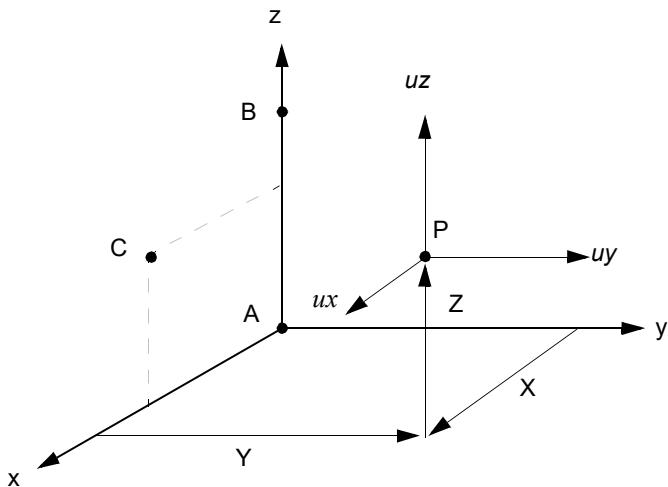


Figure 9-44 CORD2R Definition

7. The displacement coordinate directions at P are shown by (u_x, u_y, u_z) .
8. If any CORD2C, CORD2R, or CORD2S entry is changed or added on restart, then a complete re-analysis is performed. Therefore, CORD2C, CORD2R, or CORD2S changes or additions are not recommended on restart.

CORD2S**Spherical Coordinate System Definition, Form 2**

Defines a spherical coordinate system using the coordinates of three points.

Format:

1	2	3	4	5	6	7	8	9	10
CORD2S	CID	RID	A1	A2	A3	B1	B2	B3	
	C1	C2	C3						

Example:

CORD2S	3	17	-2.9	1.0	0.0	3.6	0.0	1.0	
	5.2	1.0	-2.9						

Descriptor	Meaning
CID	Coordinate system identification number. (Integer > 0)
RID	Identification number of a coordinate system that is defined independently from this coordinate system. (Integer ≥ 0 ; Default = 0 is the basic coordinate system.)
Ai, Bi, Ci	Coordinates of three points given with respect to the coordinate system defined by RID. (Real)

Remarks:

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, CORD2S, and CORD3G entries must all be unique.
2. The three points [(A1, A2, A3), (B1, B2, B3), (C1, C2, C3)] must be unique and noncolinear. Noncolinearity is checked by the geometry processor. The first point defines the origin. The second point defines the direction of the z-axis. The third lies in the plane of the azimuthal origin. The reference coordinate system must be independently defined.
3. The continuation entry is required.
4. If RID is zero or blank, the basic coordinate system is used.
5. An example of defining a coordinate system with respect to another coordinate system (RID) may be found in the Linear Static Analysis manual [Example](#) (Ch. 3).

6. The location of a grid point (P in [Figure 9-45](#)) in this coordinate system is given by (R, θ, ϕ) , where θ and ϕ are measured in degrees.

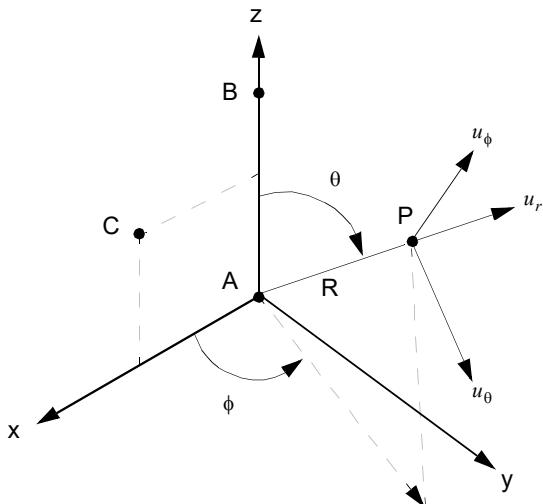


Figure 9-45 CORD2S Definition

7. The displacement coordinate directions at P are shown above by (u_r, u_θ, u_ϕ) .
8. It is recommended that points on the z-axis not have their displacement directions defined in this coordinate system. See the discussion of spherical coordinate systems in [Grid Point and Coordinate System Definition](#) (p. 34) in the *MSC Nastran Reference Guide*.

If SPHERICAL and $R = 0.0$ (point at origin) then R, θ, ϕ triad coordinates are exactly aligned with the local defining system triad. User caution: if $R \approx 0.0$ but not exactly 0.0 should be avoided as unexpected direction cosine matrix may result.

If SPHERICAL and r in azimuth plane is $r = 0.0$ and $z \neq 0.0$ then the coordinate system is defined as R global along $\pm Z$ -local defining system triad according if $z > 0.$ or $z < 0.$, θ -global along X-local defining system triad and ϕ -global along Y-local defining system triad. User caution: if $r \approx 0.0$ but not exactly 0.0 should be avoided as unexpected direction cosine matrix may result.

9. If any CORD2C, CORD2R, or CORD2S entry is changed or added on restart, then a complete re-analysis is performed. Therefore, CORD2C, CORD2R, or CORD2S changes or additions are not recommended on restart.

CORD3G**General Coordinate System**

Defines a general coordinate system using three rotational angles as functions of coordinate values in the reference coordinate system. The CORD3G entry is used with the MAT9 entry to orient material principal axes for 3-D composite analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CORD3G	CID	METHOD	FORM	THETAID1	THETAID2	THETAID3	CIDREF		

Example:

CORD3G	100	E313	EQN	110	111	112	0		
--------	-----	------	-----	-----	-----	-----	---	--	--

Descriptor	Meaning
CID	Coordinate system identification number. See Remark 1. (Integer > 0)
METHOD	E313 or S321 for Euler angle rotation in 313 sequence or space-fixed rotation in 321 sequence. See Remark 2. (Character; Default = "E313")
FORM	Specifies the Bulk Data entry which defines angles. FORM = "EQN" for DEQATN entry or FORM = "TABLE" for TABLE3D entry. (Character; Default = "EQN")
THETAID	Identification number for DEQATN or TABLE3D Bulk Data entry which defines the three angles (in radians) measured from reference coordinates to the general material coordinate system. See Remark 3. (Integer > 0)
CIDREF	Identification number for the coordinate system from which the orientation of the general coordinate system is defined. (Integer ≥ 0 ; Default = 0)

Remarks:

1. CID must be unique with respect to all other coordinate systems. CID cannot be referenced on GRID entries.
2. Three Euler angles specify the rotation of the CORD3G coordinate axes (xyz) with respect to the local Cartesian coordinate axes (XYZ) in CIDREF as follows: first rotate about Z-axis by θ_1 , next rotate about rotated x-axis by θ_2 , and then rotate about rotated z-axis by θ_3 . On the other hand, the space-fixed rotations in 321 sequence specify all the rotations about the fixed coordinate axes: first rotate about Z by θ_1 , next about Y by θ_2 , then about X by θ_3 .
3. The three rotations define a coordinate transformation which transforms position vectors in the reference coordinate system into the general coordinate system.
4. The DEQATN option must have three arguments representing the three axes of CIDREF, although not all arguments are necessarily needed in the equation.

5. If FORM = EQN is used with CIDREF pointing to any CORD1k or CORD2k, k = C,S, and a spatial relationship is required, then the equations are written using R, θ, φ as input variables. Non spatial fixed values such as THETA1 (A,B,C)=PI(1.) may of course be entered for any type of coordinate defined by the CIDREF.

If FORM = TABLE3D is used with CIDREF pointing to any CORD1k or CORD2k, k = C,S, and a spatial relationship is required then the equations are written using the variables X,Y, Z defined at the origin of the CIDREF as input values and not the variables R, θ, Z or R, θ, φ input values. For example, if a CORD1S is used then the table input value is X internally computed as $R \cdot \sin\theta \cdot \cos\varphi$.

Whether the EQN or TABLE3D FORM is used, the equation must return an angle measure in radians to have any meaning.

CORD3R**Moving Rectangular Coordinate System**

Defines a moving rectangular coordinate system using three points in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
CORD3R	CID	N1	N2	N3	CID	N1	N2	N3	

Example:

CORD3R	1001	1	144	300					
--------	------	---	-----	-----	--	--	--	--	--

Descriptor	Meaning
CID	Unique coordinate system number. (Integer > 0)
N1, N2, N3	Grid point numbers (must be unique). (Integer > 0)

Remarks:

1. Available in SOL 700 only.
2. Two different coordinate systems may be defined on one entry.
3. The grid points must be defined in an independent coordinate system.
4. The first grid point is the origin, the second lies on the z-axis and the third lies in the x-z plane.
5. The position and orientation of the coordinate system is updated as the grid points move.
6. The three grid points must not be colinear.

COUCOHF**Cohesive Friction Model To Be Used For a COUPLE Entry**

Defines a cohesive friction model suited for Euler Coupled analyses. The friction model is defined as part of the coupling surface. Use SOL700 only.

Format:

COUCOHF	CID	COHFRID	SUBID	COHFRICID				
---------	-----	---------	-------	-----------	--	--	--	--

Example:

COUCOHF	112	14	12	12				
---------	-----	----	----	----	--	--	--	--

Descriptor	Meaning
CID	Unique number of a COUCOHF entry. (Integer ≥ 0 ; required)
COHFRID	Number of a set of COUCOHF entries. COUCOHF must be referenced from a COUPLE entry. (Integer ≥ 0 ; required)
SUBID	BSURF, BCSEG or BCPROP id, which must be a part of the BSID field of the COUPLE entry. See Remark 2. (Integer ≥ 0 ; default=blank)
COHFRICID	Cohesive friction ID (Integer ≥ 0 ; required)

Remarks:

1. One couple entry can reference more than one COUCOHF entry. This allows a cohesive friction that varies along the coupling surface.
2. When SUBID is left blank, then the cohesive friction applies to the whole coupling surface.
3. A coupling surface segment can only have one porosity or one cohesive friction definition assigned.

COUOPT

Coupling Surface Pressure Definition

Defines the interaction factor and a pressure load from the covered side acting on a BSURF, BCPROP, BCMATL, BCSEG, BCBOX. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
COUOPT	CID	OPTID	SUBID	FACTOR	FACTORY				
	PLCOVER	PLCOVERV							

Example:

COUOPT	1	80	42	CONSTANT					
	CONSTANT	1.E5							

Descriptor	Meaning
CID	Unique number of a COUOPT entry. (Integer > 0; Required)
OPTID	Number of a set of COUOPT entries. OPTID must be referenced from a COUPLE entry. (Integer > 0; Required)
SUBID	<p>> 0 Number of a BSURF, BCBOX, BCPROP, BCMATL or BCSEG, which must be part of the surface as defined in the COUPLE entry. (Integer ≥ 0, 0)</p> <p>= 0 COUOPT definitions used for the entire surface as defined in the COUPLE entry.</p>
FACTOR	<p>Method of defining the interaction FACTORV with which the Eulerian pressure acting on the surface is multiplied. (Character, CONSTANT)</p> <p>CONSTANT The FACTOR is constant and specified in FACTORV</p>
FACTORV	The interaction factor. (Real, 1.)
PLCOVER	<p>Method of defining the pressure load exerted on the faces of the surface from the covered side. The pressure load is applied only when the Eulerian pressure is greater than zero. (Character, CONSTANT)</p> <p>CONSTANT The PLCOVER is constant and specified in PLCOVERV.</p>
TABLE	The PLCOVER varies with time. PLCOVERV is the number of a TABLED1 entry giving the variation of the PLCOVER (y-value) with time (x-value).
PLCOVERV	The pressure load or the number of a TABLED1 entry depending on the PLCOVER entry. (Real ≥ 0 , 0.)

Remarks:

1. The effect of specifying an interaction FACTOR is similar to specifying a porosity coefficient on a COUPOR entry. The difference is that in this case the surface still acts as a wall boundary for the Eulerian material.
2. Applying a PLCOVER instead of applying a pressure load on the faces through either a PLOAD, PLOAD4, or DAREA entry gives the following differences:
 - a. PLCOVER is applied only when there is a balancing Eulerian pressure greater than zero.
 - b. Possible porosity as defined on a COUPOR entry is taken into account when applying the PLCOVER.
 - c. With PARAM,PLCOVCUT you can define a cut-off time that is applied to PLCOVER.
3. The covered side of a surface lies on the side where there is no Eulerian material.

COUP1FL**Coupling Surface Failure**

Defines the surrounding variables when a segment of a coupling surface fails. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
COUP1FL	CFID	RHO	SIE	XVEL	YVEL	ZVEL	PRESSURE	MATERIAL	

Example:

COUP1FL	3	1.225	204082.	900.					
---------	---	-------	---------	------	--	--	--	--	--

Descriptor	Meaning
CFID	Unique ID of a COUP1FL entry referenced from the COUPLE entry. (Integer > 0; Required)
RHO	Surrounding density. See Remark 2. (Real > 0)
SIE	Surrounding specific internal energy. See Remark 2. (Real)
XVEL	Surrounding x-velocity. See Remark 2. (Real)
YVEL	Surrounding y-velocity. See Remark 2. (Real)
ZVEL	Surrounding z-velocity. See Remark 2. (Real)
PRESSURE	Surrounding pressure. See Remark 4. (Real)
MATERIAL	MATDEUL ID. Only used when the multi-material Euler solver is active. (Blank)

Remarks:

1. This entry can only be used in combination with DYPARAM,FASTCOUP, ,FAIL and with either the HYDRO, MMHYDRO or MMSTREN Euler Solver. For restrictions on the use of COUP1FL refer to param,flow-method. The coupling surface failure is associated with the element failure of the structure to which the surface is connected. Therefore, you have to define a failure model for the structure for the entry to take effect in the analysis.
2. For the first order Euler solvers no restriction apply to the values of the surrounding variables. For the Roe solver at least one of the surrounding variables should be defined. The default value of the density (RHO) will be set equal to the reference density as defined on the MATDEUL entry. By default, the other variables (SIE, XVEL, YVEL and ZVEL) are set equal to zero.
3. The coupling surface must only consist of CQUAD and/or CTRIA elements.
4. The field PRESSURE has to be left blank in combination with the Roe solver.
5. In combination with multi-material Euler only outflow of material is allowed. Each material in an outflow Euler element is transported. The materials are transported in proportion to their relative volume fractions.

COUPINT

Coupling Surface Interaction

Defines the surrounding variables when a segment of a coupling surface fails. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
COUPINT	CID	CID1	CID2						

Example:

COUPINT	33	2							
---------	----	---	--	--	--	--	--	--	--

Descriptor	Meaning
CIID	Unique number of a COUPINT entry.
CID1	Number of COUPLE entry 1.
CID2	Number of COUPLE entry 2.

Remarks:

1. This entry can only be used in combination with DYPARAM,FASTCOUP, ,FAIL and with either the HYDRO, MMHYDRO or MMSTREN Euler Solver. The interaction will be activated when failure of a Lagrangian structure with which the coupling surface is associated occurs. Therefore, you have to define a failure model for the material of the structure.
2. The coupling surface must consist of CQUAD and/or CTRIA elements.

COUPLE**General Euler-Lagrange Coupling Surface**

Defines a coupling surface that acts as the interface between an Eulerian (finite volume) and a Lagrangian (finite element) domain. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
COUPLE	CID	BSID	COVER	REVERSE	CHECK	PORID	OPTID	CTYPE	
	INFID	HTRID	FS	FK	EXP	INTID			
	SET1ID	MESHID	TDEAC	COUP1FL	HYDSTAT	SKFRIC	COHFRID		

Example:

COUPLE	100	37	INSIDE	ON	ON				
				0.3	0.0				

Descriptor	Meaning	
CID	Unique number of a COUPLE entry. (Integer > 0; Required)	
BSID	Number of a BSURF, BCBOX, BCPROP, BCMATL or BCSEG entry defining the coupling surface. (Integer > 0; Required)	
COVER	The processing strategy for Eulerian elements inside and outside of the coupling surface. (Character, INSIDE)	
	INSIDE The Eulerian elements inside the closed volume of the coupling surface are not processed.	
	OUTSIDE The Eulerian elements outside the closed volume of the coupling surface are not processed	
	NONE The Eulerian elements on both sides of the coupling surface are processed.	
REVERSE	Auto-reverse switch for the coupling surface segments. (Character, ON)	
	ON If required, the normals of the coupling surface segments are automatically reversed so that they all point in the same general direction as to give a positive closed volume.	
	OFF The segments are not automatically reversed. The user is responsible for defining the correct general direction of the segment normals.	
CHECK	Check switch for coupling surface segments. (Character, ON)	
	ON The normals of the segments are checked to verify that they all point in the same general direction and yield a positive closed volume.	
	OFF The segments are not checked. It is the responsibility of the user to ensure that the direction of the segment normals yield a positive closed volume.	

Descriptor	Meaning
	When “REVERSE” is set to “ON”, the “CHECK” option will be automatically activated.
PORID	Number of a set of “LEAKAGE” entries that define the porosity for the BSURF entries. (Integer > 0 , 0 (no porosity))
OPTID	Not available for the Roe solver Number of a set of “COUOPT” entries that define special options for the BSURF entries (Integer ≥ 0 , 0 (no special options))
CTYPE	Not available for the Roe solver Coupling surface type definition. (Character, STANDARD) STANDARD Standard Euler-Lagrange interaction. AIRBAG Coupling for airbag applications. It is equivalent to the standard coupling algorithm with the following exceptions that tailor the solution for airbag applications: Inflow through a porous (sub)-surface will only occur when there is already some material (gas) in the Eulerian element. Almost empty Eulerian elements will be automatically eliminated. The standard algorithm redistributes the small mass to the most suitable neighbor elements.
INFID	Number of a set of “ABINFL” entries that define the inflator(s) on the subsurface(s) of the coupling surface. Not available for the Roe solver. (Integer > 0 , 0 (no inflators))
HTRID	Number of a set of “HEATLOS” entries that define the heat transfer definition(s) on the subsurface(s) of the coupling surface. Not available for the Roe solver. (Integer ≥ 0 , 0 (no heat transfer))
FS	Static friction coefficient. See Remark 6. Not available for the Roe solver. (Real ≥ 0.0 , 0.0)
FK	Kinetic friction coefficient. See Remark 6. Not available for the Roe solver. (Real ≥ 0.0 , 0.0)
EXP	Exponential decay coefficient. See Remark 6. Not available for the Roe solver. (Real ≥ 0.0 , 0.0)
INTID	ID of an INITGAS entry specifying initial gas composition for the Euler mesh. See Remark 7. (Integer ≥ 0 , 0 (no initial gas composition))
SET1ID	The number of a SET1 entry, which defines the Eulerian elements associated with this coupling surface. See Remark 8. (Integer ≥ 0 ; Default = blank)
MESHID	ID of a MESH entry, which defines the Eulerian mesh associated with this coupling surface. See Remark 8. (Integer ≥ 0 ; Default = blank)
TDEAC	Time of deactivation of the coupling surface and the associated Eulerian mesh. (Real ≥ 0.0 , 1.E20)

Descriptor	Meaning
COUP1FL	The number of a COUP1FL entry, which defines the surrounding variables for the coupling surface when its segments fail. See Remark 9. (Integer ≥ 0 ; Default = blank)
HYDSTAT	The number of a HYDSTAT entry, which specifies a hydrostatic preset. The preset is applied to all Euler element specified by the SET1ID and MESHID. See Remark 11. (Integer ≥ 0 , 0 (no hydrostatic preset))
SKFRIC	Skin friction value. See Remark 13. (Real $\geq 0.0, 0.0$)
COHFRID	Number of a set of COUCOHF entries that define the cohesive friction on the subsurface(s) of the coupling surface. (I ≥ 0 Default = blank)

Remarks:

1. All coupling surfaces must form a multi-faceted closed volume. If necessary, additional segments must be defined to achieve the closed volume. The closed volume must intersect at least one Euler element initially.
2. All segments must be attached to the face of an element. Dummy elements can be used to define any additional segments that are required to create the closed volume.
3. The normals of all segments that form the coupling surface must point in the same general direction and result in a positive closed volume. Setting the “REVERSE” option to “ON” ensures that this condition is satisfied, regardless of the initial definition of the segments.
4. The “COVER” field determines how Eulerian elements that are inside and outside of the coupling surface are processed. The default setting of INSIDE is appropriate for most of the problems. In the majority of analyses, the Eulerian material flows around the outside of the coupling surface. Therefore, the Eulerian elements that fall within the coupling surface do not contain material. For some specific applications, such as airbag inflation, the Eulerian material (gas) is completely contained within the coupling surface. In these cases, the “COVER” definition should be set to OUTSIDE.
5. By default the fast coupling algorithm is used. The algorithm then used is substantially faster than the general coupling. The restriction is that you cannot use an arbitrarily shaped Euler mesh with the fast coupling algorithm. All element faces of the Euler mesh must have their normal pointing in any of the three basic coordinate directions. If you want to use the general coupling algorithm, you can define the parameter “DYPARAM,FASTCOUP,NO” in the input file.
6. The friction model implemented for the coupling algorithm is a simple Coulomb friction definition. The friction coefficient μ is defined as:

$$\mu = \mu_k + (\mu_s - \mu_k) \cdot e^{-(\beta \cdot v)}$$

where μ_s is the static friction coefficient, μ_k is the kinetic friction coefficient, β the exponential decay coefficient and v the relative sliding at the point of contact.

7. An initial gas composition is for use with the single-material hydrodynamic Euler solver and an ideal-gas equation of state (EOSGAM) only.

8. Multiple coupling surfaces are available when you associate one Eulerian domain with a single coupling surface by either using the SET1ID or the MESHID option. Note that only one of the two options may be set and will work only in combination with the fast coupling algorithm.
9. The COUP1FL option is available and valid only in combination with the fast coupling algorithm with the failure option (DYPARAM,FASTCOUP, ,FAIL). If no number is given, the default values of the surrounding variables will be used; the density (RHO) is set equal to the reference density as defined on the MATDEUL entry. By default, the other variables (SIE, XVEL, YVEL and ZVEL) are set equal to zero.
10. If an ACTIVE entry is present, its definition is ignored in case the TDEAC value is defined in combination with the fast coupling algorithm.
11. If there is only one coupling surface and no adaptive meshing is used, the HYDSTAT field can be left blank. To impose a boundary condition that matches the hydrostatic initialization, the PORHYDS entry can be used.
12. A mixture of BSURF, BCBOX, BCPROP, BCMATL or BCSEG with the same BSID is allowed. However multiple BSID of the same type is not allowed. When using this option, special care must be taken to assure the same element is not part of multiple BSID definitions.
13. The skin friction is defined as:

$$C_f = \frac{\tau_w}{0.5 \cdot \rho u^2}$$

Here, τ_w denotes the shear friction in an Euler element adjacent to a couple surface segment where ρ is the density and u is the tangential relative velocity in the Euler element that is adjacent to a couple surface segment.

SKFRIC will only be used when VISC has been set on either an EOSGAM or an EOSPOL entry. If VISC has been set and if SKFRIC has not been set then a no slip condition will be prescribed at the interface between fluid and structure.

14. If the coupling surface is a structural solid then Sol700 can automatically create a coupling surface for the solid. This coupling surface consists of the boundary faces of the solid. To activate this, a BCPROP has to be created for the solid and the BCPROP has to be used for the BSID of the COUPLE entry.
15. Option NONE requires the use of DYPARAM, AUTCOUP. For details refer to this DYPARAM.

CPENTA**Five-Sided Solid Element Connection**

Defines the connections of a five-sided solid element with six to fifteen grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CPENTA	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15								

Example:

CPENTA	112	2	3	15	14	4	103	115	
	5	16	8				120	125	
	130								

Descriptor	Meaning	Type	Default
EID	Element identification number.	0 < Integer < 100,000,000	Required
PID	Property identification number of a PSOLID or PLSOLID entry.	Integer > 0	Required
Gi	Identification numbers of connected grid points.	Integer ≥ 0 or blank	Required

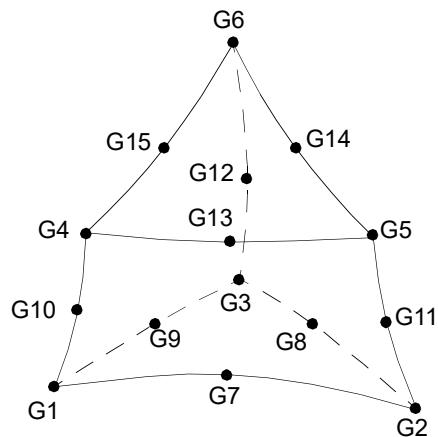


Figure 9-46 CPENTA Element Connection

Remarks:

1. Element ID numbers must be unique with respect to all other element ID numbers.
2. The topology of the diagram must be preserved; i.e., G1, G2, and G3 define a triangular face, G1, G10, and G4 are on the same edge, etc.
3. For Nastran conventional element, the edge grid points, G7 to G15, are optional. Any or all of them may be deleted. In the example shown, G10, G11, and G12 have been deleted. The continuations are not required if all edge grid points are deleted.
Note that for advanced nonlinear elements, partial specification of mid-points is not allowed. i.e. either none of the mid-points should be specified or all of the mid points should be specified.
4. Components of stress are output in the material coordinate system except for hyperelastic elements, which are output in the basic coordinate system.
5. For nonhyperelastic elements the element coordinate system for the CPENTA element is derived accordingly. The origin of the coordinate system is located at the midpoint of the straight line connecting the points G1 and G4. The Z axis points toward the triangle G4-G5-G6 and is oriented somewhere between the line joining the centroids of the triangular faces and a line perpendicular to the midplane. The midplane contains the midpoints of the straight lines between the triangular faces. The X and Y axes are perpendicular to the Z axis and point in a direction toward, but not necessarily intersecting, the edges G2 through G5 and G3 through G6, respectively.

Solid elements have both a material and an element coordinate system. Both systems are defined for the initial geometry, and for geometric nonlinear analysis they will rotate with the element. The material coordinate system is used to input anisotropic material properties and for stress output. The material coordinate system is defined by the CORDM field of the PSOLID entry. The element coordinate system is used for element stiffness integration (reduced shear for example) and optionally to define the material coordinate system (only if PSOLID,CORDM=-1).

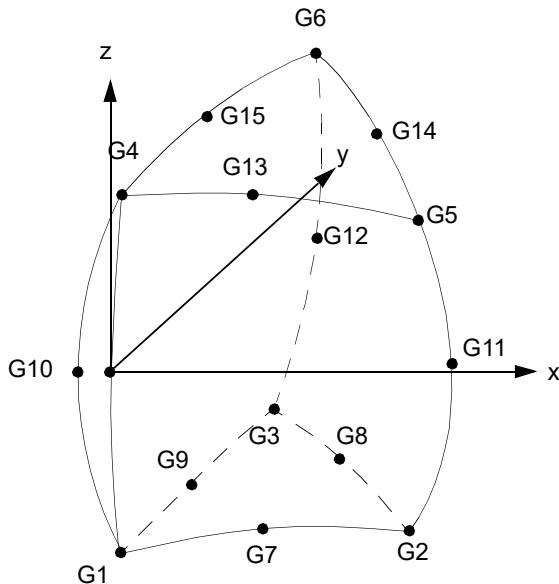


Figure 9-47 CPENTA Element Coordinate System

6. We recommend that the edge grid points be located within the middle third of the edge.
7. For hyperelastic elements, the plot codes are specified under the CPENTAFD element name in [Item Codes, 1045](#).
8. If a CPENTA element is referenced on a PSET or PVAL entry, then a p-version formulation is used and the element can have curved edges.
 - If a curved edge of a p-element is shared by an h-element without midside nodes, the geometry of the edge is ignored and set straight.
 - Elements with midside nodes cannot be p-elements and edges with midside nodes cannot be shared by p-elements.
9. By default, all of the nine edges of the element are considered straight unless:
 - For p-elements there is an FEEDGE or FEFACE entry that contains the two grids of any edge of this element. In this case, the geometry of the edge is used in the element.
 - For h-elements any of G7 through G15 are specified.
10. The internal coordinate system of the element is used internally and is based on eigenvalue techniques to insure non bias in the element formulation. For stress/strain output this internal coordinate system (CORDM=-1 on PSOLID entry) is hard to visualize. Thus a CORDM=-2 on the PSOLID is available as shown in [Figure 9-48](#).

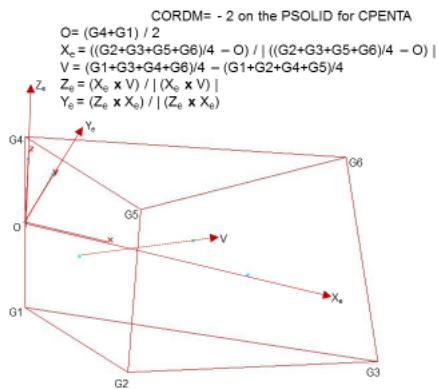


Figure 9-48 PSOLID for CPENTA

CPYRAM**Five-Sided Solid Element Connection**

Defines the connections of the five-sided solid element with five or thirteen grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CPYRAM	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13		

Example:

CPYRAM	7	4	15	16	17	18	40	21	
	22	23	24	31	32	33	34		

Descriptor	Meaning	Type	Default
EID	Element identification number.	0 < Integer < 100,000,000	Required
PID	Property identification number of a PSOLID entry.	Integer > 0	Required
Gi	Identification numbers of connected grid points.	Integer ≥ 0 or blank	Required

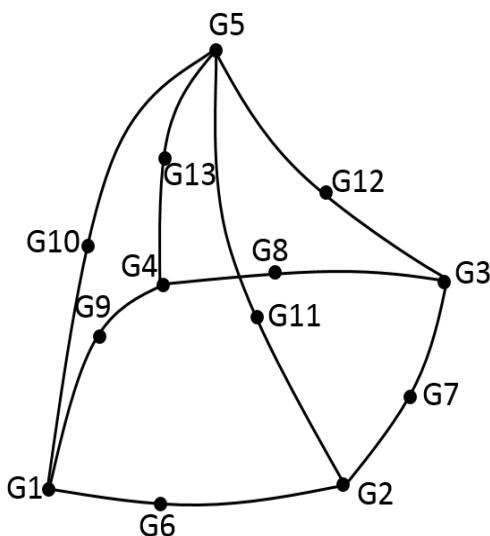


Figure 9-49 CPYRAM Element Connection

Remarks:

1. Element ID numbers must be unique with respect to all other element ID numbers.
2. The topology of the diagram must be preserved, that is, G1, G2, G3, G4 define the quadrilateral base, G5 defines the apex, G1, G5, G10 are on the same edge, etc.
3. For a 5 noded CPYRAM element, only G1 - G5 should be provided. For a 13 noded CPYRAM element, all the corner and mid-side nodes (G1 - G13) should be provided. Note that partial specification of mid-points is not allowed. i.e. either none of the mid-points should be specified or all of the mid points should be specified.
4. The pyramid elements use the basic coordinate system as the element coordinate system. They can use a material coordinate system defined by the CORDM field of the corresponding PSOLID entry. CORDM = 0, -1, -2 or blank is the same as the basic coordinate system.
5. For CORDM > 0, the material coordinate system is defined with respect to the initial geometry and is used to input orthotropic or anisotropic material properties and for stress output. For geometric nonlinear analysis, the material coordinate system rotates with the element. Component of stresses are output in the material coordinate system referenced by the CORDM field of PSOLID entry (CORDM > 0).

CQUAD**Fully Nonlinear Plane Strain Element**

Defines a plane strain quadrilateral element with up to nine grid points for use in fully nonlinear (i.e., large strain and large rotation) hyperelastic analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CQUAD	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	THETA or MCID					

Example:

CQUAD	111	203	31	74	75	32			
-------	-----	-----	----	----	----	----	--	--	--

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PLPLANE or PLCOMP entry. (Integer > 0)
G1, G2, G3, G4	Identification numbers of connected corner grid points. Required data for all four grid points. (Unique Integers > 0)
G5, G6, G7, G8	Identification numbers of connected edge grid points. Optional data for any or all four grid points. (Integer ≥ 0 or blank)
G9	Identification number of center grid point. Optional. (Integer ≥ 0 or blank)
THETA	Material property orientation angle in degrees. THETA is only applicable if PLCOMP referenced, or the PLPLANE entry has an associated PSHLN2 entry which is honored only in SOL 400. For PSHLN2 BEHi=PSTRS or PLSTRN codes, THETA is measured relative to the line defined from G1-G2. For PSHLN2 or PLCOMP BEHi=COMPS code the THETA value on the element connection entry will be ignored. (Real; Default = 0.0)
MCID	Material coordinate system identification number MCID is only applicable if the PLPLANE entry has an associated PSHLN2 entry which is honored only in SOL 400. The x-axis of the material coordinate system is determined by projecting the T1-axis of the MCID coordinate system onto the surface of the shell element as follows: CORD1R, x -axis of MCID the coordinate is projected onto shell surface and the CORD2R material angle is measured from the G1-G2 line to the to the projected x-axis

Descriptor	Meaning
CORD1C,	r-axis of MCID the coordinate is projected onto shell surface through the
CORD2C	element center and the material angle is measured from the G1-G2 line to
CORD1S,	the to the projected r-axis
CORD2S	
	For PSHLN2 (or PLCOMP) BEHi=COMPS code the MCID value on the element connection entry will be ignored. (Integer ≥ 0 ; if blank, then THETA = 0.0 is assumed.)

Remarks:

1. Element identification numbers should be unique with respect to all other element IDs of any kind.
2. Grid points G1 to G9 must be numbered as shown and must lie on a plane. G5 - G9 are Optional data for any or all four grid points for Nastran conventional element only.
Note that for advanced nonlinear elements, partial specification of mid-points is not allowed. i.e. either none of the mid-points should be specified or all of the mid points should be specified.
3. It is recommended that the edge points be located within the middle third of the edge.
4. The plot codes are specified under the CQUADFD element name in [Item Codes, 1045](#).
5. Stresses and strains are output in the coordinate system identified by the CID field of the PLPLANE entry.

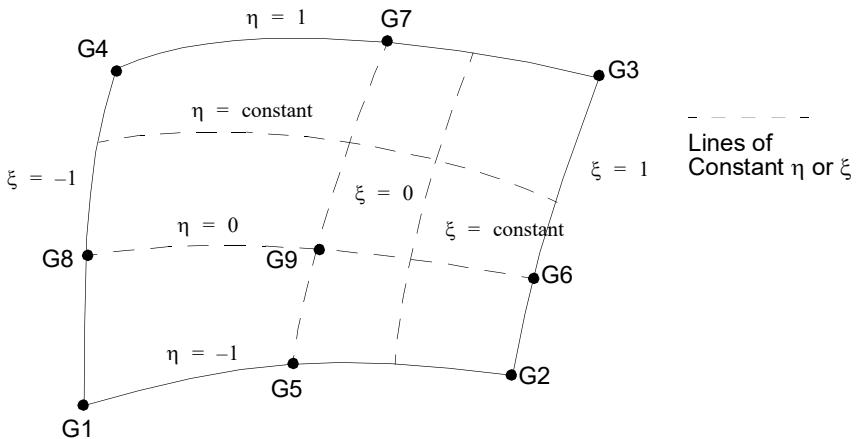


Figure 9-50 CQUAD Element Coordinate System

CQUAD4

Quadrilateral Plate Element Connection

Defines an isoparametric membrane-bending or plane strain quadrilateral plate element.

Format:

1	2	3	4	5	6	7	8	9	10
CQUAD4	EID	PID	G1	G2	G3	G4	THETA or MCID	ZOFFS	
		TFLAG	T1	T2	T3	T4			

Example:

CQUAD4	111	203	31	74	75	32	2.6	0.3	
			1.77	2.04	2.09	1.80			

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PSHELL, PCOMP, PCOMPG or PLPLANE or PLCOMP entry. (Integer > 0; Default = EID)
Gi	Grid point identification numbers of connection points. (Integers > 0, all unique.)
THETA	Material property orientation angle in degrees. THETA is ignored for hyperelastic elements. See Figure 9-52 . See Remark 10. (Real; Default = 0.0)
MCID	Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the T1-axis of the MCID coordinate system onto the surface of the shell element as follows: CORD1R, x -axis of MCID the coordinate is projected onto shell surface and the CORD2R material angle is measured from the G1-G2 line to the to the projected x-axis CORD1C, r-axis of MCID the coordinate is projected onto shell surface through the CORD2C element center and the material angle is measured from the G1-G2 line CORD1S, to the to the projected r-axis CORD2S Use DIAG 38 to print the computed THETA values. MCID is ignored for hyperelastic elements. For SOL 600, only CORD2R is allowed. See Remark 10. (Integer \geq 0; If blank, then THETA = 0.0 is assumed.)

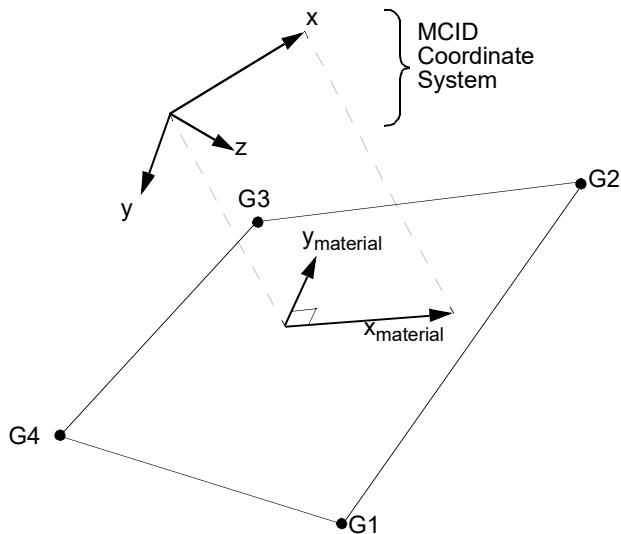


Figure 9-51 MCID Coordinate System Definition

ZOFFS	Offset from the surface of grid points to the element reference plane. ZOFFS is ignored for hyperelastic elements. See Remark 6. (Real)
TFLAG	An integer flag, signifying the meaning of the Ti values. (Integer 0, 1, or blank)
Ti	Membrane thickness of element at grid points G1 through G4. If "TFLAG" is zero or blank, then Ti are actual user specified thicknesses. See Remark 4. for default. (Real ≥ 0.0 or blank, not all zero.) If "TFLAG" is one, then the Ti are fractions relative to the T value of the PSHELL. (Real > 0.0 or blank, not all zero. Default = 1.0) Ti are ignored for hyperelastic elements.

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Grid points G1 through G4 must be ordered consecutively around the perimeter of the element.
3. All interior angles must be less than 180° .
4. The continuation is optional. If it is not supplied, then T1 through T4 will be set equal to the value of T on the PSHELL entry.

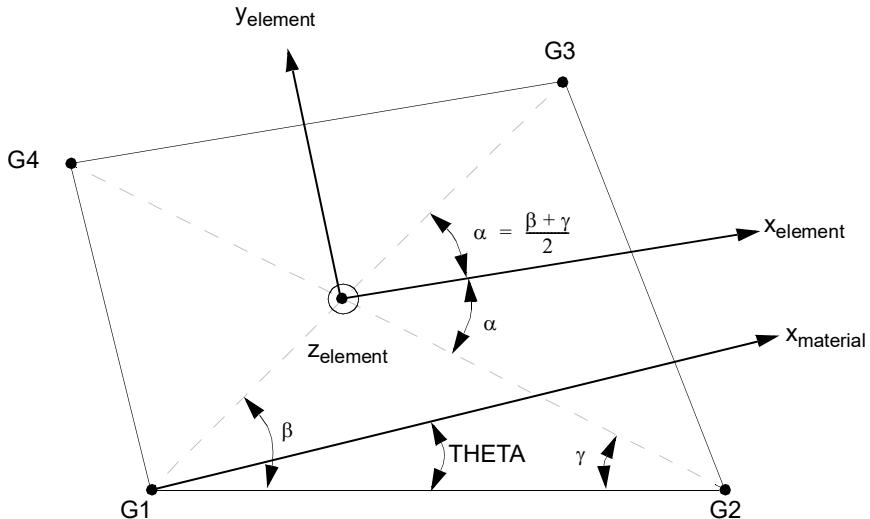


Figure 9-52 CQUAD4 Element Geometry and Coordinate Systems

5. The reference coordinate system for the output of stress, strain and element force depends on the element type.
 - For CQUAD4 elements which are not p-elements and not hyperelastic, the reference coordinate system is the default for output is the element coordinate system. See PARAM,OMID for output in the material system.
 - For CQUAD4 elements referenced by a PSET or PVAL entry, the stresses, strains and element forces are output in the local tangent plane of the element. The local tangents are oriented in a user defined direction which is uniform across a set of elements. By default, the local tangent x-direction is oriented in the positive x-direction of the basic coordinate system. See the Bulk Data entry, [OUTRCV, 2662](#) for user defined output coordinate systems.
 - For hyperelastic elements the stress and strain are output according to CID on the PLPLANE entry.
 6. Elements may be offset from the connection points by means of ZOFFS. Other data, such as material matrices and stress fiber locations, are given relative to the reference plane. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive Z-axis of the element coordinate system. If the ZOFFS field is used, then the MID1 and MID2 fields must be specified on the PSHELL entry referenced both by PID.
- Two methods are available for the computation of offsets: original and enhanced. The default method is the original method. The enhanced method is requested by the Bulk Data entry [MDLPRM,OFFDEF,LROFF](#). For options of offsets, please refer to the Bulk Data entry [MDLPRM,OFFDEF,option](#).

For solution sequences that require differential stiffness such as SOL103 (with preloading), SOL105, and SOL400, it is **highly recommended** to use MDLPRM, OFFDEF, LROFF. In SOL103, if the case control ADAMSMNF FLEXBODY=YES or AVLEXB EXBBODY=YES is present, then MDLPRM, OFFDEF, LROFF must be replaced by **MDLPRM, OFFDEF, NOMASS**.

For SOLs 106, 129, 153, and 159 the differential stiffness for offset vectors will give incorrect results with PARAM, LGDISP, 1. In addition in SOLs 106 and 129 offset vectors will produce incorrect results with thermal loading.

7. For finite deformation hyperelastic analysis, the plot codes are given by the CQUADFD element name in [Item Codes, 1045](#).
8. If a CQUAD4 element is referenced by a PSET or PVAL entry, then a p-version formulation is used and the element can have curved edges.
 - If a curved edge of a p-element is shared by an h-element CQUAD4, the geometry of the edge is ignored and set straight.
9. By default, all of the four edges of the element are considered straight unless the element is a p-element and the edge is associated to curved geometry with a FEEDGE or FEFACE entry.
10. If element has an associated PSHLN2 or PLCOMP entry with BEHi=COMPS code, the THETA/MCID value on the element connection entry will be ignored.
11. For RC network solver in thermal analysis, the ZOFFS is ignored.
12. In SOL 600, when PCOMP is used, one must define the material property with orientation, MCID>0.

CQUAD8**Curved Quadrilateral Shell Element Connection**

Defines a curved quadrilateral shell or plane strain element with eight grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CQUAD8	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	T1	T2	T3	T4	THETA or MCID	ZOFFS	
	TFLAG								

Example:

CQUAD8	207	3	31	33	73	71	32	51	
	53	72	0.125	0.025	0.030	.025	30.	.03	

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PSHELL, PCOMP, PCOMPG or PLPLANE or PLCOMP entry. (Integer > 0)
G1, G2, G3, G4	Identification numbers of connected corner grid points. Required data for all four grid points. (Unique Integers > 0)
G5, G6, G7, G8	Identification numbers of connected edge grid points. Optional data for any or all four grid points for Nastran conventional element only. (Integer > 0 or blank). Note that for advanced nonlinear elements, partial specification of mid-points is not allowed. i.e. either none of the mid-points should be specified or all of the mid points should be specified.
Ti	Membrane thickness of element at grid points G1 through G4. If "TFLAG" zero or blank, then Ti are actual user specified thickness. See Remark 4. for default. (Real ≥ 0.0 or blank, not all zero.) If "TFLAG" one, then the Ti are fraction relative to the T value of the PSHELL. (Real > 0.0 or blank, not all zero. Default = 1.0) Ti are ignored for hyperelastic elements.
THETA	Material property orientation angle in degrees. See Figure 9-53. THETA is ignored for hyperelastic elements. See Remark 10. (Real; Default = 0.0)
MCID	Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the T1-axis of the MCID coordinate system onto the surface of the shell element as follows: CORD1R x -axis of MCID the coordinate is projected onto shell surface and the CORD2R material angle is measured from the G1-G2 line to the to the projected x-axis

Descriptor	Meaning
CORD1C, CORD2C CORD1S, CORD2S	r-axis of MCID the coordinate is projected onto shell surface through the element center and the material angle is measured from the G1-G2 line to the to the projected r-axis (see Remark 3.) MCID is ignored for hyperelastic elements. For SOL 600, only CORD2R is allowed. See Remark 10. (Integer ≥ 0 ; if blank, then THETA = 0.0 is assumed.)
ZOFFS	Offset from the surface of grid points to the element reference plane. See Remark 6. ZOFFS is ignored for hyperelastic elements. (Real)
TFLAG	An integer flag, signifying the meaning of the Ti values. (Integer 0, 1, or blank)

Remarks:

- Element identification numbers should be unique with respect to all other element IDs of any kind.
- Grid points G1 to G8 must be numbered as shown in [Figure 9-53](#).
- The orientation of the material property coordinate system is defined locally at each interior integration point by THETA, which is the angle between x_{material} and the line of constant η .

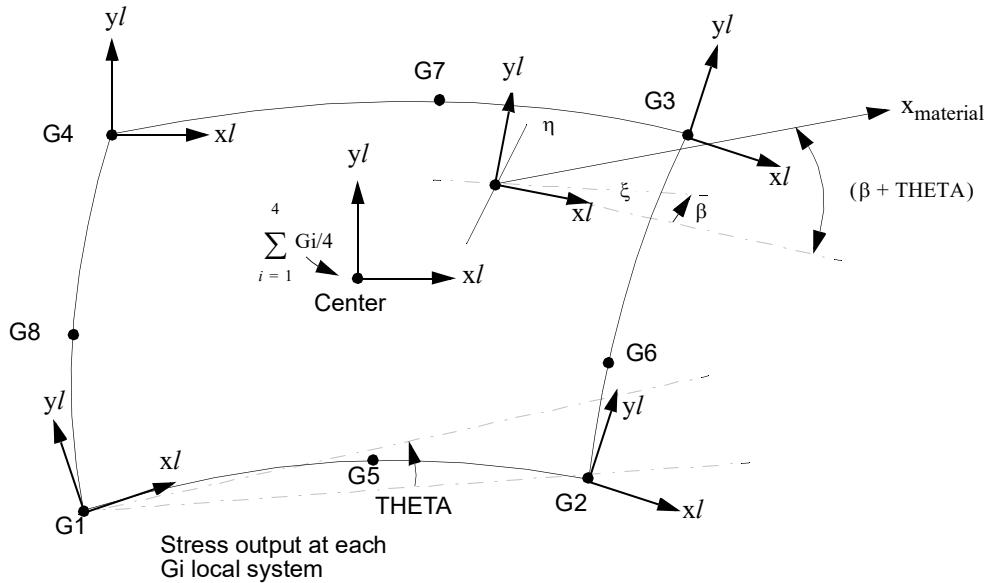
The definition of the material coordinate system by projection is used to calculate an angle THETA. Please note that since x_i changes directions throughout the element based on element shape, the material coordinate system varies similarly. Because of this an orthotropic or anisotropic material will cause the CQUAD8's stiffness to be biased by both its shape and grid ordering. Use the QUAD4 element if a constant material coordinate system direction is desired with orthotropic and anisotropic materials.

- T1, T2, T3 and T4 are optional. If they are not supplied and no TFLAG, then T1 through T4 will be set to the value of T on the PSHELL entry.
- It is recommended that the midside grid points be located within the middle third of the edge. If the edge point is located at the quarter point, the program may fail with a divide-by-zero error or the calculated stresses will be meaningless.
- Elements may be offset from the connection points by means of the ZOFFS field. Other data, such as material matrices and stress fiber locations, are given relative to the reference plane. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive z-axis of the element coordinate system. If the ZOFFS field is used, then both the MID1 and MID2 fields must be specified on the PSHELL entry referenced by PID.

Two methods are available for the computation of offsets: original and enhanced. The default method is the original method. The enhanced method is requested by the Bulk Data entry MDLPRM, OFFDEF, LROFF. For options of offsets, please refer to the Bulk Data entry MDLPRM, OFFDEF, option.

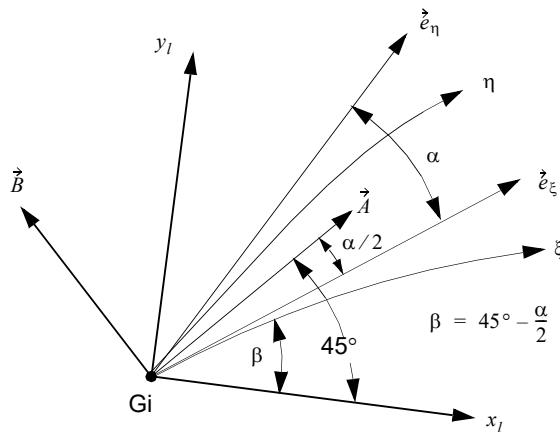
For solution sequences that require differential stiffness such as SOL103 (with preloading), SOL105, and SOL400, it is **highly recommended** to use MDLPRM, OFFDEF, LROFF. In SOL103, if the case control ADAMSMNF FLEXBODY=YES or AVLEXB EXBBODY=YES is present, then MDLPRM, OFFDEF, LROFF must be replaced by MDLPRM, OFFDEF, NOMASS.

7. If all midside grid points are deleted, then the element will be excessively stiff and the transverse shear forces incorrect. A User Warning Message is printed, and a CQUAD4 element is recommended instead. If the element is hyperelastic, then it is processed identically to the hyperelastic CQUAD4 element.
8. For a description of the element coordinate system, see [Shell Elements \(CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, CQUADR\)](#) (p. 123) in the *MSC Nastran Reference Guide*. Stresses and strains are output in the local coordinate system identified by x_l and y_l in [Figure 9-53](#). However, for hyperelastic elements the stress and strain are output in the coordinate system identified by the CID field on the PLPLANE entry.
9. For hyperelastic elements the plot codes are specified under the CQUADFD element name in [Item Codes, 1045](#).



where

- \hat{e}_η is tangent to η at Gi
- \hat{e}_ξ is tangent to ξ at Gi
- \hat{A} is formed by bisection of \hat{e}_η and \hat{e}_ξ
- \hat{B} and \hat{A} are perpendicular



y_l is formed by bisection of \vec{A} and \vec{B}

x_l is perpendicular to y_l

Figure 9-53 CQUAD8 Element Geometry and Coordinate Systems

10. If element has an associated PSHLN2 or PLCOMP entry with BEHi=COMPS code, the THETA/MCID value on the element connection entry will be ignored.
11. For RC network solver in thermal analysis, the ZOFFS is ignored.
12. In SOL 600, when PCOMP is used, one must define the material property with orientation, MCID>0.

CQUADR

Quadrilateral Plate Element Connection

Defines an isoparametric membrane and bending quadrilateral plate element. This element has a normal rotational (drilling) degrees-of-freedom. It is a companion to the CTRIAR element.

Format:

1	2	3	4	5	6	7	8	9	10
CQUADR	EID	PID	G1	G2	G3	G4	THETA or MCID	ZOFFS	
		TFLAG	T1	T2	T3	T4			

Example:

CQUADR	82	203	31	74	75	32	2.6		
			1.77	2.04	2.09	1.80			

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PSHELL, PCOMP or PCOMPEntry. (Integer > 0; Default = EID)
Gi	Grid point identification numbers of connection points. (Integers > 0, all unique)
THETA	Material property orientation angle in degrees. See Figure 9-55 . (Real; Default = 0)
MCID	Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the T1-axis of the MCID coordinate system onto the surface of the shell element as follows: CORD1R, CORD2R x -axis of MCID the coordinate is projected onto shell surface and the material angle is measured from the G1-G2 line to the to the projected x-axis CORD1C, CORD2C r-axis of MCID the coordinate is projected onto shell surface through the element center and the material angle is measured from the G1-G2 line to the to the projected r-axis CORD1S, CORD2S
	Use DIAG 38 to print the computed THETA values. For SOL 600, only CORD2R is allowed. (Integer \geq 0; If blank, then THETA = 0.0 is assumed.)
ZOFFS	Offset from the surface of grid point to the element plane. See Remark 8.

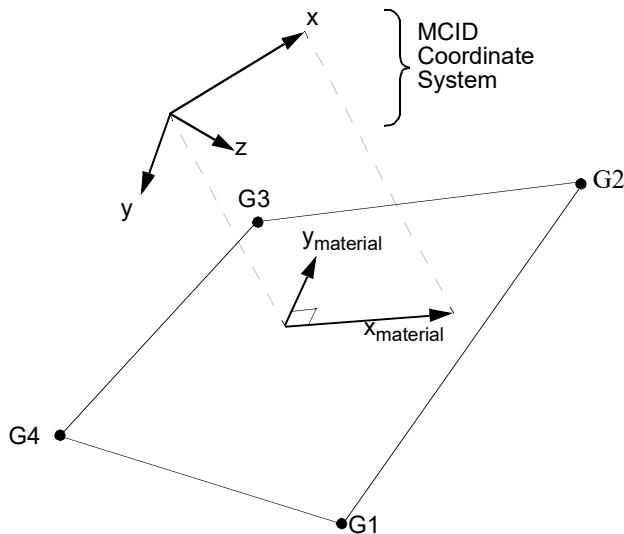


Figure 9-54 MCID Coordinate System Definition

TFLAG	An integer flag, signifying the meaning of the Ti values. (Integer 0, 1, or blank)
Ti	Membrane thickness of element at grid points G1 through G4. If "TFLAG" zero or blank, then Ti are actual user specified thickness. (Real ≥ 0.0 or blank, not all zero. See Remark 4. for default.) If "TFLAG" one, then the Ti are fraction relative to the T value of the PSHELL. (Real > 0.0 or blank, not all zero. Default = 1.0) Ti are ignored for hyperelastic elements.

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Grid points G1 through G4 must be ordered consecutively around the perimeter of the element.
3. All the interior angles must be less than 180° .
4. The continuation is optional. If it is not supplied, then T1 through T4 will be set equal to the value of T on the PSHELL entry.
5. Stresses and strains are output in the element coordinate system at the centroid and grid points G1 through G4.
6. Inaccurate results will be obtained if interior grids have the rotation normal (drilling) to the element constrained. At the boundary of a model, the drilling degrees-of-freedom must be constrained if the user wants a fixed boundary. Also, for this element it is critical that consistent membrane (in plane) edge loads be applied. Reference the [PLOAD4entry \(SORL option\)](#) and the [Consistent Surface and Edge Loads](#) (Ch. 3) in the *MSC Nastran Reference Guide* for additional information.

7. The CTRIAR element is the triangular companion to the CQUADR element and should be used instead of CTRIA3 or CTRIA6.

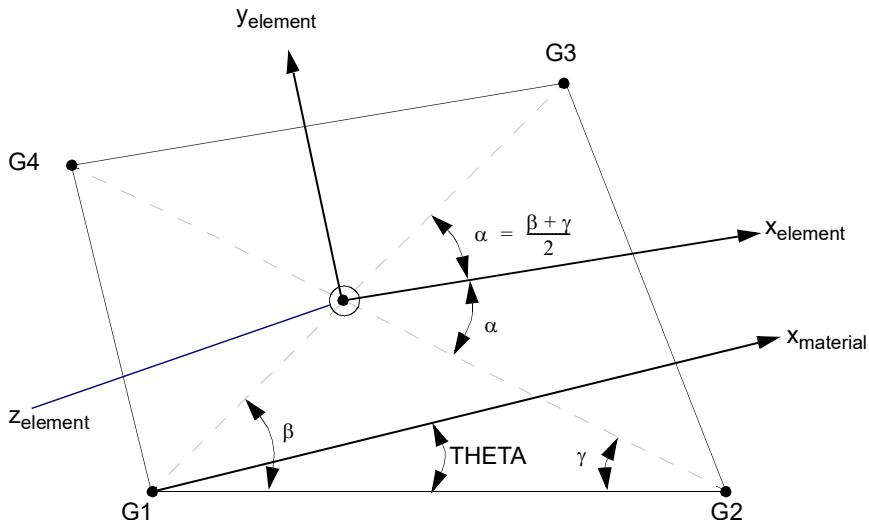


Figure 9-55 CQUADR Element Geometry and Coordinate Systems

8. Elements may be offset from the connection points by means of ZOFFS. Other data, such as material matrices and stress fiber locations, are given relative to the reference plane. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive Z-axis of the element coordinate system. If the ZOFFS field is used, then the MID1 and MID2 fields must be specified on the PSHELL entry referenced both by PID.

Two methods are available for the computation of offsets: original and enhanced. The default method is the original method. The enhanced method is requested by the Bulk Data entry MDLPRM, OFFDEF, LROFF. For options of offsets, please refer to the Bulk Data entry MDLPRM, OFFDEF, option.

For solution sequences that require differential stiffness such as SOL103 (with preloading), SOL105, and SOL400, it is **highly recommended** to use MDLPRM, OFFDEF, LROFF. In SOL103, if the case control ADAMSMNF FLEXBODY=YES or AVLEXB EXBBODY=YES is present, then MDLPRM, OFFDEF, LROFF must be replaced by MDLPRM, OFFDEF, NOMASS.

For SOLs 106, 129, 153, and 159 the differential stiffness for offset vectors will give incorrect results with PARAM, LGDISP, 1. In addition in SOLs 106 and 129 offset vectors will produce incorrect results with thermal loading.

9. In SOL 600, when PCOMP is used, one must define the material property with orientation, MCID>0.

CQUADX**Axisymmetric Quadrilateral Element (Fully Nonlinear or Linear Harmonic)**

Defines an axisymmetric quadrilateral element with up to nine grid points for use in fully nonlinear (i.e., large strain and large rotations) analysis or a linear harmonic or rotordynamic analysis. The element has between four and nine grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CQUADX	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	THETA or MCID					

Example:

CQUADX	111	203	31	74	75	32			
--------	-----	-----	----	----	----	----	--	--	--

Descriptor	Meaning
EID	Element identification number. ($0 < \text{Integer} < 100,000,000$). See Remark 1.
PID	Property identification number of a PLPLANE or PAXSYMH or PLCOMP entry. ($\text{Integer} > 0$). See Remark 2.
G1, G2	Identification numbers of connected corner grid points. Required data for all four grid points. ($\text{Unique Integers} > 0$). See Remark 3., 6., 7.
G3, G4	
G5, G6	Identification numbers of connected edge grid points. Optional data for any or all four grid points. ($\text{Integer} \geq 0$ or blank). See Remark 3., 4., 6., 7.
G7, G8	
G9	Identification number of center grid point. Optional. ($\text{Integer} \geq 0$ or blank). Not used for linear harmonic elements. See Remark 3., 6., 7.
THETA	Material property orientation angle in degrees. THETA is only applicable if PLCOMP referenced, or the PLPLANE entry has an associated PSHLN2 entry which is honored only in SOL 400. For PSHLN2 BEHi=AXSOLID code, THETA is measured relative to the R axis of the element. For PSHLN2 or PLCOMP BEHi=AXCOMP code the THETA value on the element connection entry will be ignored. (Real; Default = 0.0). THETA is ignored if PID refers to a PAXSYMH entry.
MCID	Material coordinate system identification number, MCID is only applicable if PLCOMP referenced, or the PLPLANE entry has an associated PSHLN2 entry which is honored only in SOL 400. The x-axis of the material coordinate system is determined by projecting the x-axis of the MCID coordinate system onto the surface of the element. For PSHLN2 BEHi=AXSOLID the resulting angle is measured relative to the R axis of the element. For PSHLN2 (or PLCOMP) BEHi=AXCOMP code the MCID value on the element connection entry will be ignored. (Integer ≥ 0 ; If blank, then THETA = 0.0 is assumed.)

Remarks:

1. Element identification numbers should be unique with respect to all other element IDs of any kind.
2. If PID refers to a PLPLANE or PLCOMP entry, CQUADX defines an element for use in fully nonlinear analysis. If PID refers to a PAXSYMH entry, CQUADX defines a linear harmonic element for use in rotordynamic or harmonic analysis.
3. Gi must be numbered as shown in [Figure 9-56](#).
4. It is recommended that the edge points be located within the middle third of the edge.
5. The plot codes are specified under the CQUADXF element name in [Item Codes, 1045](#).
6. All Gi must lie on the x-y plane of the basic coordinate system. Stress and strain are output in the basic coordinate system.
7. A concentrated load (e.g., FORCE entry) at Gi is multiplied by the radius to Gi and then applied as a force per unit circumferential length. For example, in order to apply a load of 100 N/m on the circumference at G1, which is located at a radius of 0.5 m, then the magnitude specified on the static load entry must result in:

$$(100 \text{ N/m}) \cdot (0.5 \text{ m}) = 50 \text{ N}$$

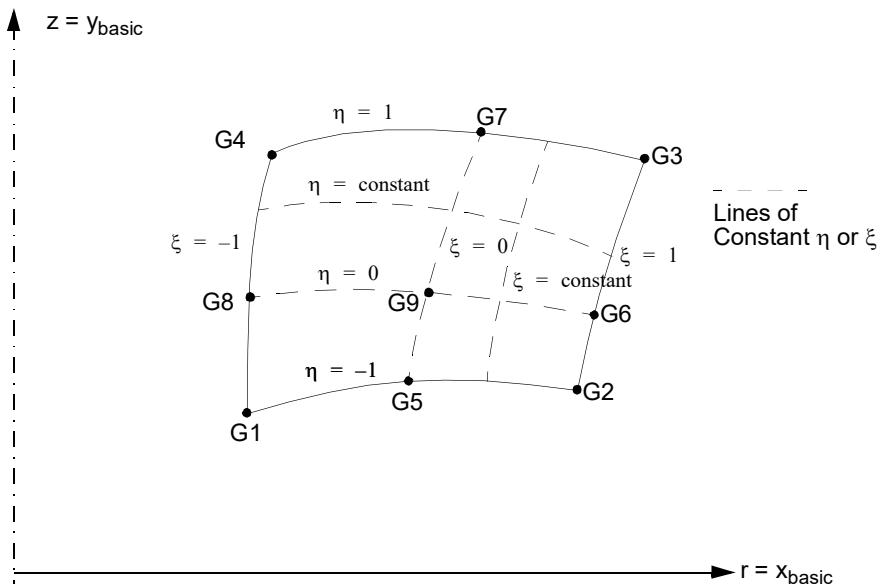


Figure 9-56 CQUADX Element Coordinate System

CRAC2D**Two-Dimensional Crack Tip Element**

Defines a two-dimensional crack tip element.

Format:

1	2	3	4	5	6	7	8	9	10
CRAC2D	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15	G16	G17	G18					

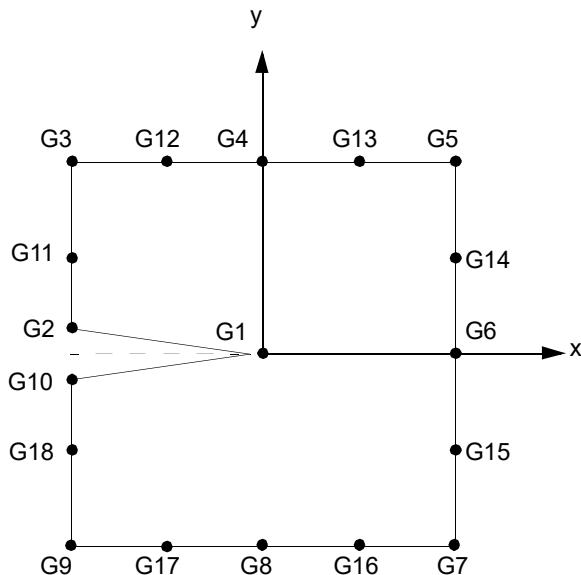
Example:

CRAC2D	114	108	2	5	6	8	7	11	
	12	14	16	17		20	22		

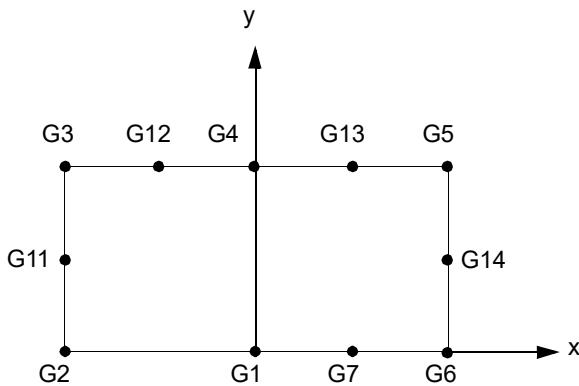
Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PRAC2D entry. (Integer > 0)
Gi	Grid point identification numbers of connection points. (Integer ≥ 0 ; G11 through G18 may be blank.)

Remarks:

1. CRAC2D is a dummy element and requires the presence of this Bulk Data entry:
- | | | | | | | | | | |
|-------|----|---|---|---|--------|--|--|--|--|
| ADUM8 | 18 | 0 | 5 | 0 | CRAC2D | | | | |
|-------|----|---|---|---|--------|--|--|--|--|
2. The element should be planar. Significant deviations will produce fatal errors.
 3. Grid points G1 through G10 are required while grid points G11 through G18 are optional for the quadrilateral form of the element.
 4. The stresses and stress intensity factors are calculated assuming that G2 and G10 are coincident. Deviations from this will produce erroneous results.
 5. For the symmetric half-crack option, grid points G1 through G7 are required while grid points G11 through G14 are optional. Grid points G8 through G10 and G15 through G18 must not be present for this option.
 6. The ordering conventions for the full-crack and half-crack options are shown in [Figure 9-57](#).
 7. The ratio of the element dimensions in the y to x axis shown for the element coordinate system in [Figure 9-57](#) should be in the range 2.0 to 0.5.
 8. The stress output is interpreted as shown in [Crack Tip Elements \(CRAC2D, CRAC3D\)](#) (p. 177) in the *MSC Nastran Reference Guide*.



(a) Full Crack Option



(b) Symmetric Half-Crack Option

Figure 9-57 CRAC2D Element Connection for Full and Symmetric Options

CRAC3D**Three-Dimensional Crack Tip Element**

Defines a three-dimensional crack tip element.

Format:

1	2	3	4	5	6	7	8	9	10
CRAC3D	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15	G16	G17	G18	G19	G20	G21	G22	
	G23	G24	G25	G26	G27	G28	G29	G30	
	G31	G32	G33	G34	G35	G36	G37	G38	
	G39	G40	G41	G42	G43	G44	G45	G46	
	G47	G48	G49	G50	G51	G52	G53	G54	
	G55	G56	G57	G58	G59	G60	G61	G62	
	G63	G64							

Example:

CRAC3D	113	101	2	5	7	8	4	10	
	11	14	15	17		3	6	9	
	12		16		102	105	107	108	
	104	110	111	114	115	117		103	
	106	109	112		116		202	205	
	207	208	204	210	211	214	215	217	
	225	226							

Descriptor	Meaning
EID	Element identification number. ($0 < \text{Integer} < 100,000,000$)
PID	Property identification number of a PRAC3D entry. ($\text{Integer} < 0$)
Gi	Grid point identification numbers of connection points. ($\text{Integer} \geq 0$)

Remarks:

1. CRAC3D is a dummy element and requires the presence of this Bulk Data entry:

ADUM9	64	0	6	0	CRAC3D				
-------	----	---	---	---	--------	--	--	--	--

2. Element identification numbers should be unique with respect to all other element identification numbers.

3. This element, including grid point numbering conventions, is shown in [Figure 9-58](#) and [Figure 9-59](#). Grid points G1 through G10, and G19 through G28 are required; midside and surface grid points G11 through G18, G29 through G36, and G37 through G64 are optional. Either all or none of grid points G37 through G46 should be present. A fatal error message will be issued for partial connectivity.
4. The ratio of the element dimensions in the y to x axis shown for the element coordinate system in [Figure 9-58](#) should be in the range 2.0 to 0.5.
5. For the symmetric half-crack option Grid Points G1 through G7, and G19 through G25 are required, whereas grid points G11 through G14, G29 through G32, and G37 through G42 are optional. Grid points G8 through G10, G15 through G18, G26 through G28, G33 through G36, G43 through G46, G51 through G55, and G60 through G64 should not be specified to invoke this option.
6. It is recommended that both the faces (formed by grid points G2 through G18 and grid points G20 through G36) and the midplane (formed by grid points G37 through G46 and grid points G37 through G46) be planar. It is also recommended that midside grid points G37 through G46 be located within the middle third of the edges.
7. The midside nodes on both the faces should be defined in pairs. For example, if grid point G11 is not defined, then grid point G29 should not be defined and vice versa.
8. The stresses and stress intensity factors are calculated with the assumptions that grid points G2 and G10, G20 and G28, and G38 and G46 are coincident. Deviation from this condition will produce erroneous results.
9. The stress output is interpreted as shown in [Crack Tip Elements \(CRAC2D, CRAC3D\)](#) (p. 177) in the *MSC Nastran Reference Guide*.
10. As depicted in [Figure 9-58](#) and [Figure 9-59](#), the element is a right-handed element. Thus define the vectors $\overline{G1G9}$ and $\overline{G1G3}$, then the cross-product $\overline{G1G9} \times \overline{G1G3}$ points to the face defined by G19, G20, ...

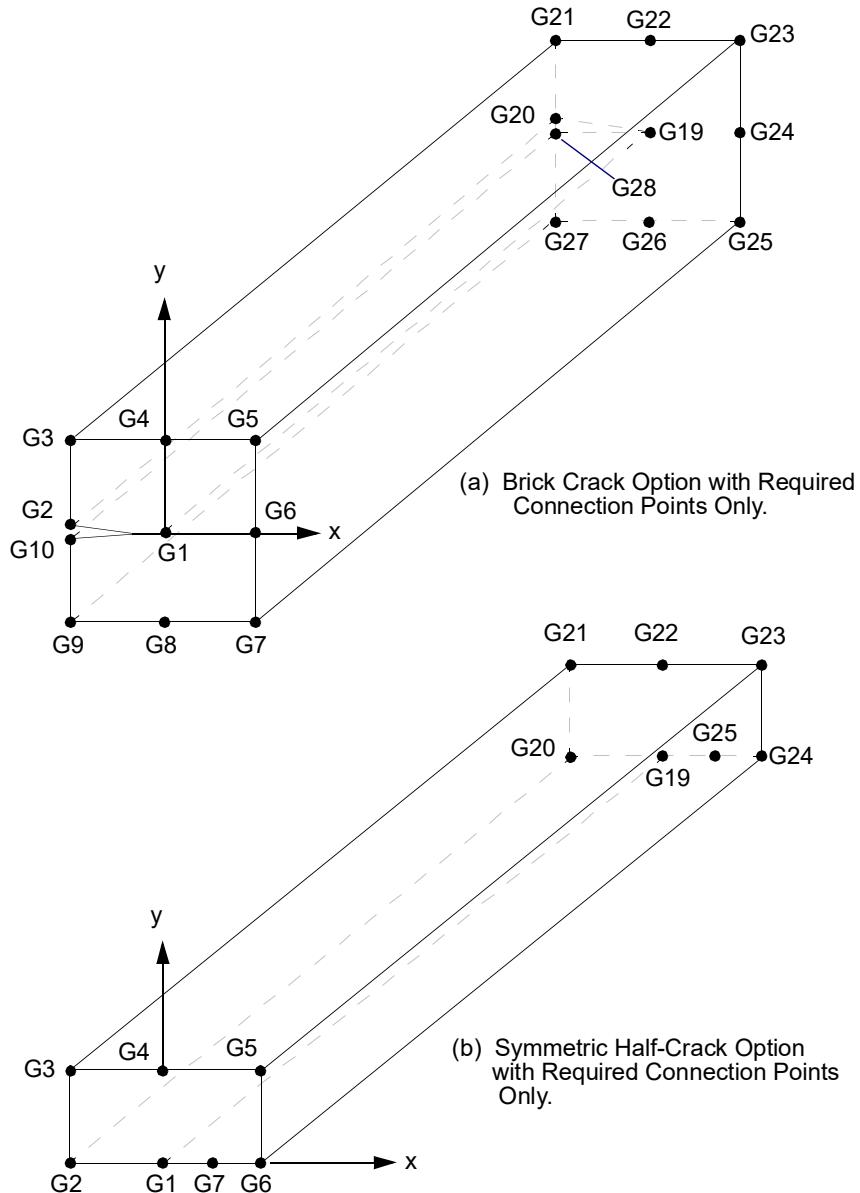


Figure 9-58 CRAC3D Solid Crack Tip Element with Required Connection Points Only

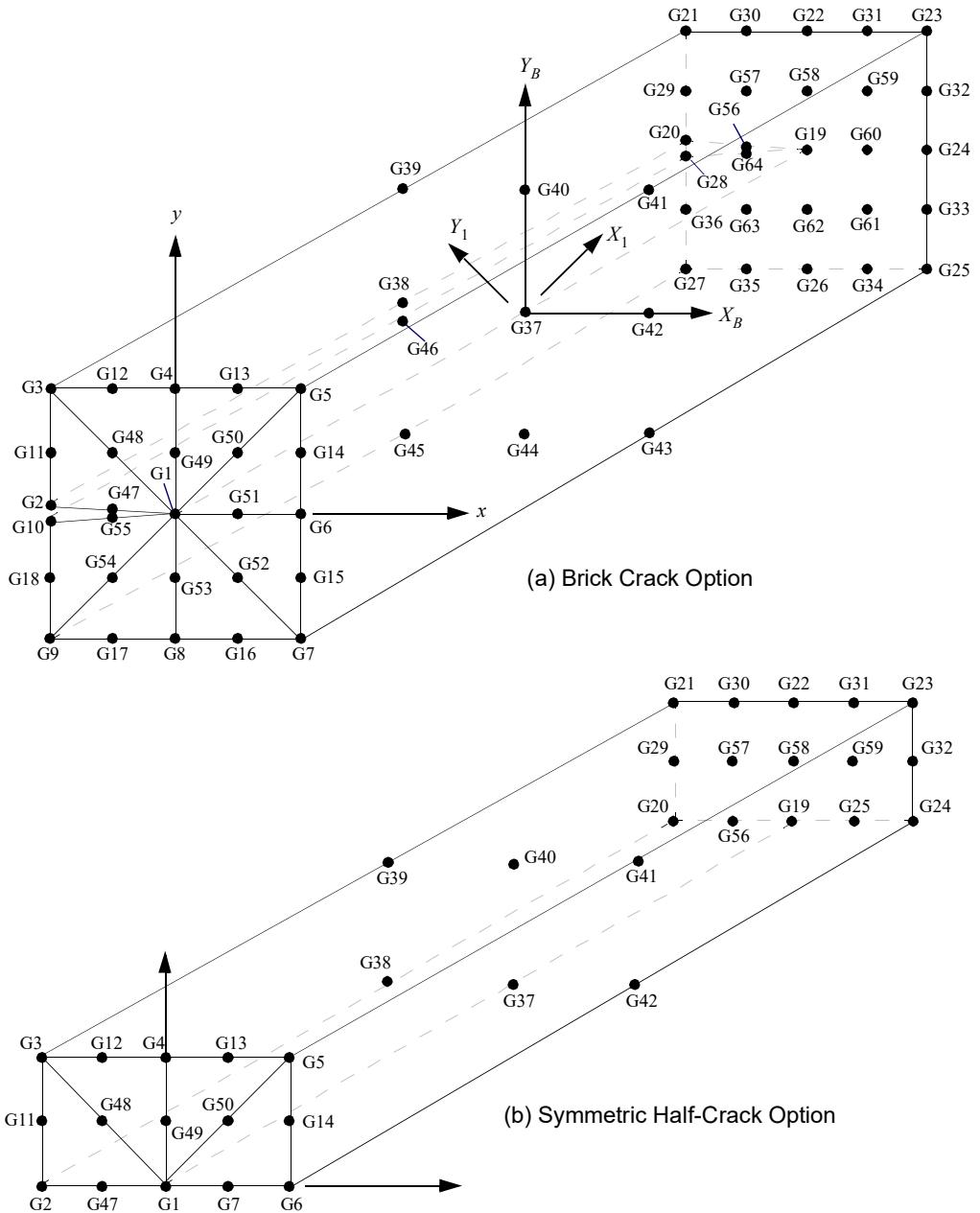


Figure 9-59 CRAC3D Solid Crack Tip Element with All Connection Points

CREEP**Creep Characteristics**

Defines creep characteristics based on experimental data or known empirical creep law. This entry will be activated if a MAT1, MAT2, or MAT9 entry with the same MID is used and the NLPARM entry is prepared for creep analysis. The creep formulation is principally suited for isotropic materials and, in general, when used with anisotropic materials may produce incorrect results. However, slightly anisotropic materials may produce acceptable results.

Format:

1	2	3	4	5	6	7	8	9	10
CREEP	MID	T0	EXP	FORM	TIDKP	TIDCP	TIDCS	THRESH	
	TYPE	a	b	c	d	e	f	g	

Example:

CREEP	8	1100.		CRLAW					
	121	6.985-6	2.444	7.032-4	0.1072	6.73-9	0.1479	3.0	

Descriptor	Meaning
MID	Material identification number of a MAT1, MAT2, or MAT9 entry. (Integer > 0)
T0	Reference temperature at which creep characteristics are defined. See Remark 2. (Real; Default = 0.0)
EXP	Temperature-dependent term, $e^{(-\Delta H/(R \cdot T0))}$, in the creep rate expression. See Remark 2. (0.0 < Real \leq 1.0; Default = 1.0E-9)
FORM	Form of the input data defining creep characteristics. (Character: “CRLAW” for empirical creep law, or “TABLE” for tabular input data of creep model parameters.)
TIDKP, TIDCP, TIDCS	Identification number of a TABLES1 entry, which defines the creep model parameters $K_p(\sigma)$, $C_p(\sigma)$, and $C_s(\sigma)$, respectively. See Remarks 3. through 4. (Integer > 0)
THRESH	Threshold limit for creep process. Threshold stress under which creep does not occur is computed as THRESH multiplied by Young’s modulus. (0.0 < Real < 1.0E-3; Default = 1.0E-5)
TYPE	Identification number of the empirical creep law type. See Remark 1. (Integer: 111, 112, 121, 122, 211, 212, 221, 222, or 300)
a through g	Coefficients of the empirical creep law specified in TYPE. Continuation should not be specified if FORM = “TABLE”. See Remark 1. (Real)

Remarks:

- Two classes of empirical creep law are available.

Creep Law Class 1

The first creep law class is expressed as:

$$\dot{\epsilon}^c(\sigma, t) = A(\sigma)[1 - e^{-R(\sigma)t}] + K(\sigma)t \quad (9-3)$$

Parameters $A(\sigma)$, $R(\sigma)$, and $K(\sigma)$ are specified in the following form, as recommended by Oak Ridge National Laboratory:

Parameter	Function 1	Digit	Function 2	Digit
$A(\sigma)$	$a\sigma^b$	i=1	$ae^{b\sigma}$	i=2
$R(\sigma)$	$ce^{d\sigma}$	j=1	$c\sigma^d$	j=2
$K(\sigma)$	$e \cdot [\sinh(f\sigma)]^g$	k=1	$ee^{f\sigma}$	k=2

TYPE=ijk where i, j, and k are digits equal to 1 or 2, according to the desired function in the table above. For example, TYPE=122 defines $A(\sigma) = a\sigma^b$, $R(\sigma) = c\sigma^d$, and $K(\sigma) = ee^{f\sigma}$.

Creep Law Class 2

The second creep law class (TYPE=300) is expressed as:

$$\dot{\epsilon}^c(\sigma, t) = a\sigma^b t^d \quad (9-4)$$

where the values of b and d must be defined as follows:

The above expression is determined by curve fitting using a Newton-Raphson procedure based on the expressions:

$$\lambda = C_p / K_{pi} = \left(\frac{\bar{\epsilon}^c}{a\bar{\sigma}^b} \right)^{1/d} \quad (9-5)$$

where $\bar{\sigma}$ is the effective stress and the nonlinear equation:

$$\lambda^2 (e^{t_0/\lambda} - 1) - \lambda t_0 - t_0^2 / d = 0$$

Then the creep model parameters are determined by:

$$\frac{1}{C_s} = \frac{\bar{\epsilon}^c d}{t_0 \bar{\sigma}} \left[1 - \frac{\lambda(1-d)}{t_0} \right]$$

$$\frac{1}{K_p} = \left(\frac{\lambda}{t_0} \right)^2 \frac{\bar{\epsilon}^c}{\bar{\sigma}} (1-d)d - \frac{t_0}{C_s} + \frac{\bar{\epsilon}^c}{\bar{\sigma}}$$

$$C_p = \lambda K_p$$

To keep the solution of these expressions stable, the values of b and d are recommended to be defined as follows:

$$1.0 < b < 8.0$$

and

$$0.2 < d < 2.0$$

The coefficient g should be blank if TYPE = 112, 122, 222, or 212 and c, e, f, and g should be blank if TYPE = 300. The coefficients a through g are dependent on the structural units; caution must be exercised to make these units consistent with the rest of the input data.

2. Creep law coefficients a through g are usually determined by least squares fit of experimental data, obtained under a constant temperature. This reference temperature at which creep behavior is characterized must be specified in the T0 field if the temperature of the structure is different from this reference temperature. The conversion of the temperature input ($^{\circ}\text{F}$ or $^{\circ}\text{C}$) to $^{\circ}\text{K}$ (degrees Kelvin) must be specified in the PARAM,TABS entry as follows:

PARAM,TABS,273.16 (If Celsius is used.)

PARAM,TABS,459.69 (If Fahrenheit is used.)

When the correction for the temperature effect is required, the temperature distribution must be defined in the Bulk Data entries (TEMP, TEMPP1 and/or TEMPRB), which are selected by the Case Control command TEMP(LOAD) = SID within the subcase.

From the thermodynamic consideration, the creep rate is expressed as:

$$\dot{\varepsilon}^c = \dot{\varepsilon}_A (e^{-\Delta H / RT}) \quad (9-6)$$

where:

ΔH = energy of activation

R = gas constant ($= 1.98 \text{ cal/mole } ^{\circ}\text{K}$)

T = absolute temperature ($^{\circ}\text{K}$)

$\dot{\varepsilon}_A$ = strain/sec per activation

If the creep characteristics are defined at temperature T0, the creep rate at temperature T is corrected by a factor

$$\frac{\dot{\varepsilon}^c}{\dot{\varepsilon}_o} = \text{EXP}^{\left(\frac{T_0}{T} - 1\right)} \quad (9-7)$$

where:

$\dot{\varepsilon}^c$ = corrected creep rate

$\dot{\varepsilon}_o$ = creep rate at T0

$\text{EXP}^{\left(\frac{T_0}{T} - 1\right)}$ = correction factor

3. Creep model parameters K_p , C_p , and C_s represent parameters of the uniaxial rheological model as shown in [Figure 9-60](#).

Tabular values (X_i , Y_i) in the TABLES1 entry correspond to (σ_i, K_{pi}) , (σ_i, C_{pi}) , and (σ_i, C_{si}) for the input of K_p , C_p , and C_s , respectively. For linear viscoelastic materials, parameters K_p , C_p and C_s are constant and two values of σ_i must be specified for the same value of K_{pi} , C_{pi} and C_{si} .

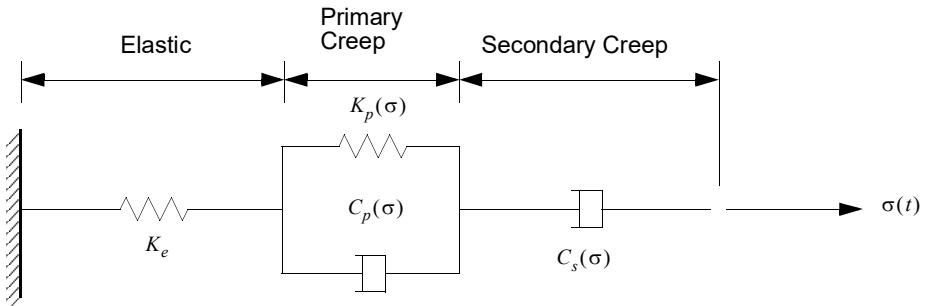


Figure 9-60 CREEP Parameter Idealization

Creep model parameters, as shown in [Figure 9-61](#) through [Figure 9-63](#), must have positive values. If the table look-up results in a negative value, the value will be reset to zero and a warning message (TABLE LOOK-UP RESULTS IN NEGATIVE VALUE OF CREEP MODEL PARAMETER IN ELEMENT ID=*****) will be issued.

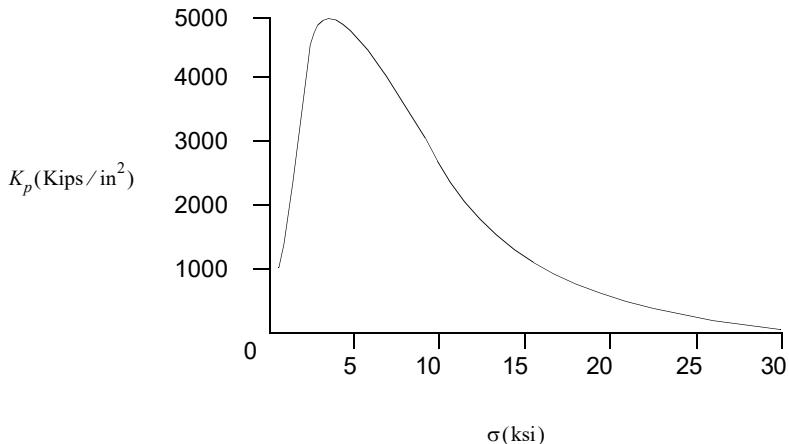
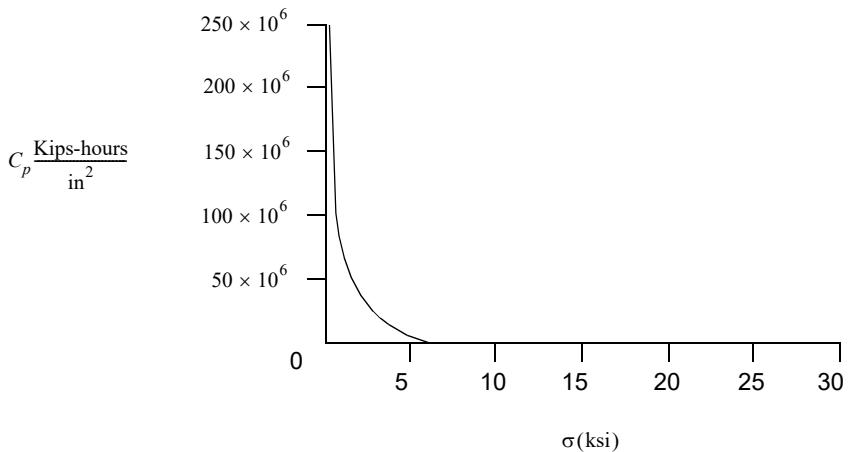
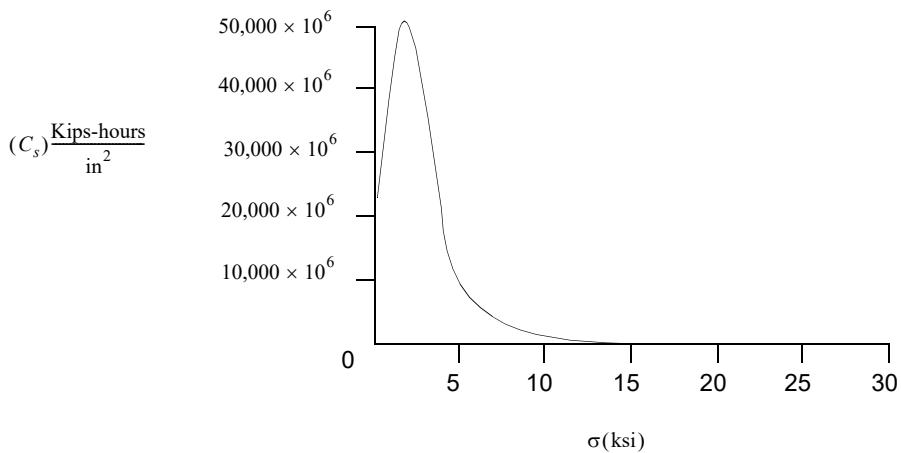


Figure 9-61 K_p Versus σ Example for CREEP

Figure 9-62 C_p Versus σ Example for CREEPFigure 9-63 C_s Versus σ Example for CREEP

4. Creep analysis requires an initial static solution at $t = 0$, which can be obtained by specifying a subcase that requests an NLPARM entry with DT = 0.0.

CROD**Rod Element Connection**

Defines a tension-compression-torsion element.

Format:

1	2	3	4	5	6	7	8	9	10
CROD	EID	PID	G1	G2					

Example:

CROD	12	13	21	23					
------	----	----	----	----	--	--	--	--	--

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PROD entry. (Integer > 0; Default = EID)
G1, G2	Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. See [CONROD, 1567](#) for alternative method of rod definition.
3. Only one element may be defined on a single entry.

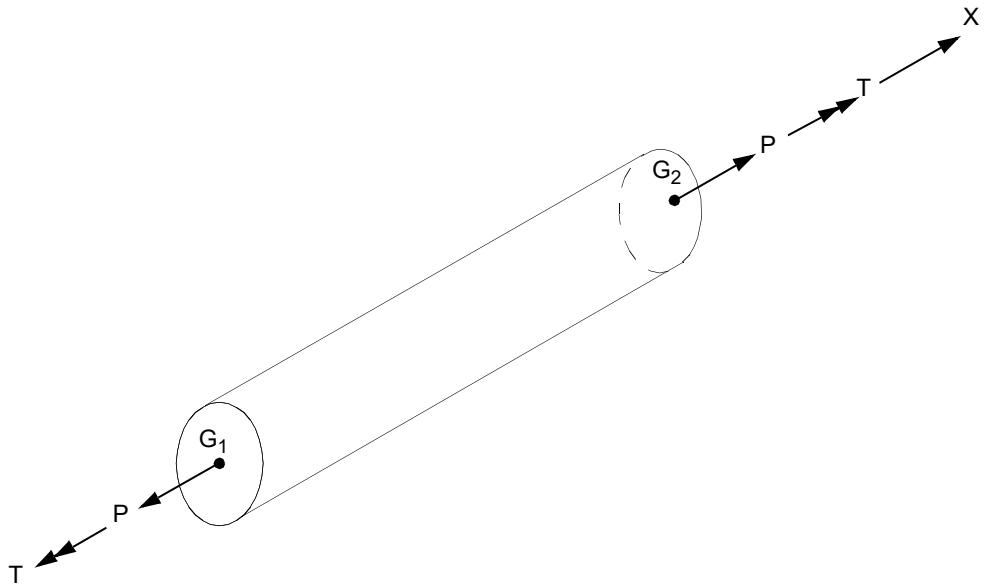


Figure 9-64 CROD Element Internal Forces and Moments

CSEAM

A Shell Patch SEAM Connection

Defines a SEAM connecting two surface patches.

Format:

1	2	3	4	5	6	7	8	9	10
CSEAM	EID	PID	SMLN	CTYPE	IDAS	IDBS	IDAE	IDBE	
	GS	GE							

Alternate Format:

1	2	3	4	5	6	7	8	9	10
CSEAM	EID	PID		CTYPE	IDAS	IDBS	IDAE	IDBE	
	XS	YS	ZS	XE	YE	ZE			

Example:

CSEAM	552	297			43	48			
	30422	77987							

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PSEAM entry. (Integer > 0)
SMLN	SEAM line identification. See Remark 2. (CHAR or blank)
CTYPE	Connectivity search type. (Character) If CTYPE = “PSHELL”, IDAS and IDBS are property identification numbers of PSHELL’s. (Default) If CTYPE = “ELEM”, IDAS and IDBS are element identification numbers.
IDAS, IDBS	Used to define patch A and B or the start of patch A or B for a tailored blank. See Remark 4. (Integer > 0) If CTYPE = “PSHELL”, required property id defining patches A and B. If CTYPE = “PSHELL” and IDAS = IDBS or IDBS = blank the patch will be considered as two-sided and the property identification numbers of PSHELL’s will be the same for both the top and bottom. See Remark 6. If CTYPE = “ELEM”, required element id defining patches A and B. IDAS ≠ IDBS.
IDAE, IDBE	Used to define the end of patch A and the end of patch B for a tailored blank. See Remark 4. (Integer ≥ 0 or blank) If CTYPE = “PSHELL”, property id defining patches A and B. If CTYPE = ‘PSHELL’ and IDAE = IDBE or IDBE=blank the patch will be considered as two-sided and the property identification numbers of PSHELL’s will be the same for both the top and bottom. If CTYPE = “ELEM”, element id defining patches A and B. IDAE ≠ IDBE.

Descriptor	Meaning
GS, GE	Grid ids of piercing points on patches A and B of the Start and End of the SEAM. (Integer > 0)
XS,YS,ZS	Location of the SEAM Start. (Real or blank)
XE,YE,ZE	Location of the SEAM End. (Real or blank)

Remarks:

1. Element ID numbers must be unique with respect to all other element ID numbers.
2. With no embedded blanks any combination of up to eight of the acceptable characters in MSC Nastran may be employed for the SMLN entry.

The seam line will be considered continuous between each connected element and where any two elements have a common face, the faces of the resulting internal CHEXA's will be adjusted to a single common face. If a CSEAM's GS or GE is not common to the GE or GS of any other CSEAM the faces will not be adjusted.

A SMLN cannot have a branch.

3. GS and GE define the start and end points of the SEAM element. At these points and using the value W specified on the PSEAM entry, surface patches A and B are determined. Points are projected onto the surface patches A and B with the four points at end GS and the four points at end GE then used to form faces of a CHEXA element.

The auxiliary points forming the faces of the CHEXA element are then connected to the physical grids of the patches. The number of unique physical grids per patch ranges from a possibility of 6 to 64 grids.

The auxiliary points must have a projection on patches A and B, but they do not have to lie on patch A or B.

A maximum of three shell elements of patch A and three shell elements of patch B can be connected with one CSEAM element, see [Figure 9-65](#).

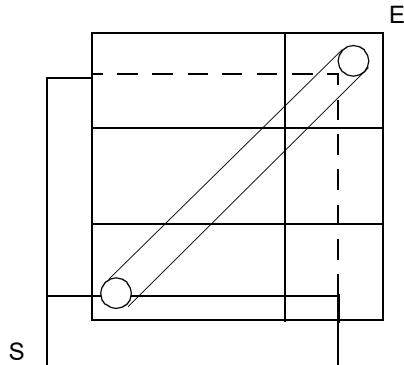


Figure 9-65 Connected Shell Elements for a CSEAM Element

4. For CTYPE = ‘PSHELL’

- If patch A is uniform in thickness, then only its IDAS is needed to define it.
- If patch B is uniform in thickness, then only its IDBS is needed to define it.
- If patch A has stepped tapering, then IDAS and IDAE are used to define it.
- If patch B has stepped tapering, then IDBS and IDBE are used to define it.

5. Projection Algorithms for the CSEAM Elements

Because of complex geometry, the user supplied start point GS may not have a projection SA and SB, and the end point GE may not have a projection EA and EB. Even though these four projection points are found, the program still has to find projections for the eight auxiliary points SA1, SA2, ..., EB2, and EB1 of the HEXA. The default projection strategy can be changed by overwriting the default values of the flags and parameters in the SWLDPRM Bulk Data entry.

a. Find Projections for SA, SB, EA, and EB

For CTYPE = “PSHELL”, the program finds the closest shell grids to GS and GE. The shell elements that are connected to these closest grids are defined as the candidate shell elements. While looping through each candidate shell element to compute the projection of GS and GE onto that element, the program always tries to get the most accurate projection. Even though a projection is found with PROJTOL > 0.0, the program still continues the projection calculations using PROJTOL=0.0. If a projection is found with PROJTOL=0.0, that shell element will be selected as the connecting element. Otherwise, the shell element that gets projection with PROJTOL>0.0 is selected as EIDSA, EIDS, EIDEA, or EIDEB. For CTYPE = “ELEM”, the above processes are skipped, because EIDSA, EIDS, EIDEA, and EIDEB have already been specified by the user.

If GSTOL > 0.0 and the distance GS-SA, GS-SB, GE-EA, or GE-EB is greater than GSTOL, a UFM 7549 is issued and the CSEAM element is rejected.

If the projection of GS or GE lies outside the shell sheet, or the connected shell elements fail the geometry check with GMCHK > 0, the program will issue a UFM and the CSEAM element will be rejected.

If GMCHK > 0, the program checks errors of CSEAM across a cutout or over a corner with elements in plane. The program also computes the angle between the shell normal vectors of EIDSA and EIDEA and the angle between the shell normal vectors of EIDS and EIDEB to check a corner with elements out of plane.

For CTYPE = “PSHELL”, if there is an error detected, the program loops back to compute the other possibility of projection until a correct connection is found or all candidate shell elements are processed. In the latter case, either UFM 7638 (the seam spans a cutout) or UFM 7667 (the seam spans a corner) will be issued. If GMCHK=2, the program also lists all candidate shell elements with their projection status for each connecting type after issuing a UFM. This will help the user to select the correct shell elements for EIDSA, EIDS, EIDEA, and EIDEB.

For CTYPE=“ELEM”, the program only checks errors and issues UFM 7638 or 7667 for the kind of error encountered. No looping back will be performed.

If $GMCHK > 0$ and $GSPROJ \geq 0.0$, the program also computes the angle between the shell normal vectors of EIDSA and EIDS_B and the angle between the shell normal vectors of EIDEA and EIDE_B. A UFM 7595 is issued if the angle between the shell normal vectors is greater than GSPROJ. By default, GSPROJ = 20°, that means the shell patches A and B can be tilted relative to each other by not more than 20°.

- Find Projections of the Eight Auxiliary Points SA1, SA2, ..., EB2, and EB1

After the projections for SA, SB, EA, and EB have been found, eight auxiliary points for an internal hexagonal polygon are formed. If the GS or GE of a CSEAM element is connected to the GE or GS of another CSEAM element, then the internal HEXA elements are adjusted to a common face.

If $GSPROJ \geq 0.0$ and the angle between the face vectors parallel to the thickness direction of the internal HEXA and the normal vector of the shell element that gets projection exceeds GSPROJ, the program will skip picking this shell element and will proceed to process next candidate shell element.

The most common error condition occurs when the seam lies on the edge of the shell patches. Under this situation, half of the seam hangs outside the shell sheets ([Figure 9-66](#)). It is required that each of the eight points has a projection. If at least one point does not have a projection and $GSMOVE > 0$, GS will be moved by $W/2$. Same algorithms apply to end E. The move will be repeated until either all projections are found or the number of moves reaches $GSMOVE$.

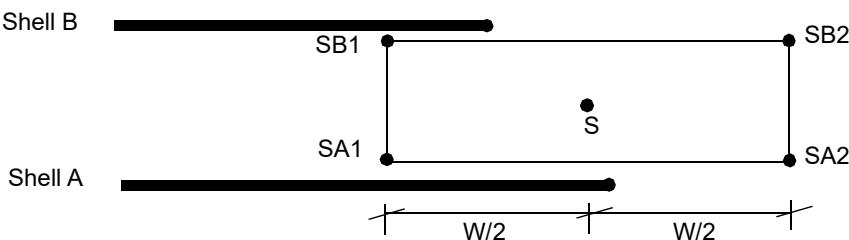


Figure 9-66 Seam Weld at an Edge

- Error Checks by GMCHK Parameter

The GMCHK parameter specified in the SWLDPRM Bulk Data entry checks the errors of CSEAM elements across cutouts or over corners. There are three allowable values of GMCHK.

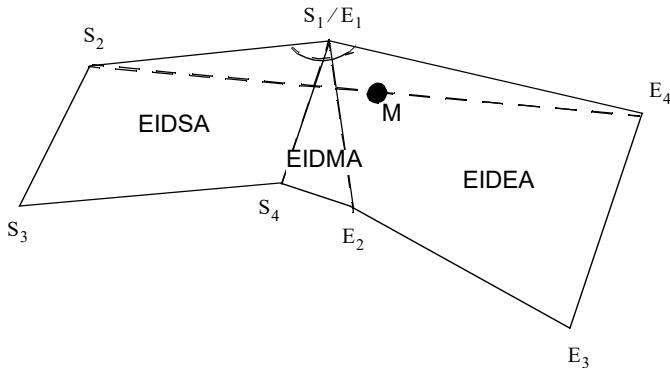
- GMCHK = 0 (Default) Do not check errors
- GMCHK = 1 Check errors
- GMCHK = 2 Check errors and output all candidate shell elements if there is an error encountered

If GMCHK is turned on, Nastran will perform the following checking while searching for the projected shell elements. Note that EIDSA is the shell element that gets projection from GS on shell A; EIDEA is the shell element that gets projection from GE on shell A. Same algorithms are applied to EIDS_B and EIDE_B for shell B.

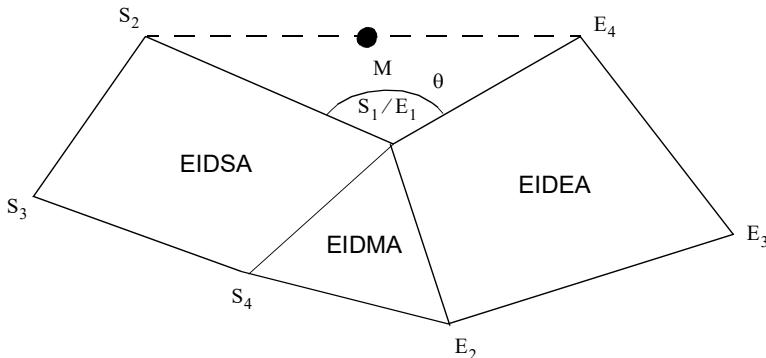
- Check the CSEAM Across a Cutout or Over a Corner with Elements in Plane
 - If EIDSA is equal to EIDEA, the seam lies within one element. No checks are required.

- If EIDSA and EIDEA share two corner grids, these elements are adjacent. No checks are required.
- If EIDSA and EIDEA share only one corner grid, the seam is over a corner. There are two exceptions:

There exists a shell element (EIDMA) that shares two corner grids with EIDSA and EIDEA. Also, either the angle θ between vector S_1S_2 and vector E_1E_4 is greater than CNRAGLI degrees or the middle point (M) of line segment S_2E_4 projects to EIDSA, EIDMA, or EIDEA.

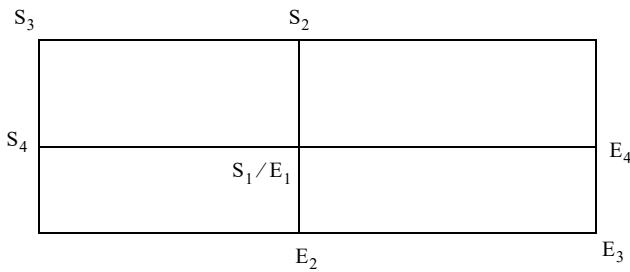


This model is acceptable - CONVEX ($>$ CNRAGLI).



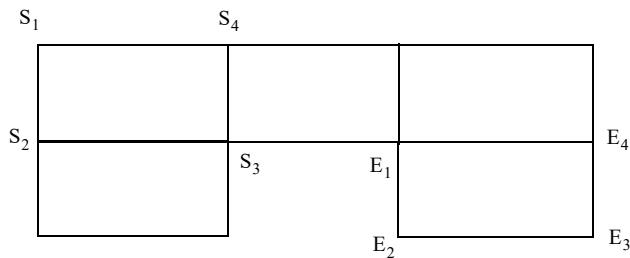
This model fails - not CONVEX ($<$ CNRAGLI and point M does not project to EIDSA, EIDMA or EIDEA).

This shared grid is a shell grid of another two different shell elements.

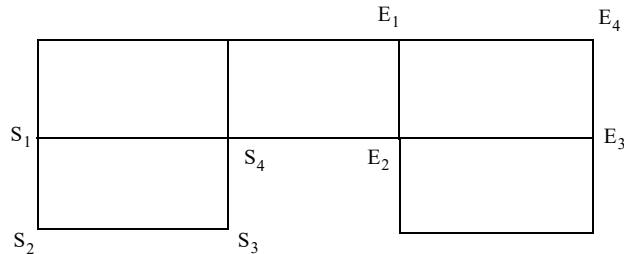


- If EIDSA and EIDEA do not share any corner grid, Nastran will check if there is an element (EIDMA) lying between EIDSA and EIDEA. EIDMA must share two corner grids with EIDSA and another different one corner grid with EIDEA, or vice versa. The following five examples demonstrate the acceptable and failed cases.

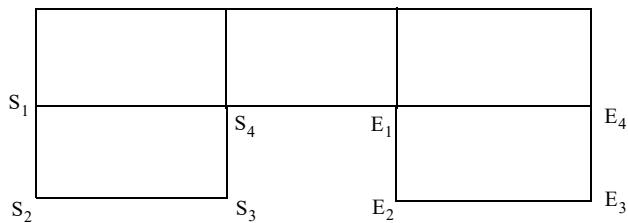
EIDMA shares one edge with EIDSA and shares one corner grid with EIDEA. This case is acceptable.



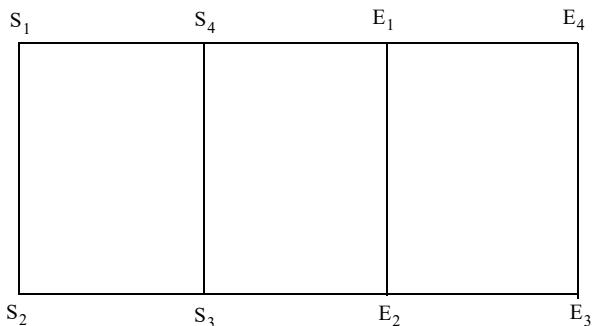
EIDMA shares one edge with IEDEA and shares one corner grid with EIDSA. This case is acceptable.



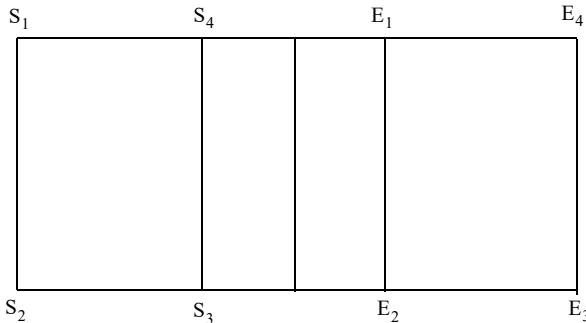
EIDMA shares one corner grid with EIDSA and shares another corner grid with EIDEA. An error is detected because the seam spans a cutout.



EIDMA shares one edge with EIDSA and shares another edge with EIDEA. This case is acceptable.

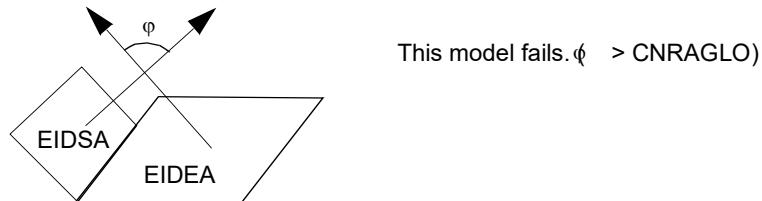


There does not exist a single element that shares an edge or corner grid with EIDSA or EIDEA. An error is detected because the length of the seam spans more than three elements.



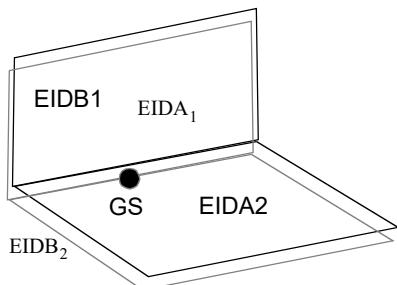
e. Check the CSEAM Over a Corner with Elements Out of Plane

The CNRAGLO parameter is used to check the error of a seam over a corner with EIDSA and EIDEA not lying on a same plane. An error is detected if the angle φ between the shell normal vectors of EIDSA and EIDEA is greater than CNRAGLO. The default value of CNRAGLO is 20° . No angles will be checked if CNRAGLO = -1.



f. Modeling Guidelines

When there exist multiple pairs of connections, it is recommended that either the GMCHK and GSPROJ flags be turned on to filter out tilted connections or the ELEM option be used to specify the IDs of the connected shell elements directly. For example, if EIDA1 is connected to EIDB2 or EIDA2 is connected to EIDB1, the element tangent vectors will be computed wrong and the auxiliary points will not be able to find connected shell elements.



6. The projection algorithm for the two-sided option will be the same as in Remark 5. above once the two patches A and B have been established. The program will find the closest shell grids to GS and GE as usual for candidate shell elements for patch A.

It will compute the normal for the candidate patch A (similar for GE) and for the candidate patch B. If the normals are approximately aligned (within a tolerance) the algorithm will proceed as in Remark 5.

If the normals of the candidate patch's A and B do not align within a specified tolerance, the algorithm will use another set of pairs of grids for candidate patches to find a new patch A and B. If their normals align within a specified tolerance it will proceed as in Remark 5. If after processing all reasonable pairs of patches, no alignment of normals are found or the patches A and B at GE have different normal alignment from the patches A and B at GB, a user fatal will be issued.

Caution:

For the two-sided option, GS and GB must lie between patches A and B. Also, the shell elements that get projections from GS/GE cannot share a common shell grid. This option always selects the patch with the shell grids closest to GS/GE as patch A. Avoid having GS/GE exactly midway between the two patches.

7. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer subsstep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 9 for that entry.
8. The CSEAM contributes to MASS if its PSEAM entry has an associated MATi entry with a non-zero density. PARAM,COUPMASS effects the mass calculation. In SOL400, the behavior of this element in regard to large rotation is affected by the Case Control Command Rigid.
9. If partitioned superelements are present, then CSEAM is supported in the main Bulk Data section only.

CSET**Free Boundary Degrees-of-Freedom**

Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

1	2	3	4	5	6	7	8	9	10
CSET	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Example:

CSET	124	1	5	23	6	16			
------	-----	---	---	----	---	----	--	--	--

Descriptor	Meaning
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component number. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points. No embedded blanks.)

Remarks:

1. CSET and BNDFREE entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BSETi/BNDFIXi or CSETi/BNDFREE/BNDFRE1 entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi/BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are only CSETi/BNDFREE/BNDFRE1 entries present or both BSETi/BNDFIXi and CSETi/BNDFREE/BNDFRE1 entries present, the c-set degrees-of-freedom are defined by the CSETi/BNDFREE/BNDFRE1 entries, and any remaining a-set points are placed in the b-set.
3. Degrees-of-freedom specified on CSETi/BNDFREE/BNDFRE1 entries form members of the mutually exclusive c-set. They may not be specified on other entries that define mutually exclusive sets. See the [Degree-of-Freedom Sets, 1111](#) for a list of these entries.
4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
 - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
 - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

CSET1

Free Boundary Degrees-of-Freedom, Alternate Form of CSET Entry

Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

1	2	3	4	5	6	7	8	9	10
CSET1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	ID10	-etc.-					

Example:

CSET1	124	1	5	7	6	9	12	122	
	127								

Alternate Formats and Examples:

CSET1	C	ID1	"THRU"	ID2					
CSET1	3	6	THRU	32					

CSET1		"ALL"							
CSET1		ALL							

Descriptor	Meaning
C	Component numbers. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.)
IDi	Grid or scalar point identification numbers. (Integer > 0; For "THRU" option, ID1 < ID2)

Remarks:

1. CSET1 and BNDFRE1 entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BSETi/BNDFIXi or CSETi/BNDFREE/BNDFRE1 entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi/BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are only CSETi/BNDFREE/BNDFRE1 entries present or both BSETi/BNDFIXi and CSETi/BNDFREE/BNDFRE1 entries present, the c-set degrees-of-freedom are defined by the CSETi/BNDFREE/BNDFRE1 entries, and any remaining a-set points are placed in the b-set.
3. Degrees-of-freedom specified on CSETi/BNDFREE/BNDFRE1 entries form members of the mutually exclusive c-set. They may not be specified on other entries that define mutually exclusive sets. See the [Degree-of-Freedom Sets, 1111](#) for a list of these entries.

4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
 - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
 - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

CSHEAR

Shear Panel Element Connection

Defines a shear panel element and effective extensional stiffener rods.

Format:

1	2	3	4	5	6	7	8	9	10
CSHEAR	EID	PID	G1	G2	G3	G4			

Example:

CSHEAR	3	6	1	5	3	7			
--------	---	---	---	---	---	---	--	--	--

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PSHEAR entry. (Integer > 0; Default = EID)
Gi	Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2 ≠ G3 ≠ G4)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Grid points G1 through G4 must be ordered consecutively around the perimeter of the element.
3. All interior angles must be less than 180°.

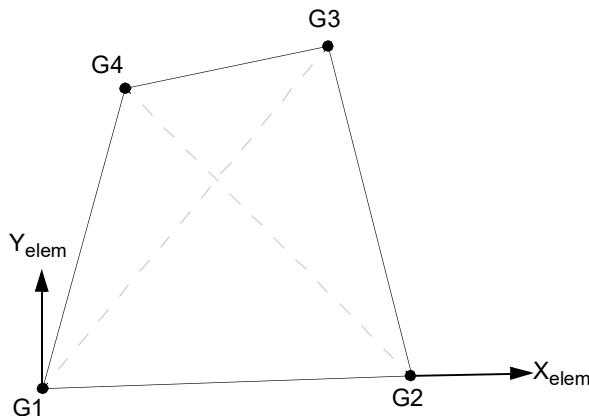


Figure 9-67 CSHEAR Element Connection and Coordinate System

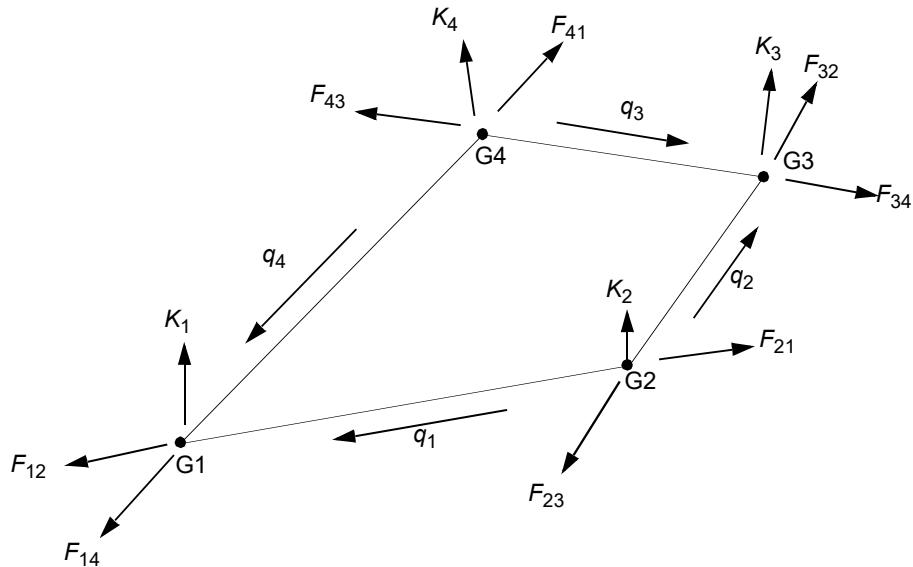
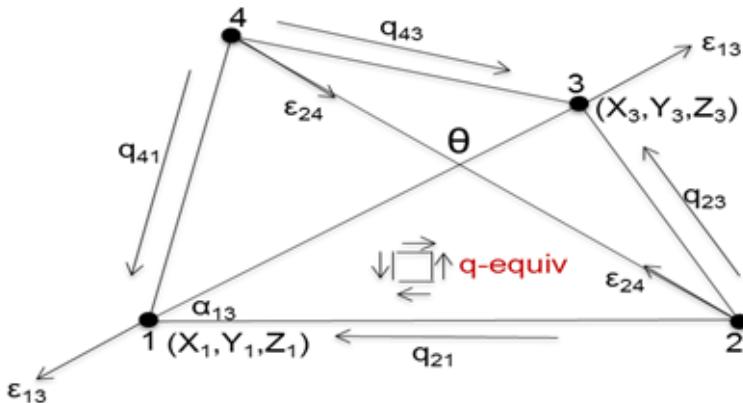


Figure 9-68 CSHEAR Element Corner Forces and Shear Flows

- The parameter entry MDLPRM,SHEARP,GARVEY(default) selects the standard Garvey shear panel. MDLPRM,SHEARP,HARDER selects the Harder shear panel: The Harder panel is based on the following:

Consider the following quadrilateral shear panel.



The shear stress τ is related to the shear strain by the relationship $\tau = G\gamma$ where G is the shear modulus.

Determining the value for shear for the general quadrilateral shown is a heuristic process. For the Harder element it is determined in the following manner. The strain along a diagonal such as L13 is found from the expression (or Mohr's circle):

$$\varepsilon_{13} = \varepsilon_x \cos^2 \alpha_{13} + \varepsilon_x \sin^2 \alpha_{13} + \gamma_{xy} \cos \alpha_{13} \sin \alpha_{13}$$

With a similar expression for ε_{24} , where $\alpha_{24} = 180 - (\alpha_{13} + \theta)$. For shear panels, the direct strains are negligible compared to the shear strain. For a rectangular shear panel $\alpha_{24} = \alpha_{13}$ and we have

$$\Delta\varepsilon = \varepsilon_{13} - \varepsilon_{24} = \gamma \sin \theta \text{ or } \gamma = (\varepsilon_{13} - \varepsilon_{24}) / (\sin \theta).$$

In the above, the subscripts have been dropped from the term for shear strain. The Harder shear panel uses this definition for the measure of shear strain in the general quadrilateral. For a rectangular panel it is exact. For a reasonable panel it is within an error of < 1 degree. To insure reasonableness, all the geometry checks required by the Garvey element are still performed.

Stress Recovery:

The output for either the Garvey or the Harder panel is the same with the following exception:

MAX SHEAR	AVG SHEAR	SHEARP
MAX($\tau_1, \tau_2, \tau_3, \tau_4$)	MAX($\tau_1, \tau_2, \tau_3, \tau_4$) / 4	GARVEY
MAX($\tau_1, \tau_2, \tau_3, \tau_4$)	(q-Equiv) / t	HARDER

For both Garvey and Harder panels, q-Equiv is the shear flow on which all stress calculations are based.

CSLOT3**Three Point Slot Element Connection**

Defines an element connecting three points that solve the wave equation in two dimensions. Used in the acoustic cavity analysis for the definition of evenly spaced radial slots.

Format:

1	2	3	4	5	6	7	8	9	10
CSLOT3	EID	IDS1	IDS2	IDS3		RHO	B	M	

Example:

CSLOT3	100	1	3	2		3.0-3		6	
--------	-----	---	---	---	--	-------	--	---	--

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
IDSi	Identification number of connected GRIDS points. (Integer > 0)
RHO	Fluid density in mass units. (Real > 0.0; Default is the value of RHOD on the AXSLOT entry)
B	Fluid bulk modulus. (Real \geq 0.0; Default is the value of BD on the AXSLOT entry)
M	Number of slots in circumferential direction. (Integer \geq 0; Default is the value of MD on the AXSLOT entry)

Remarks:

1. CSLOT3 is allowed only if an AXSLOT entry is also present.
2. This element identification number (EID) must be unique with respect to all other fluid or structural elements.
3. If RHO, B, or M are blank, then the RHOD, BD, or MD fields on the AXSLOT entry must be specified.
4. This element generates three plot elements, connecting points IDS1 to IDS2, IDS2 to IDS3, and IDS3 to IDS1.
5. If B=0.0, then the slot is considered to be an incompressible fluid.
6. If M=0, then no matrices for CSLOT3 elements are generated.

CSLOT4**Four Point Slot Element Connection**

Defines an element connecting four points that solve the wave equation in two dimensions. Used in acoustic cavity analysis for the definition of evenly spaced radial slots.

Format:

1	2	3	4	5	6	7	8	9	10
CSLOT4	EID	IDS1	IDS2	IDS3	IDS4	RHO	B	M	

Example:

CSLOT4	101	1	3	2	4		6.2+4	3	
--------	-----	---	---	---	---	--	-------	---	--

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
IDSi	Identification number of connected GRIDS points. (Integer > 0)
RHO	Fluid density in mass units. (Real > 0.0; Default is the value of RHOD on the AXSLOT entry.)
B	Fluid bulk modulus. (Real \geq 0.0; Default is the value of BD on the AXSLOT entry.)
M	Number of slots in circumferential direction. (Integer \geq 0; Default is the value of MD on the AXSLOT entry.)

Remarks:

1. This entry is allowed only if an AXSLOT entry is also present.
2. This element identification number (EID) must be unique with respect to all other fluid or structural elements.
3. If RHO, B, or M are blank, then the RHOD, BD, or MD fields on the AXSLOT entry must be specified.
4. This element generates four plot elements connecting points IDS1 to IDS2, IDS2 to IDS3, IDS3 to IDS4, and IDS4 to IDS1.
5. If B = 0.0, then the slot is considered to be an incompressible fluid.
6. If M = 0, then no matrices for CSLOT4 elements are generated.

CSPR**Springs With Offsets**

Springs for use in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
CSPR	EID	PID	G1	G2					

Example:

CSPR	1	22	456	457					
------	---	----	-----	-----	--	--	--	--	--

Descriptor	Meaning	Type	Default
EID	Element ID. A unique number has to be used.	I > 0	Required
PID	Property ID of PSPRMAT entry	I > 0	Required
G1	Gridpoint 1	I > 0	Required
G2	Gridpoint 2.	I > 0	Required

CSSCHD**Aerodynamic Control Surface Schedule Input**

Defines a scheduled control surface deflection as a function of Mach number and angle of attack.

Format:

1	2	3	4	5	6	7	8	9	10
CSSCHD	SID	AESID	LALPHA	LMACH	LSCHD				

Example:

CSSCHD	5	50	12	15	25				
--------	---	----	----	----	----	--	--	--	--

Descriptor	Meaning
SID	Set identification number. (Integer > 0)
AESID	ID of an AESURF Bulk Data entry to which the schedule is being attached.
LALPHA	ID of an AEFACT Bulk Data entry containing a list of angles of attack (in radians) at which schedule information is provided. (Integer > 0: Default = no angle information provided.)
LMACH	ID of an AEFACT Bulk Data entry containing a list of Mach numbers at which schedule information is provided. (Integer > 0; Default = no Mach information provided)
LSCHD	ID of an AEFACT Bulk Data entry which contains the scheduling information. See Remarks 4. and 5. (Integer > 0; no Default)

Remarks:

1. Control system schedules must be selected with the Case Control command CSSCHD = SID.
2. The AESID cannot appear on an AELINK or TRIM Bulk Data entry for the same subcase.
3. The control surface deflection is computed using a linear interpolation for the Mach number provided on the associated TRIM entry and the angle of attack derived as part of the trim calculation.
4. The LSCHD data are provided as a list of deflections (in radians) as a function of Mach numbers and angles of attack. If there are NMACH Mach numbers and NALPHA angles of attack, the first NALPHA deflections are for the first Mach number, the next NALPHA are for the second Mach number, and so on, until the last NALPHA deflections are for the final Mach number.
5. If LALPHA is blank, LSCHD contains NMACH deflections to define the Mach schedule. If LMACH is blank, LSCHD contains NALPHA deflections to define the angle of attack schedule.
6. LALPHA and LMACH cannot be simultaneously blank. If LALPHA or LMACH are not blank, at least two values of angle of attack or Mach number must be defined in order to perform interpolation.
7. If the Mach number or angle of attack is outside the range specified by the tabulated values, the value at the table end is used. That is, data are not extrapolated.

CSSHL**Solid Shell Element Connection**

Defines a connection for a Solid Shell with 6 or 8 grid points in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
CSSHL	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8							

Example:

CSSHL	44	11	1	2	3	4	5	6	quad
	7	8							

CSSHL	51	22	11	12	13		21	22	tria
	23								

CSSHL	51	22	11	12	13	13	21	22	tria
	23	23							

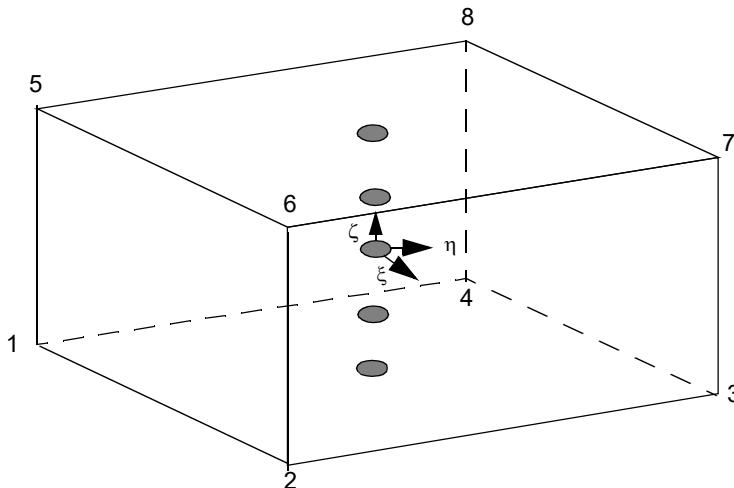
(Note: the 2nd and 3rd examples are equivalent to each other.)

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000; Required)
PID	Property identification of a PSSH entry. (Integer > 0; Required)
Gi	Grid point identification number of connection points. (Integer U or blank, for quad shapes all 8 values are required, for triangle shapes only G4 and G8 may be left blank in which case G4=G3 and G8=G7.)

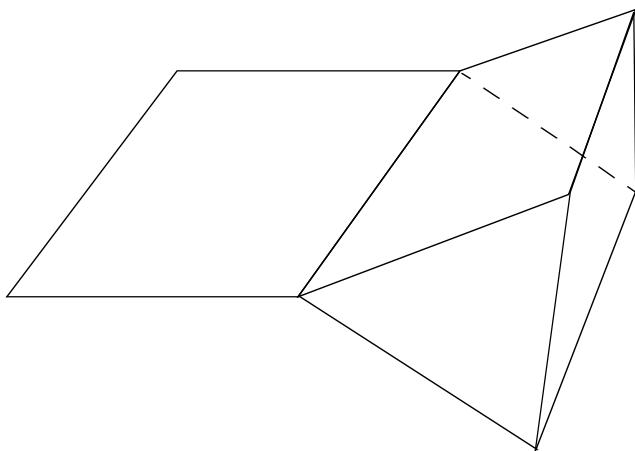
Remarks:

1. This element can degenerate to a triangle either by leaving G4 and G8 blank or by entering G4=G3 and G8=G7 (see 2nd and 3rd examples).
2. This element is usually only used when contact on each of the shell is anticipated.
3. Mid-side nodes are not available for this element.

4. Grid point ordering is shown in the following figure.



5. 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.
6. The number of integration points through the thickness is given by PARAM,MARCSLHT
7. This element may be collapsed to a triangular solid shell to attach to a standard shell such as CQUAD4 as follows:



8. This element is not currently available with Total Lagrange, finite strain plasticity or hyperelastic materials.

CSSHLH**CHEXA to Solid Shell Element Connection**

Defines conversion of CHEXA elements to Solid Shell elements in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
CSSHLH	EID1	PID	EID2	EID3	EID4	EID5	EID6		

Example:

CSSHLH	44	11	54	200	250				
--------	----	----	----	-----	-----	--	--	--	--

Descriptor	Meaning
EIDi	CHEXA element identification numbers. See Remark 1. (0 < Integer < 100,000,000; no Default; EID1 is required)
PID	Property identification of a PSSH entry to be used with all the elements with EIDi. (Integer > 0; Required)

1. All CHEXA elements in the range EID1 to EID2 will be converted to solid shells. The original CHEXA elements will be deleted and the solid shell elements will have the same element ID's as the original CHEXA elements.
2. All CHEXA elements in the range EID3 to EID4 as well as EID5 to EID6 will be converted to solid shells if these fields are entered.
3. Mid-side nodes are not allowed.
4. Elements for this entry are mapped to Marc element type 185.
5. Please see the remarks for CSSHL for additional items.
6. The PSSH entries associated with CSSHLH are not generated automatically. They must be input using a GUI or with a text editor.

CSSHLM**CHEXA or CPENTA to Solid Shell Material Conversion**

Defines conversion of CHEXA or CPENTA elements described by material ID to Solid Shell elements in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
CSSHLM	MID1		MID2	MID3	MID4	MID5	MID6	MID7	
	MID8	MID9							

Example:

CSSHLM	100		200	700	90250				
--------	-----	--	-----	-----	-------	--	--	--	--

Descriptor	Meaning
MIDI	Material identification numbers. See Remark 1. (Integer > 0; no Default; MID1 is required)

Remarks:

1. All CHEXA and CPENTA elements with the MIDI specified will be converted to solid shells. The original CHEXA and CPENTA elements will be deleted and the solid shell elements will have the same element ID's as the original elements.
2. All PSOLID entries which reference all MIDI values will automatically be converted to PSSHL entries retaining the MID and CORDM values. The PSSHL property ID's will be the same as the PSOLID ID's plus IPOFF. The value of IPOFF should not be zero since the original PSOLID entries are not deleted and all property ID's should normally be unique.
3. Mid-side nodes are not allowed.
4. Elements for this entry are mapped to Marc element type 185.
5. Please see the remarks for CSSHL for additional items.
6. If this entry is used, CSSHLH and CSSHLP entries should not be used.
7. See parameters MCSSHLCK and MCSSHORR for additional options for this entry.

CSSHLH**CPENTA to Solid Shell Element Connection**

Defines conversion of CPENTA elements to Solid Shell elements in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
CSSHLH	EID1	PID	EID2	EID3	EID4	EID5	EID6		

Example:

CSSHLH	44	11	54	200	250				
--------	----	----	----	-----	-----	--	--	--	--

Descriptor	Meaning
EIDi	CPENTA element identification numbers. See Remark 1. (0 < Integer < 100,000,000; no Default; EID1 is required)
PID	Property identification of a PSSH entry to be used with all the elements with EIDi. (Integer > 0; Required)

Remarks:

1. All CPENTA elements in the range EID1 to EID2 will be converted to solid shells. The original CPENTA elements will be deleted and the solid shell elements will have the same element ID's as the original CPENTA elements.
2. All CPENTA elements in the range EID3 to EID4 as well as EID5 to EID6 will be converted to solid shells if these fields are entered.
3. The solid shell elements will be collapsed Marc element 185 type elements with the 3rd and 4th grids set to G3 of the CPENTA and the 7th and 8th grids set to the 6th grid of the CPENTA.
4. Mid-side nodes are not allowed.
5. Please see the remarks for CSSHL for additional items.
6. The PSSH entries associated with CSSHLH are not generated automatically. They must be input using a GUI or with a text editor.

CSUPER**Secondary Superelement Connection**

Defines the grid or scalar point connections for identical or mirror image superelements or superelements from an external source. These are all known as secondary superelements.

Format:

1	2	3	4	5	6	7	8	9	10
CSUPER	SSID	PSID	GP1	GP2	GP3	GP4	GP5	GP6	
	GP7	GP8	-etc.-						

Example:

CSUPER	120003	21	3	6	4	10			
--------	--------	----	---	---	---	----	--	--	--

Descriptor	Meaning
SSID	Coded identification number for secondary superelement. See Remark 1. (Integer > 0)
PSID	Identification number for referenced primary superelement. See Remark 2. (Integer > 0 or blank)
GPi	Grid or scalar point identification numbers of the exterior points of the secondary superelement. See Remark 3. (Integer > 0)

Remarks:

- The value of SSID is written in the form XXX0000 + n, where n is the referenced secondary superelement identification number and n must be less than 10000 and XXX is a displacement component sign reversal code as follows:
The sign reversal code specifies the displacement component(s) normal to the plane of the mirror through which the reflection is to be made

Blank or 0 no reversal for identical superelement. If PSID is preceded by a minus sign and there is no xxx code on SSID, then a z-reversal mirror is generated.

1	x-reversal	}	Mirror Images
2	y-reversal		
3	z-reversal		
12	x and y-reversal	}	Identical Images
23	y and z-reversal		
31	z and x-reversal		
123	x, y and z-reversal	}	Mirror Images

2. If PSID = 0 or blank, the superelement boundary matrices are obtained from an external source (such as a database or external file). See also PARAM, [EXTDRUNT, 821](#).
If PSID ≠ 0, the secondary superelement is identical to, or is a mirror image of, a primary superelement.
3. For identical or mirror image superelements, the grid point IDs, GPi, may appear in any order. However, if they are not in the same order as the external GRIDs of the primary superelement, then the SEQSEP entry is also required. In case of external superelements, the GRID IDs must be in the order that the terms in the associated matrices occur in.
4. Image superelements and their primaries must be congruent. The identical or mirror image superelement must have the same number of exterior grid points as its primary superelement. The exterior grid points of the image superelement must have the same relative location to each other as do the corresponding points of the primary superelement. The global coordinate directions of each exterior grid point of the image superelement must have the same relative alignment as those of the corresponding grid points of the primary superelement. If congruency is not satisfied because of round-off, then the tolerance may be adjusted with PARAM,CONFAC or DIAG 37.
5. For superelements from an external source, please refer to PARAMS [EXTDR, 821](#), [EXTDRUNT, 821](#) and [EXTUNIT, 824](#).

CSUPEXT**Superelement Exterior Point Definition**

Assigns exterior points to a superelement.

Format:

1	2	3	4	5	6	7	8	9	10
CSUPEXT	SEID	GP1	GP2	GP3	GP4	GP5	GP6	GP7	

Example:

CSUPEXT	2	147	562	937					
---------	---	-----	-----	-----	--	--	--	--	--

Alternate Format and Example:

CSUPEXT	SEID	GP1	"THRU"	GP2					
---------	------	-----	--------	-----	--	--	--	--	--

CSUPEXT	5	12006	THRU	12050					
---------	---	-------	------	-------	--	--	--	--	--

Descriptor	Meaning
SEID	Identification number of a primary superelement. (Integer > 0)
GPi	Grid or scalar point identification number in the downstream superelement or residual structure. (Integer > 0 or "THRU"; for "THRU" option, GP1 < GP2)

Remarks:

1. Grid or scalar points are connected (that is, are exterior) to a superelement only if they are connected by structural, rigid, or plot elements. MPC entries are not regarded as elements. This entry is a means of providing connectivity for this purpose.
2. Open sets are allowed with the "THRU" option.
3. Scalar points may be interior to the residual structure (SEID = 0) only.
4. This entry may be applied only to the primary superelements. The CSUPER entry is used for secondary superelements (identical image, mirror image, and external superelements).

CTETRA

Four-Sided Solid Element Connection

Defines the connections of the four-sided solid element with four to ten grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CTETRA	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10					

Example:

CTETRA	112	2	3	15	14	4	103	115	
	5	16	8	27					

Descriptor	Meaning	Type	Default
EID	Element identification number.	0 < Integer < 100,000,000	Required
PID	Property identification number of a PSOLID or PLSOLID entry.	Integer > 0	Required
Gi	Identification numbers of connected grid points.	Integer ≥ 0 or blank	Required

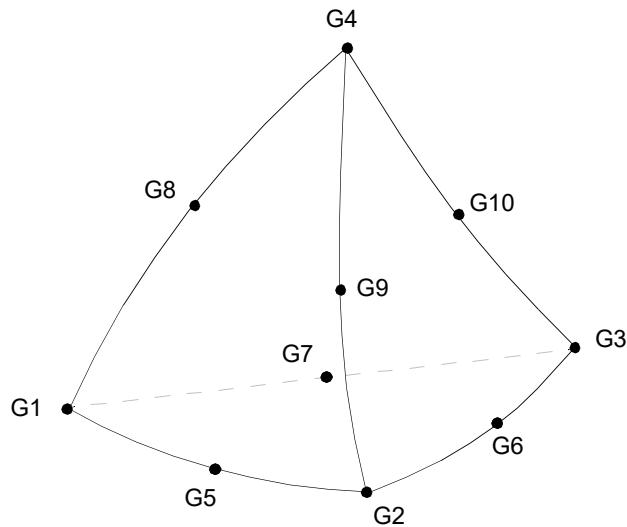


Figure 9-69 CTETRA Element Connection

Remarks:

1. Element ID numbers must be unique with respect to all other element ID numbers.
2. The topology of the diagram must be preserved, i.e., G1, G2, G3 define a triangular face; G1, G8, and G4 are on the same edge, etc.
3. The edge points, G5 to G10, are optional. For Nastran conventional elements, any or all of them may be deleted. If the ID of any edge connection point is left blank or set to zero, the equations of the element are adjusted to give correct results for the reduced number of connections. Corner grid points cannot be deleted. The element is an isoparametric element in all cases.
Note that for advanced nonlinear elements, partial specification of mid-points is not allowed. i.e. either none of the mid-points should be specified or all of the mid points should be specified.
4. Components of stress are output in the material coordinate system, except hyperelastic elements which outputs stress in the basic coordinate system.
5. For nonhyperelastic elements, the element coordinate system is derived from the three vectors R, S, and T, which join the midpoints of opposite edges.

R vector joins midpoints of edges G1-G2 and G3-G4.

S vector joins midpoints of edges G1-G3 and G2-G4.

T vector joins midpoints of edges G1-G4 and G2-G3.

The origin of the coordinate system is located at G1. The element coordinate system is chosen as close as possible to the R, S, and T vectors and points in the same general direction. (Mathematically speaking, the coordinate system is computed in such a way that, if the R, S, and T vectors are described in the element coordinate system, a 3×3 positive definite symmetric matrix would be produced.)

Solid elements have both a material and an element coordinate system. Both systems are defined for the initial geometry, and for geometric nonlinear analysis they will rotate with the element. The material coordinate system is used to input anisotropic material properties and for stress output. The material coordinate system is defined by the CORDM field of the PSOLID entry. The element coordinate system is used for element stiffness integration (reduced shear for example) and optionally to define the material coordinate system (only if PSOLID,CORDM=-1).

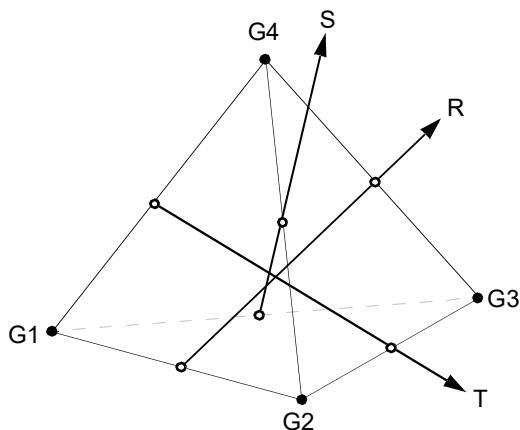


Figure 9-70 CTETRA Element R, S, and T Vectors

6. It is recommended that the edge points be located within the middle third of the edge.
7. For hyperelastic elements, the plot codes are specified under the CTETRAFD element name in [Item Codes, 1045](#).
8. If a CTETRA element is referenced by a PSET or PVAL entry, then p-version formulation is used and the element can have curved edges.
 - If a curved edge of a p-element is shared by an h-element without midside nodes, the geometry of the edge is ignored and set straight.
 - Elements with midside nodes cannot be p-elements and edges with midside nodes cannot be shared by p-elements.
9. By default, all of the six edges of the element are considered straight unless:
 - For p-elements, there is an FEEDGE or FEFACE entry that contains the two grids of any edge of this element. In this case, the geometry of the edge is used in the element.
 - For h-elements, any of G5 through G10 are specified.
10. The internal coordinate system of the element is used internally and is based on eigenvalue techniques to insure non bias in the element formulation. For stress/strain output this internal coordinate system (CORDM=-1 on PSOLID entry) is hard to visualize. Thus a CORDM=-2 on the PSOLID is available as shown in [Figure 9-71](#).

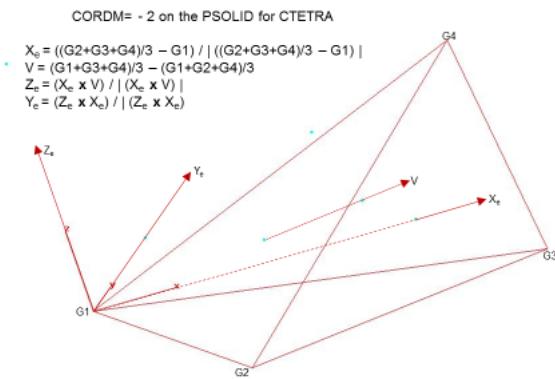


Figure 9-71 PSOLID on CTETRA

CTRIA3**Triangular Plate Element Connection**

Defines an isoparametric membrane-bending or plane strain triangular plate element.

Format:

1	2	3	4	5	6	7	8	9	10
CTRIA3	EID	PID	G1	G2	G3	THETA or MCID	ZOFFS		
		TFLAG	T1	T2	T3				

Example:

CTRIA3	111	203	31	74	75	3.0	0.98		
			1.77	2.04	2.09				

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PSHELL, PCOMP, PCOMPG or PLPLANE entry. (Integer > 0; Default = EID)
Gi	Grid point identification numbers of connection points. (Integers > 0, all unique)
THETA	Material property orientation angle in degrees. THETA is ignored for hyperelastic elements. (Real; Default = 0.0)
MCID	Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the T1-axis of the MCID coordinate system onto the surface of the shell element as follows: CORD1R, x -axis of MCID the coordinate is projected onto shell surface and the CORD2R material angle is measured from the G1-G2 line to the to the projected x-axis CORD1C, r-axis of MCID the coordinate is projected onto shell surface through CORD2C the element center and the material angle is measured from the G1-G2 CORD1S, line to the to the projected r-axis CORD2S Use DIAG 38 to print the computed THETA values. MCID is ignored for hyperelastic elements. For SOL 600, only CORD2R is allowed. (Integer ≥ 0 ; if blank, then THETA = 0.0 is assumed.)

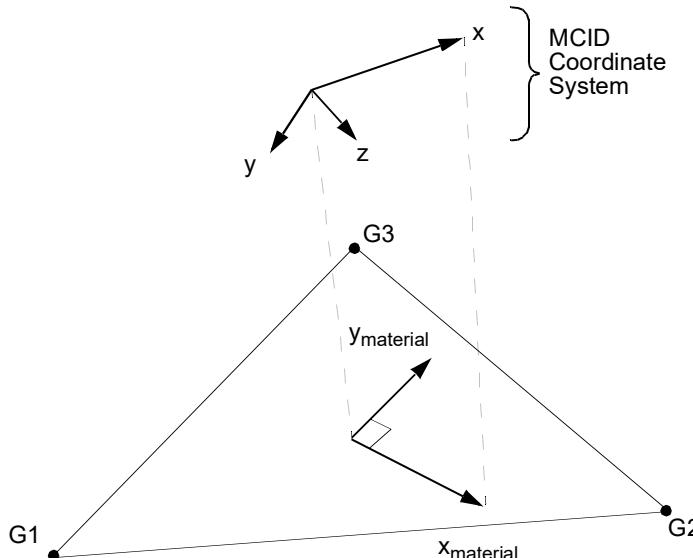


Figure 9-72 MCID Coordinate System Definition

ZOFFS	Offset from the surface of grid points to the element reference plane. See Remark 3. ZOFFS is ignored for hyperelastic elements. (Real)
TFLAG	An integer flag, signifying the meaning of the Ti values. (Integer 0, 1, or blank)
Ti	Membrane thickness of element at grid points G1 through G3. If "TFLAG" zero or blank, then Ti are actual user specified thickness. (Real ≥ 0.0 or blank, not all zero. See Remark 4. for default.) If "TFLAG" one, then the Ti are fraction relative to the T value of the PSHELL. (Real > 0.0 or blank; not all zero. Default = 1.0) Ti are ignored for hyperelastic elements.

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. The continuation is optional. If it is not supplied, then T1 through T3 will be set equal to the value of T on the PSHELL entry.
3. Elements may be offset from the connection points by means of the ZOFFS field. Other data, such as material matrices and stress fiber locations, are given relative to the reference plane. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive Z-axis of the element coordinate system. If the ZOFFS field is used, then both the MID1 and MID2 fields must be specified on the PSHELL entry referenced by PID.

Two methods are available for the computation of offsets: original and enhanced. The default method is the original method. The enhanced method is requested by the Bulk Data entry MDLPRM, OFFDEF, LROFF. For options of offsets, please refer to the Bulk Data entry MDLPRM, OFFDEF, option.

For solution sequences that require differential stiffness such as SOL103 (with preloading), SOL105, and SOL400, it is highly recommended to use MDLPRM, OFFDEF, LROFF. In SOL103, if the case control ADAMSMNF FLEXBODY=YES or AVLEXB EXBBODY=YES is present, then MDLPRM, OFFDEF, LROFF must be replaced by MDLPRM, OFFDEF, NOMASS.

For SOLs 106, 129, 153, and 159 the differential stiffness for offset vectors will give incorrect results with PARAM, LGDISP, 1. In addition in SOLs 106 and 129 offset vectors will produce incorrect results with thermal loading.

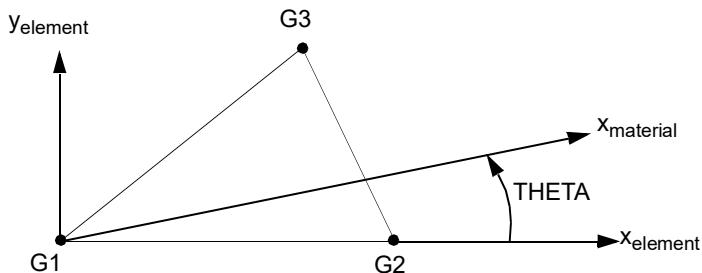


Figure 9-73 CTRIA3 Element Geometry and Coordinate Systems

4. The reference coordinate system for the output of stress, strain and element force depends on the element type.
 - For CTRIA3 elements, which are not p-elements and not hyperelastic, the reference coordinate system for output is the element coordinate system.
 - For CTRIA3 elements referenced by a PSET of PVAL entry, the stresses, strains and element forces are output in the local tangent plane of the element. The local tangents are oriented in a user defined direction which is uniform across a set of elements. By default, the local tangent x-direction is oriented in the positive x-direction of the basic coordinate system.
 - For hyperelastic elements the stress and strain are output according to CID on the PLPLANE entry.
5. For hyperelastic elements, the plot codes are specified under the CTRIAFD element name in [Item Codes, 1045](#).
6. SYSTEM(218), alias T3SKEW, allows the user to control the minimum vertex angle for TRIA3 elements at which USER WARNING MESSAGE 5491 is issued. The default value is 10. degrees.
7. If a CTRIA3 element is referenced by a PSET or PVAL entry, then a p-version formulation is used and the element can have curved edges.
 - If a curved edge of a p-element is shared by an h-element CTRIA3, the geometry of the edge is ignored and set straight.

8. By default, all of the three edges of the element are considered straight unless the element is a p-element and the edges are associated to curved geometry with FEEDGE or FEFACE Bulk Data entries.
9. For RC network solver in thermal analysis, the ZOFFS is ignored.
10. In SOL 600, when PCOMP is used, one must define the material property with orientation, MCID>0.

CTRIA6**Curved Triangular Shell Element Connection**

Defines a curved triangular shell element or plane strain with six grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CTRIA6	EID	PID	G1	G2	G3	G4	G5	G6	
	THETA or MCID	ZOFFS	T1	T2	T3	TFLAG			

Example:

CTRIA6	302	3	31	33	71	32	51	52	
	45	.03	.020	.025	.025				

Descriptor	Meaning
EID	Element Identification number. (0 < Integer < 100,000,000)
PID	Property identification number of PSHELL, PCOMP, PCOMPG or PLPLANE entry. (Integer > 0)
G1, G2, G3	Identification numbers of connected corner grid points. (Unique Integers > 0)
G4, G5, G6	Identification number of connected edge grid points. Optional data for any or all three points for Nastran conventional element only. (Integer > 0 or blank) Note that for advanced nonlinear elements, partial specification of mid-points is not allowed. i.e. either none of the mid-points should be specified or all of the mid points should be specified.
THETA	Material property orientation angle in degrees. THETA is ignored for hyperelastic elements. (Real; Default = 0.0)
MCID	Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the T1-axis of the MCID coordinate system onto the surface of the shell element as follows: CORD1R, x -axis of MCID the coordinate is projected onto shell surface and the CORD2R material angle is measured from the G1-G2 line to the to the projected x- axis CORD1C, r-axis of MCID the coordinate is projected onto shell surface through the CORD2C element center and the material angle is measured from the G1-G2 line CORD1S, to the to the projected r-axis CORD2S MCID is ignored for hyperelastic elements. For SOL 600, only CORD2R is allowed. (Integer \geq 0; if blank, then THETA = 0.0 is assumed)

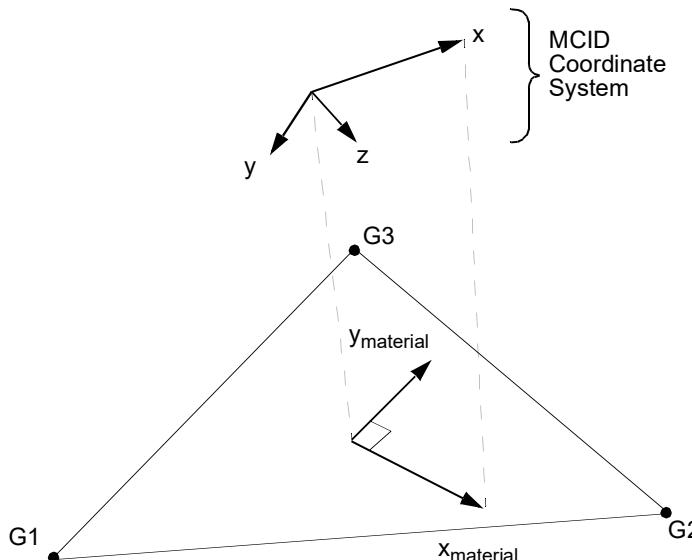


Figure 9-74 MCID Coordinate System Definition

ZOFFS	Offset from the surface of grid points to the element reference plane; see Remark 6. ZOFFS is ignored for hyperelastic elements. (Real)
Ti	Membrane thickness of element at grid points G1 through G4. If "TFLAG" zero or blank, then Ti are actual user specified thickness. (Real ≥ 0.0 or blank, not all zero. See Remark 4. for default.) If "TFLAG" one, then the Ti are fraction relative to the T value of the PSHELL. (Real > 0.0 or blank, not all zero. Default = 1.0) Ti are ignored for hyperelastic elements.
TFLAG	An integer flag, signifying the meaning of the Ti values. (Integer 0, 1, or blank)

Remarks:

1. Element identification numbers should be unique with respect to all other element IDs.
2. Grid points G1 through G6 must be numbered as shown in [Figure 9-75](#).
3. The orientation of the material property coordinate system is defined locally at each interior integration point by THETA, which is the angle between x_{material} and the line of constant η.
4. T1, T2, and T3 are optional. If they are not supplied and no TFLAG, then T1 through T3 will be set equal to the value of T on the PSHELL entry.
5. It is recommended that the midside grid points be located within the middle third of the edge.

6. Elements may be offset from the connection points by means of the ZOFFS field. Other data, such as material matrices and stress fiber locations, are given relative to the reference plane. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive z-axis of the element coordinate system. If the ZOFFS field is used, then both the MID1 and MID2 fields must be specified on the PSHELL entry referenced by PID.

Two methods are available for the computation of offsets: original and enhanced. The default method is the original method. The enhanced method is requested by the Bulk Data entry MDLPRM, OFFDEF, LROFF. For options of offsets, please refer to the Bulk Data entry MDLPRM, OFFDEF, option.

For solution sequences that require differential stiffness such as SOL103 (with preloading), SOL105, and SOL400, it is **highly recommended** to use MDLPRM, OFFDEF, LROFF. In SOL103, if the case control ADAMSMNF FLEXBODY=YES or AVLEXB EXBBODY=YES is present, then MDLPRM, OFFDEF, LROFF must be replaced by MDLPRM, OFFDEF, NOMASS.

7. If all midside grid points are deleted, then the element will be excessively stiff and the transverse shear forces will be incorrect. A User Warning Message is printed. A CTRIA3 element entry is recommended instead. If the element is hyperelastic, then the element is processed identically to the hyperelastic CTRIA3 element.
8. For a description of the element coordinate system, see [Shell Elements \(CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, CQUADR\)](#) (p. 123) in the *MSC Nastran Reference Guide*. Stresses and strains are output in the local coordinate system identified by x_l and y_l in [Figure 9-75](#). For hyperelastic elements, stresses and strains are output in the coordinate system defined by the CID field on the PLPLANE entry.

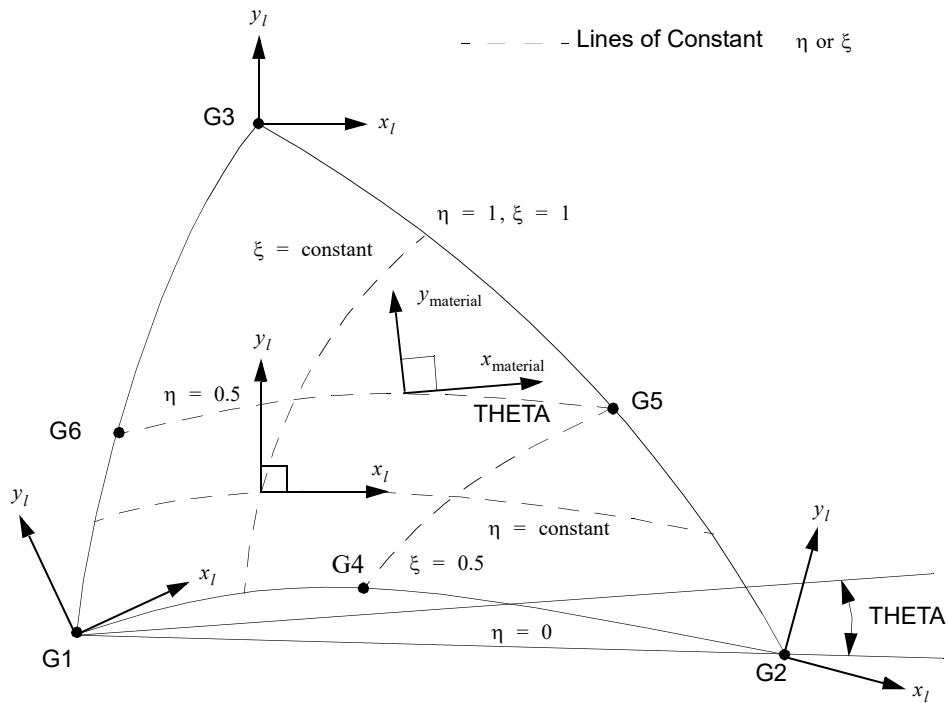


Figure 9-75 CTRIA6 Element Geometry and Coordinate Systems

9. For hyperelastic elements, the plot codes are specified under the CTRIAFD element name in [Item Codes, 1045](#).
10. For RC network solver in thermal analysis, the ZOFFS is ignored.
11. In SOL 600, when PCOMP is used, one must define the material property with orientation, MCID>0.

CTRIAR

Triangular Plate Element Connection

Defines an isoparametric membrane-bending triangular plate element. This element has a normal rotational (drilling) degrees-of-freedom. It is a companion to the CQUADR element.

Format:

1	2	3	4	5	6	7	8	9	10
CTRIAR	EID	PID	G1	G2	G3	THETA or MCID	ZOFFS		
		TFLAG	T1	T2	T3				

Example:

CTRIAR	111	203	31	74	75	3.0			
			1.77	2.04	2.09				

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PSHELL, PCOMP or PCOMPG entry. (Integer > 0; Default = EID)
G1, G2, G3	Grid point identification numbers of connection points. (Integers > 0; all unique)
THETA	Material property orientation angle in degrees. (Real; Default = 0.0)
MCID	Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the T1-axis of the MCID coordinate system onto the surface of the shell element as follows: CORD1 x -axis of MCID the coordinate is projected onto shell surface and the R, material angle is measured from the G1-G2 line to the to the projected x- CORD2 axis R CORD1 r-axis of MCID the coordinate is projected onto shell surface through the C, element center and the material angle is measured from the G1-G2 line to the to the projected r-axis C CORD1S , CORD2S
	Use DIAG 38 to print the computed THETA values. For SOL 600, only CORD2R is allowed. (Integer \geq 0; if blank, then THETA = 0.0 is assumed)
ZOFFS	Offset from the surface of grid points to the element reference plane. See Remark 5.

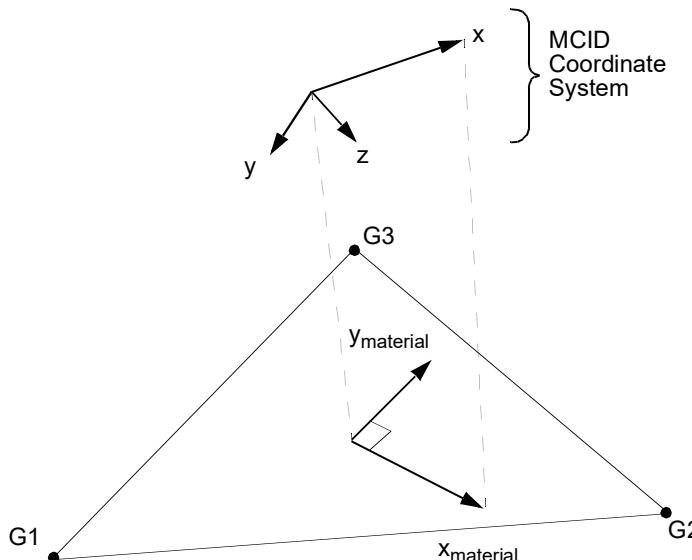


Figure 9-76 MCID Coordinate System Definition

TFLAG

An integer flag, signifying the meaning of the Ti values. (Integer 0, 1, or blank)

Ti

Membrane thickness of element at grid points G1 through G4. If “TFLAG” zero or blank, then Ti are actual user specified thickness. (Real ≥ 0.0 or blank, not all zero. See Remark 4. for default.) If “TFLAG” one, then the Ti are fraction relative to the T value of the PSHELL. (Real > 0.0 or blank, not all zero. Default = 1.0) Ti are ignored for hyperelastic elements.

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. The continuation is optional. If it is not supplied, then T1 through T3 will be set equal to the value of T on the PSHELL entry.
3. Stresses are output in the element coordinate system at the centroid and grid points G1 through G3.
4. Inaccurate results will be obtained if interior grids have the rotation normal (drilling) to the element constrained. At the boundary of a model, the drilling degrees-of-freedom must be constrained if the user wants a fixed boundary. Also, for this element it is critical that consistent membrane (in plane) edge loads be applied. Reference the [PLOAD4entry](#) (SORL option) and the [Consistent Surface and Edge Loads](#) (Ch. 3) in the *MSC Nastran Reference Guide* for additional information.

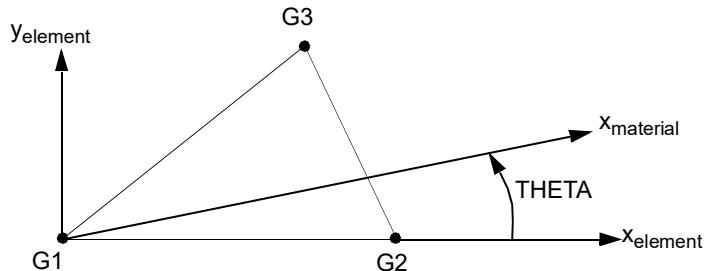


Figure 9-77 CTRIAR Element Geometry and Coordinate Systems

5. Elements may be offset from the connection points by means of the ZOFFS field. Other data, such as material matrices and stress fiber locations, are given relative to the reference plane. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive Z-axis of the element coordinate system. If the ZOFFS field is used, then both the MID1 and MID2 fields must be specified on the PSHELL entry referenced by PID.

Two methods are available for the computation of offsets: original and enhanced. The default method is the original method. The enhanced method is requested by the Bulk Data entry MDLPRM, OFFDEF, LROFF. For options of offsets, please refer to the Bulk Data entry MDLPRM, OFFDEF, option.

For solution sequences that require differential stiffness such as SOL103 (with preloading), SOL105, and SOL400, it is **highly recommended** to use MDLPRM, OFFDEF, LROFF. In SOL103, if the case control ADAMSMNF FLEXBODY=YES or AVLEXB EXBBODY=YES is present, then MDLPRM, OFFDEF, LROFF must be replaced by MDLPRM, OFFDEF, NOMASS.

For SOLs 106, 129, 153, and 159 the differential stiffness for offset vectors will give incorrect results with PARAM, LGDISP, 1. In addition in SOLs 106 and 129 offset vectors will produce incorrect results with thermal loading.

6. In SOL 600, when PCOMP is used, one must define the material property with orientation, MCID>0.

CTRIAX**Axisymmetric Triangular Element (Fully Nonlinear or Linear Harmonic)**

Defines an axisymmetric triangular element with up to six grid points for use in fully nonlinear (i.e., large strain and large rotations) hyperelastic analysis or a linear harmonic or rotordynamic analysis. The element has between three and six grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CTRIAX	EID	PID	G1	G2	G3	G4	G5	G6	
	THETA or MCID								

Example:

CTRIAX	111	203	31	74	75				
--------	-----	-----	----	----	----	--	--	--	--

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000) See Remark 1.
PID	Property identification number of a PLPLANE or PAXSYMH entry. (Integer > 0). See Remark 2.
G1, G2, G3	Identification numbers of connected corner grid points. Required data for all three grid points. (Unique Integers > 0). See Remark 3., 6., 7.
G4, G5, G6	Identification numbers of connected edge grid points. Optional data for any or all four grid points for Nastran conventional element only. (Integer > 0 or blank). See Remark 3., 4., 6., 7. Note that for advanced nonlinear elements, partial specification of mid-points is not allowed. i.e. either none of the mid-points should be specified or all of the mid points should be specified
THETA	Material property orientation angle in degrees. THETA is only applicable if the PLPLANE entry has an associated PSHLN2 entry which is honored only in SOL 400. For PSHLN2 BEHi=AXSOLID code, THETA is measured relative to the R axis of the element. (Real; Default = 0.0)
MCID	Material coordinate system identification number, MCID is only applicable if the PLPLANE entry has an associated PSHLN2 entry which is honored only in SOL 400. The x-axis of the material coordinate system is determined by projecting the x-axis of the MCID coordinate system onto the surface of the element. The resulting angle is measured relative to the R axis of the element. For PSHLN2 BEHi=AXSOLID code, THETA is measured relative to the R axis of the element. (Integer ≥ 0 ; if blank, then THETA = 0.0 is assumed.) THETA is ignored if PID refers to a PAXSYMH entry.

Remarks:

1. Element identification numbers must be unique with respect to all other element IDs of any kind.

2. If PID refers to a PLPLANE entry, CTRIAX defines an element for use in fully nonlinear analysis. If PID refers to a PAXSYMH entry, CTRIAX defines a linear harmonic element for use in rotordynamic or harmonic analysis.
3. Gi must be numbered as shown in [Figure 9-78](#).
4. It is recommended that the edge points be located within the middle third of the edge.
5. The plot codes are specified under the CTRIAXFD element name in [Item Codes, 1045](#).
6. The grid points of the axisymmetric element must lie on the x-y plane of the basic coordinate system. Stress and strain are output in the basic coordinate system.
7. A concentrated load (e.g., FORCE entry) at Gi is divided by the radius to Gi and then applied as a force per unit circumferential length. For example, in order to apply a load of 100 N/m on the circumference at G1, which is located at a radius of 0.5 m, then the magnitude specified on the static load entry must result in:

$$(100 \text{ N/m}) \cdot (0.5 \text{ m}) = 50\text{N}$$

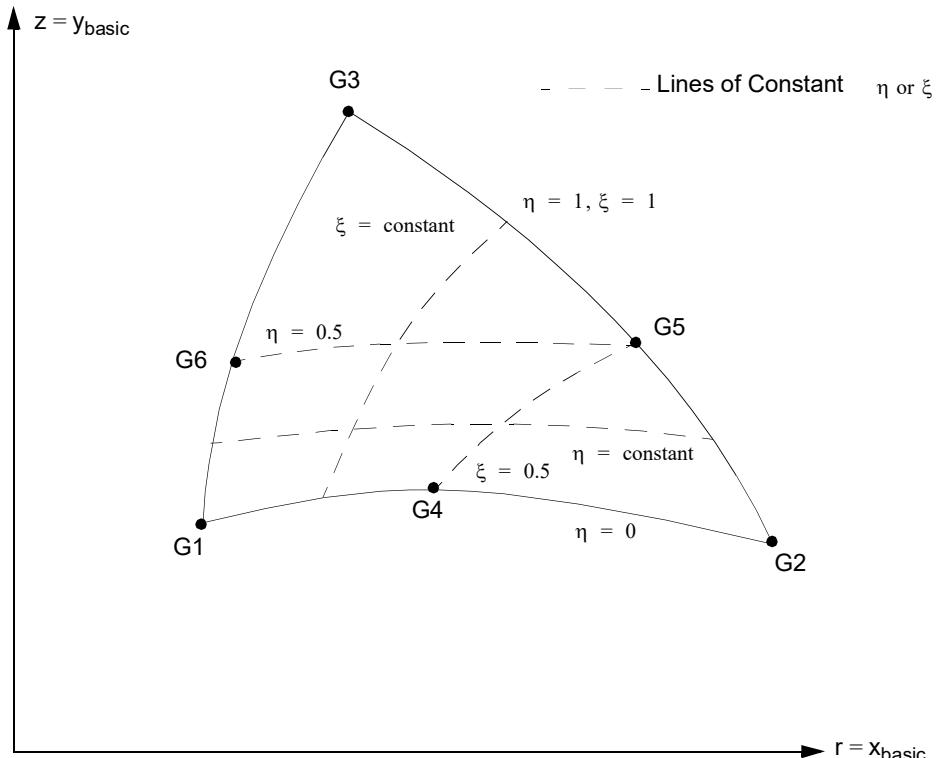


Figure 9-78 CTRIAX Element Coordinate System

CTRIAX6**Axisymmetric Triangular Element Connection**

Defines an isoparametric and axisymmetric triangular cross section ring element with midside grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CTRIAX6	EID	MID	G1	G2	G3	G4	G5	G6	
	TH								

Example:

CTRIAX6	22	999	10	11	12	21	22	32	
	9.0								

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
MID	Material identification number. (Integer > 0)
Gi	Grid point identification numbers of connected points (unique Integers > 0, or blank for deleted nodes.)
TH	Material property orientation angle in degrees. (Real; Default = 0.0)

Remarks:

1. The grid points must lie in the x-z plane of the basic coordinate system, with $x = r \geq 0$. The grid points must be listed consecutively beginning at a vertex and proceeding around the perimeter in either direction. Corner grid points G1, G3, and G5 must be present. Any or all edge grid points G2, G4, or G6 may be deleted for Nastran conventional element. Note that the alternate corner-edge grid point pattern is different from the convention used on the CTRIA6 element.
Note that for advanced nonlinear elements, partial specification of mid-points is not allowed. i.e. either none of the mid-points should be specified or all of the mid points should be specified.
2. For structural problems, the MID may refer to a MAT1 or MAT3 entry.
3. The continuation is optional.
4. Material properties (if defined on a MAT3 entry) and stresses are given in the (r_m, z_m) coordinate system shown in [Figure 9-80](#).
5. A concentrated load (e.g., FORCE entry) at Gi is multiplied by 2π times the radius to Gi and then applied as a force per unit circumferential length. (which is located at a radius of 0.5 m), the magnitude of the load specified on the static load entry must result in:

$$(100 \text{ N/m}) \cdot 2\pi \cdot (0.5 \text{ m}) = 314.159 \text{ N}$$

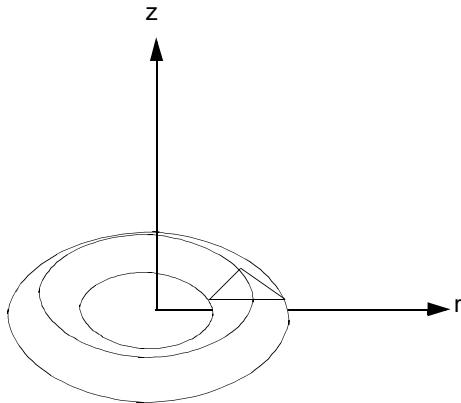


Figure 9-79 CTRIA6 Element Idealization

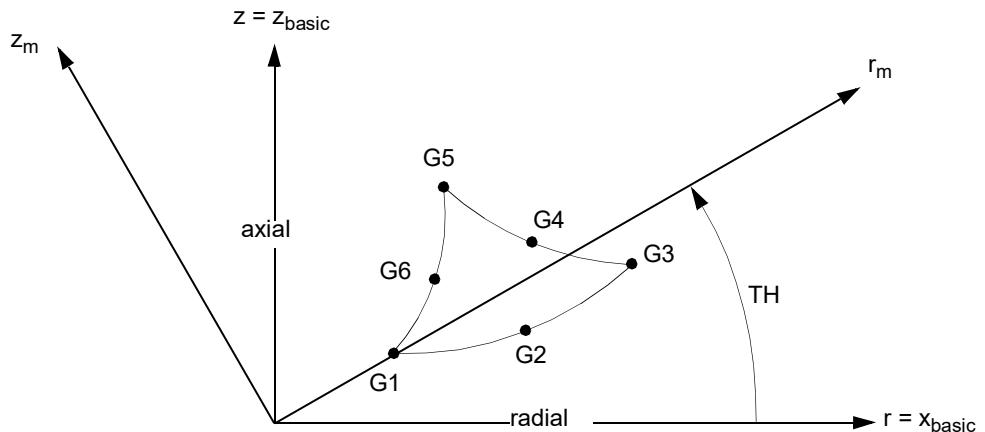


Figure 9-80 CTRIA6 Element Geometry and Coordinate Systems

6. For thermal problems, the MID may refer to a MAT4 or MAT5 entry. In order to model the convection or radiation along the edges the CHBDYG with TYPE=REV must be used.
7. To model axi-symmetric view factor computations using the CTRIA6 elements, the normal direction for the CTRIA6 element must point in the negative Y direction. The reason for this is to set up PATRAN for the correct nodal specification on the CHBDYG with REV option

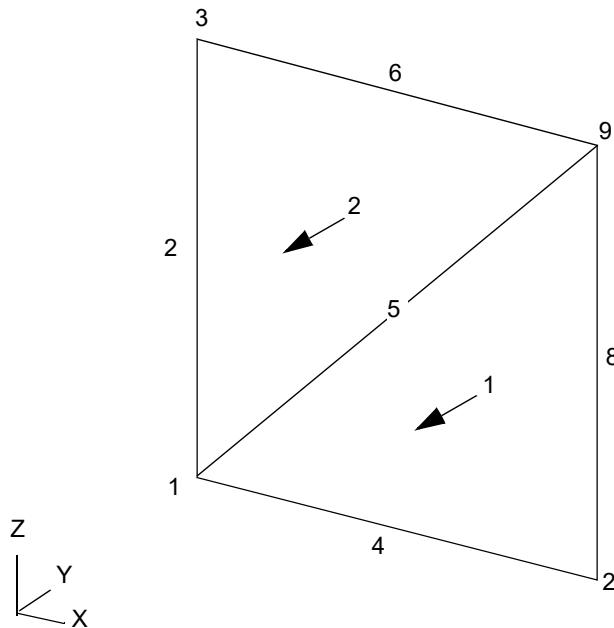


Figure 9-81 Normal Vectors That Point in a Negative Y Direction.

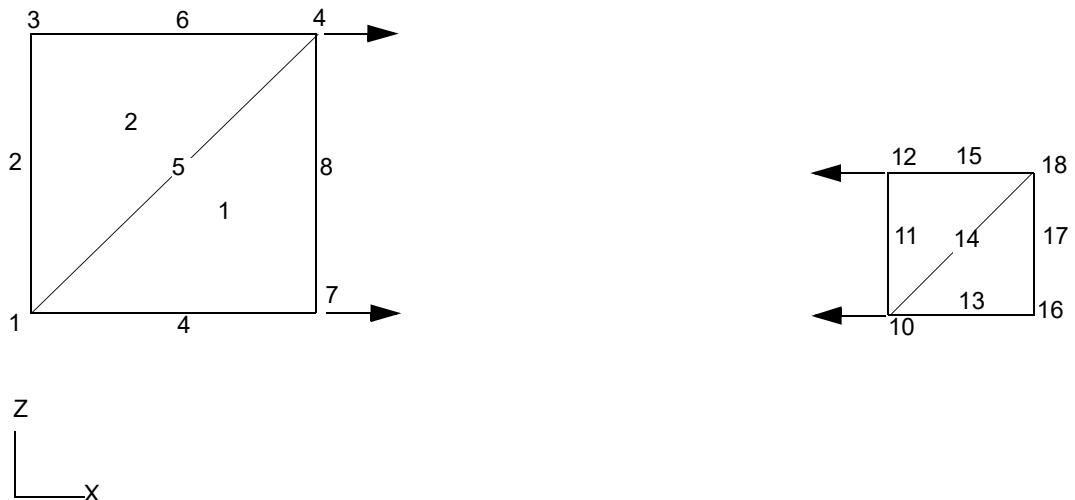


Figure 9-82 View Factor Boundary Conditions.

```
$ CHBDYG Surface Elements
CHBDYG 100001           REV      1
          7       9       8
CHBDYG 100002           REV      2
          12      10      11
```

Please note that on the CHBDYG, REV option, the nodal connectivity specification is in the clockwise fashion. In this case the CHBDYG with element ID of 100001 is defined by grids 7,9,8 which means the normal vector of the edge is pointing towards right. On the other hand the edges define by grid 12,10,11 means the normal vector of the edge is pointing towards left. If you have not reverse the element direction to a negative Y direction on the CTRIA6 element, the view factor computation will result a net view factor of zero.

CTUBE

Tube Element Connection

Defines a tension-compression-torsion tube element.

Format:

1	2	3	4	5	6	7	8	9	10
CTUBE	EID	PID	G1	G2					

Example:

CTUBE	12	13	21	23					
-------	----	----	----	----	--	--	--	--	--

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PTUBE entry. (Integer > 0; Default = EID)
G1, G2	Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Only one tube element may be defined on a single entry.

CVISC**Viscous Damper Connection**

Defines a viscous damper element.

Format:

1	2	3	4	5	6	7	8	9	10
CVISC	EID	PID	G1	G2					

Example:

CVISC	21	6327	29	31					
-------	----	------	----	----	--	--	--	--	--

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PVISC entry. (Integer > 0; Default = EID)
G1, G2	Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Only one viscous damper element may be defined on a single entry.
3. Grids G1 and G2 must not be coincident. If coincident grids are required, use either the CDAMP or CBUSH entry.

CWELD**Weld or Fastener Element Connection**

Defines a weld or fastener connecting two surface patches or points. Large displacement and large rotational effects are supported when using SOL 600 and SOL 400 only.

Format PARTPAT:

1	2	3	4	5	6	7	8	9	10
CWELD	EWID	PWID	GS	"PARTPAT"	GA	GB		MCID	
	PIDA	PIDB							
	XS	YS	ZS						

Example:

CWELD	101	8	203	PARTPAT					
	21	33							

Alternate formats and examples:

Format ELPAT:

CWELD	EWID	PWID	GS	"ELPAT"	GA	GB		MCID	
	SHIDA	SHIDB							
	XS	YS	ZS						

Example:

CWELD	103	5	403	ELPAT					
	309	511							

Format ELEMID:

CWELD	EWID	PWID	GS	"ELEMID"	GA	GB		MCID	
	SHIDA	SHIDB							

Example:

CWELD	103	5	403	ELEMID					
	309	511							

Format GRIDID:

CWELD	EWID	PWID	GS	"GRIDID"	GA	GB	SPTYP	MCID	
	GA1	GA2	GA3	GA4	GA5	GA6	GA7	GA8	
	GB1	GB2	GB3	GB4	GB5	GB6	GB7	GB8	

Example:

CWELD	7	29	233	GRIDID			QT		
	15	28	31	35	46	51	55	60	
	3	5	8						

Format ALIGN:

CWELD	EWID	PWID		"ALIGN"	GA	GB		MCID	
-------	------	------	--	---------	----	----	--	------	--

Example:

CWELD	7	29		ALIGN	103	259			
-------	---	----	--	-------	-----	-----	--	--	--

Descriptor	Meaning	Type	Default
EWID	CWELD element identification number. See Remark 1.	$0 < \text{Integer} < 100,000,000$	Required
PWID	Property identification number of a PWELD entry.	Integer > 0	Required
GS	Identification number of a grid point which defines the location of the connector. See Remarks 2. and 3.	Integer > 0 or blank	
"PARTPAT"	Character string indicating the type of connection. The format of the subsequent entries depends on the type. "PARTPAT", for example, indicates that the connectivity of surface patch A to surface patch B is defined with two property identification numbers of PSHELL entries, PIDA and PIDB, respectively. The "PARTPAT" format connects up to 3x3 elements per patch. See Remark 4.	Character	Required
GA, GB	Grid point identification numbers of piercing points on surface A and surface B, respectively. See Remark 5.	Integer > 0 or blank	
MCID	Specifies the element stiffness coordinate system. See Remark 16.	Integer ≥ -1 or blank	Default = -1
PIDA, PIDB	Property identification numbers of PSHELL entries defining surface A and B respectively.	Integer > 0	Required for "PARTPAT"
XS, YS, ZS	Coordinates of spot weld location in basic. See Remark 2.	Real	Required if GS and GA are not defined.

For the alternate formats, the descriptor meaning are described below:

Descriptor	Meaning	Type	Default
"ELPAT"	Character string indicating that the connectivity of surface patch A to surface patch B is defined with two shell element identification numbers, SHIDA and SHIDB, respectively. The "ELPAT" format connects up to 3x3 elements per patch. See Remark 6.	Character	Required
SHIDA, SHIDB	Shell element identification numbers of elements on patch A and B, respectively.	Integer > 0	Required for "ELPAT"
"ELEMID"	Character string indicating that the connectivity of surface patch A to surface patch B is defined with two shell element identification numbers, SHIDA and SHIDB, respectively. The "ELEMID" format connects one shell element per patch. See Remark 7.	Character	Required
SHIDA, SHIDB	Shell element identification numbers of elements on patch A and B, respectively.	Integer > 0	Required for "ELEMID"
"GRIDID"	Character string indicating that the connectivity of surface patch A to surface patch B is defined with two sequences of grid point identification numbers, GAi and GBi, respectively. The "GRIDID" format connects the surface of any element. See Remark 8.	Character	Required
SPTYP	Character string indicating types of surface patches A and B. SPTYP = "QQ", "TT", "QT", "TQ", "Q" or "T". See Remark 9.	Character	Required for "GRIDID"
GAi	Grid identification numbers of surface patch A. GA1 to GA3 are required. See Remark 10.	Integer > 0	Required for "GRIDID"
GBi	Grid identification numbers of surface patch B. See Remark 10.	Integer > 0	
"ALIGN"	Character string indicating that the connectivity of surface A to surface B is defined with two shell vertex grid points GA and GB, respectively. See Remark 11.	Character	Required
GA, GB	Vertex grid identification number of shell A and B, respectively.	Integer > 0	Required for "ALIGN"

Remarks:

1. CWELD defines a flexible connection between two surface patches, between a point and a surface patch, or between two shell vertex grid points. See [Figure 9-83](#) through [Figure 9-87](#).

2. Grid point GS defines the approximate location of the connector in space. GS is projected on surface patch A and on surface patch B. The resulting piercing points GA and GB define the axis of the connector. GS must have a normal projection on surface patch A and B. GS does not have to lie on the surface patches. GS is ignored for format "ALIGN". GA is used instead of GS if GS is not specified. For the formats "ELPAT" and "PARTPAT," if GS and GA are not specified, then XS, YS, and ZS will be assumed.
3. The connectivity between grid points on surface patch A and grid points on surface patch B is generated depending on the location of GS and the cross sectional area of the connector. Diagnostic print outs, checkout runs and non default settings of search and projection parameters are requested on the SWLDPRM Bulk Data entry. It is recommended to start with the default settings.
4. The format "PARTPAT" defines a connection of two shell element patches A and B with PSHELL property identification numbers PIDA and PIDB, respectively. The two property identification numbers must be different, see [Figure 9-83](#). The number of connected elements depends on the location of the piercing points GA, GB and the size of the diameter D. The diameter D is defined on the PWELD property entry. For this option, shell element patches A and B are allowed to share a common grid.

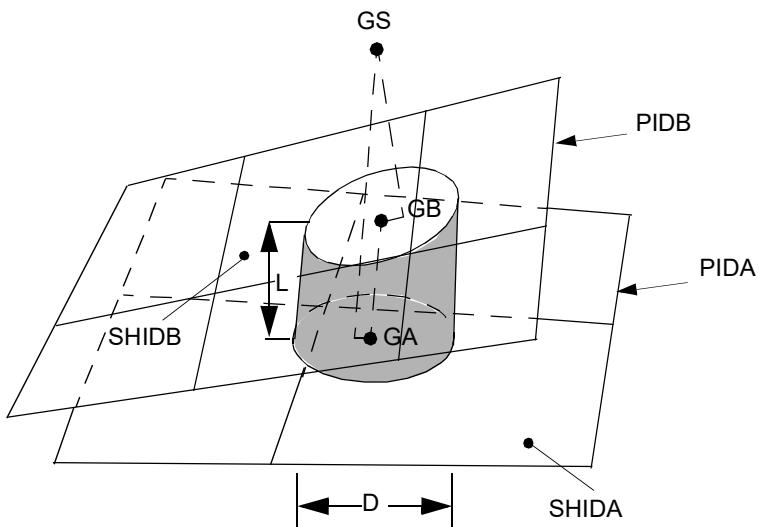


Figure 9-83 Patch to Patch Connection Defined with the Formats PARTPAT or ELPAT

5. The definition of the piercing grid points GA and GB is optional for all formats with the exception of the format "ALIGN". If GA and GB are given, GS is ignored. If GA and GB are not specified, they are generated from the normal projection of GS on surface patches A and B and internal identification numbers are generated for GA and GB starting with 101e+6 by default. The offset number can be changed with PARAM,OSWPPT. If GA and GB are specified, they must lie on or at least have a projection on surface patches A and B, respectively. By default, the locations of user specified GA and

GB will not be changed. If the user specifies "SWLDPRM, MOVGAB, 1," then the locations will be corrected so that they lie on surface patches A and B within machine precision accuracy. The length of the connector is the distance of grid point GA to GB, subject to being adjusted to the effective length defined in PWELD entry.

6. The format "ELPAT" defines a connection of two shell element patches A and B with shell element identification numbers SHIDA and SHIDB, see [Figure 9-83](#). The connectivity is similar to the format "PARTPAT". The number of connected elements depends on the location of the piercing points GA, GB and the size of the diameter D. For this option, shell element patches A and B are allowed to share a common grid.
7. The format "ELEMID" defines a connection of two shell element patches A and B with shell element identification numbers SHIDA and SHIDB, see [Figure 9-84](#). The connectivity is restricted to a single element per patch regardless of the location of GA, GB and regardless of the size of the diameter of the connector. In addition, the format "ELEMID" can define a point to patch connection if SHIDB is left blank, see [Figure 9-85](#). Then grid GS is connected to shell SHIDA.

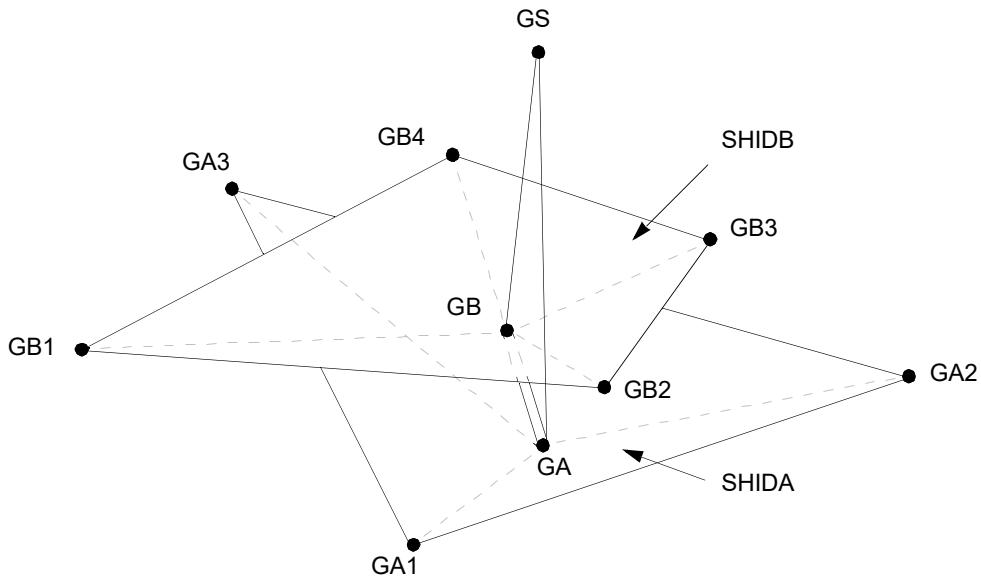


Figure 9-84 Patch to Patch Connection Defined with Format ELEMID or GRIDID

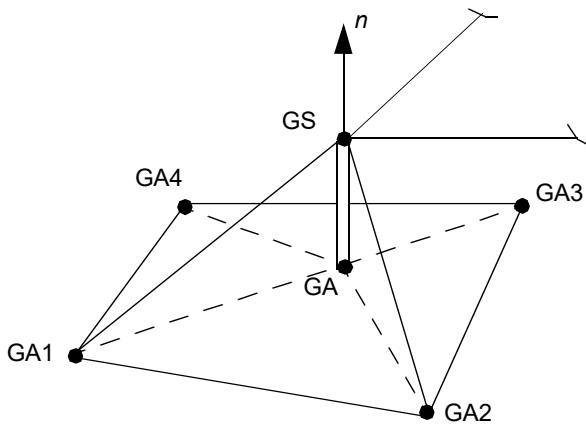
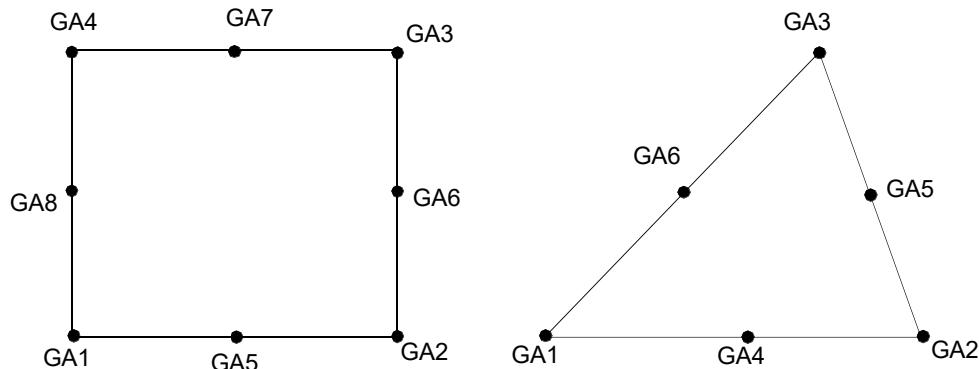


Figure 9-85 Point to Patch Connection Defined with Format ELEMID or GRID.

8. The format “GRIDID” defines a connection of two surface patches A and B with a sequence of grid points GA_i and GB_i , see [Figure 9-84](#). In addition, the format “GRIDID” can define a point to patch connection if all GB_i fields are left blank, see [Figure 9-85](#). Then grid GS is connected to grids GA_i . The grids GA_i and GB_i do not have to belong to shell elements.
9. SPTYP defines the type of surface patches to be connected. SPTYP is required for the format “GRIDID” to identify quadrilateral or triangular patches. The combinations are:

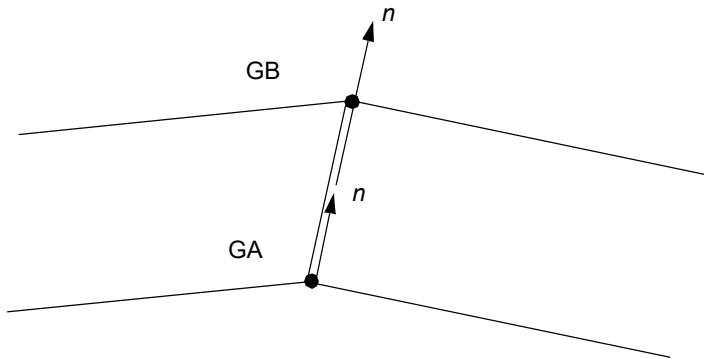
SPTYP	Description
QQ	Connects a quadrilateral surface patch A (Q4 to Q8) with a quadrilateral surface patch B (Q4 to Q8).
QT	Connects a quadrilateral surface patch A (Q4 to Q8) with a triangular surface patch B (T3 to T6).
TT	Connects a triangular surface patch A (T3 to T6) with a triangular surface patch B (T3 to T6).
TQ	Connects a triangular surface patch A (T3 to T6) with a quadrilateral surface patch B (Q4 to Q8).
Q	Connects the shell vertex grid GS with a quadrilateral surface patch A (Q4 to Q8) if surface patch B is not specified.
T	Connects the shell vertex grid GS with a triangular surface patch A (T3 to T6) if surface patch B is not specified.

10. GAi are required for the format “GRIDID”. At least 3 and at most 8 grid IDs may be specified for GAi and GBi, respectively. The rules of the triangular and quadrilateral elements apply for the order of GAi and GBi, see [Figure 9-86](#). Missing midside nodes are allowed.



[Figure 9-86](#) Quadrilateral and Triangular Surface Patches defined with Format GRIDID

11. The format “ALIGN” defines a point to point connection, see [Figure 9-87](#). GA and GB are required, they must be existing vertex nodes of shell elements. For the other formats, GA and GB are not required. Two shell normals in the direction GA-GB are generated at GA and GB, respectively.



[Figure 9-87](#) Point to Point Connection Defined with Format ALIGN

12. Forces and moments are output in the element coordinate system, see [Figure 9-88](#). The element coordinate system is constructed using the following rules:

The element x-axis points from GA to GB.

$$e_1 = \frac{x_B - x_A}{\|x_B - x_A\|} \quad \text{element x-axis}$$

In case of zero length, the normal of shell A is taken. All vector components are in basic if not noted otherwise.

Find the smallest component j of e_1

$$ce_1^i = |e_1^i|$$

$$ce_1^j = \min_{i=1,2,3} \{ce_1^i\}$$

Note that ce_1^i will be set to 10^{-6} if $ce_1^i < 10^{-6}$.

In case of two equal components we take the one with the smaller i . The corresponding basic vector

$$b_j, \text{ e.g., for } j=3, b_3 = \begin{Bmatrix} 0 \\ 0 \\ 1 \end{Bmatrix}$$

provides a good directional choice for e_2 . In addition, the vector e_2 must be orthogonal to e_1 .

$$\tilde{e}_2 = b_j - \frac{e_1^T b_j}{e_1^T e_1} e_1 \quad e_2 = \frac{\tilde{e}_2}{\|\tilde{e}_2\|} \text{ element y-axis}$$

and e_3 is just the cross product

$$e_3 = e_1 \times e_2 \quad \text{element z-axis}$$

The final transformation matrix is

$$T_{be} = \left[\begin{array}{c|c|c} & e_1 & e_2 & e_3 \\ \hline e_1 & & & \end{array} \right]$$

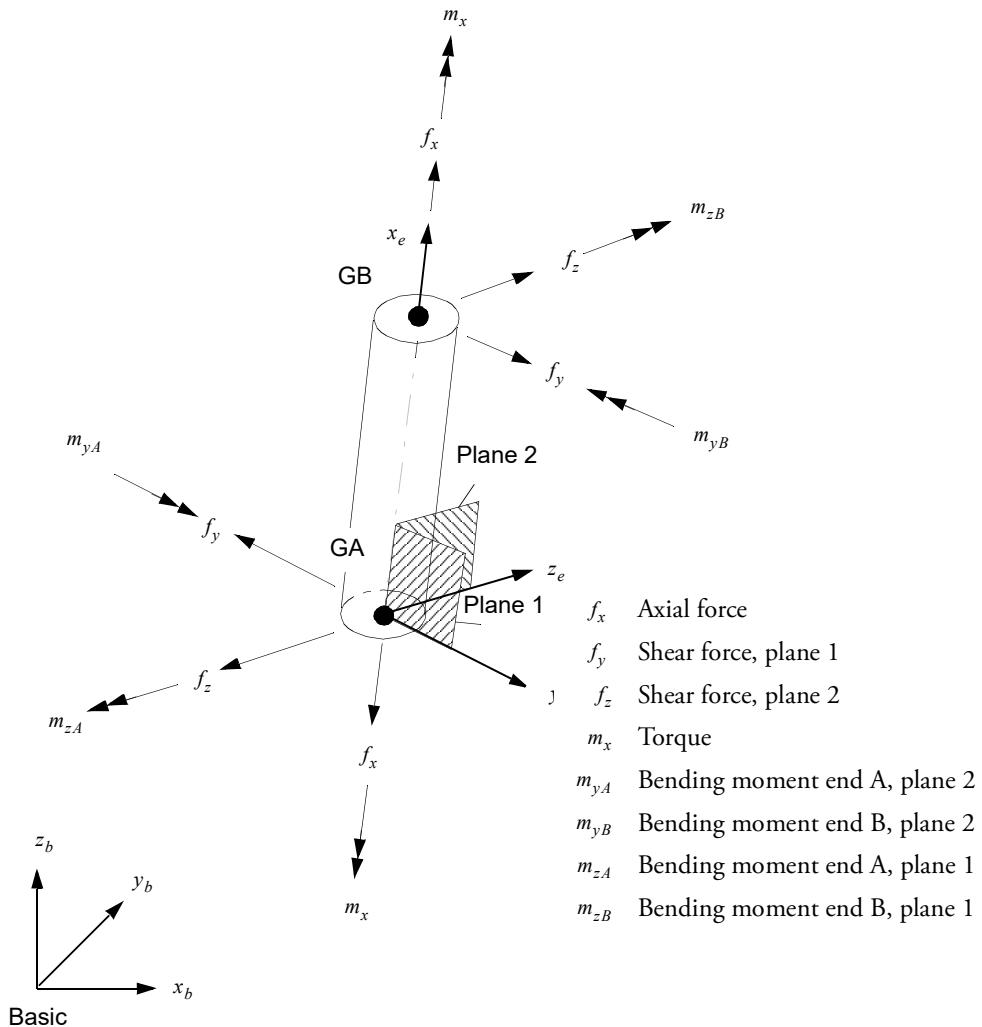


Figure 9-88 Element Coordinate System and Sign Convention of Element Forces

13. If "PARAM, OLDWELD, YES", in a SOL 400 analysis the CWELD element using method ELEMID, GRIDID, ELPAT or PARTPAT internally gets decomposed in a CBEAM element and a number of RBE3 elements. For method ALIGN the CWELD element internally gets replaced by a CBEAM element. The CBEAM element obtains a circular cross section with its diameter and material properties taken from the PWELD input and its length is determined by the final locations of the GA and GB grids of the CWELD. The RBE3 elements connect the GA and GB grids to the plate structures. For methods ELEMID and GRIDID one RBE3 element is used on each side to establish the connection. For methods ELPAT and PARTPAT five RBE3 elements are used on each side to establish the connection.

14. The output format of the forces and moments including the sign convention is identical to the CBAR element, see [Element Force Item Codes, 1085](#).
15. This entry is not supported in SOL 700.
16. MCID = -1 or blank (Default), then the coordinate system is as defined in Remark [12](#).
If MCID ≥ 0 , then a “beam” like coordinate system is defined. The x_{elem} axis direction of the connector defined as

$$\hat{e}_1 = \frac{\hat{x}_B - \hat{x}_A}{\|\hat{x}_B - \hat{x}_A\|}$$

The T2 direction defined by MCID will be used to define the orientation vector \vec{v} of the connector. Then the z_{elem} element axis will be defined as

$$\hat{e}_3 = \frac{\hat{e}_1 \times \vec{v}}{\|\hat{e}_1 \times \vec{v}\|}$$

The element y_{elem} axis is defined as

$$\hat{e}_2 = \hat{e}_3 \times \hat{e}_1$$

17. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 9 for that entry.
18. In SOL400, the behavior of this element in regard to large rotation is affected by the Case Control Command Rigid.
19. If partitioned superelements are present, then CWELD is supported in the main Bulk Data section only.

CYAX

Grid Points on Axis of Symmetry

Lists grid points that lie on the axis of symmetry in cyclic symmetry analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CYAX	G1	G2	G3	G4	G5	G6	G7	G8	
	G9	G10	-etc.-						

Example:

CYAX	27	152	THRU	160	192	11			
------	----	-----	------	-----	-----	----	--	--	--

Descriptor	Meaning
Gi	A list of grid points on the axis of symmetry. (Integer > 0 or Character “THRU”)

Remarks:

1. The displacement coordinate system (see CD field on GRID entry) for a grid point lying on the axis of symmetry must be a rectangular system with the z-component of motion aligned with the axis of symmetry. The positive axis of symmetry is defined so that the azimuthal angle from positive side 1 to side 2 of a segment is in the same direction as the angle from T1 to T2 for the axis point. This is consistent with left- or right-hand rule.
2. If the dihedral symmetry option (STYPE = “DIH” on the CYSYM entry) is selected, the y-axis must be perpendicular to side 1.
3. Grid points lying on the axis of symmetry may be constrained by SPCs but not by MPCs. If the number of segments is three or more, SPCs must be applied to both components 1 and 2 or to neither, and SPCs must be applied to both components 4 and 5 or to neither in order to satisfy symmetry. In addition, the degrees-of-freedom (not constrained by SPCs, if any) at these grid points must be in the analysis set (a-set). If all degrees-of-freedom of grid points on the axis of symmetry are constrained by SPCs (including heat transfer, where there is only one degree-of-freedom), the grid point should not be listed on the CYAX entry.
4. Grid points lying on the axis of symmetry must not be defined on side 1 or on side 2 by means of a CYJOIN entry.
5. The word “THRU” must not appear in fields 2 or 9.

CYJOIN**Cyclic Symmetry Boundary Points**

Defines the boundary points of a segment in cyclic symmetry problems.

Format:

1	2	3	4	5	6	7	8	9	10
CYJOIN	SIDE	C	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	-etc.-					

Example:

CYJOIN	1	T2	7	9	16	THRU	33	64	
	72	THRU	89						

Descriptor	Meaning
SIDE	Side identification. (Integer 1 or 2)
C	Type of coordinate system used on boundaries of dihedral or axisymmetry problems. See Remark 3. (Character: "T1", "T2", "T3", "R", "C", "S")
Gi	Grid or scalar point identification numbers. (Integer > 0 or Character "THRU")

Remarks:

1. CYJOIN entries are used only for cyclic symmetry problems. The CYSYM entry must be used to specify rotational, dihedral, or axisymmetry.
2. For rotational or axisymmetry problems, there must be one logical entry for SIDE = 1 and one for SIDE = 2. The two lists specify grid points to be connected; therefore, both lists must have the same length.
3. For dihedral problems, side 1 refers to the boundary between segments and side 2 refers to the middle of a segment. For dihedral and/or AXI type of symmetry, the grid point degree-of-freedom that is normal to the boundary must be specified in field 3 as "T1", "T2", or "T3" ("R", rectangular, and "C", cylindrical, are the same as "T2" while "S", spherical, is the same as "T3"). For scalar and extra points with one degree-of-freedom, these should be specified as blank, "T2", or "T3" if they are to have the same sign, and "T1", if the two connected points are to be opposite in sign.
4. All components of displacement at boundary points are connected to adjacent segments except those constrained by SPCi, MPC, or OMITi entries.
5. The points on the axis of symmetry of the model, defined in the CYAX entry must not be defined as a side 1 or side 2 point by means of this entry.
6. The word "THRU" may not appear in fields 4 or 9 of the parent entry and fields 2 or 9 on the continuation entries.
7. All grid points that are implicitly or explicitly referenced must be defined.

8. For rotational and axisymmetry problems, the displacement coordinate systems must be consistent between sides 1 and 2. This is best satisfied by the use of a spherical or cylindrical coordinate system.

CYLINDR

Defines the Shape of a Cylinder

Cylindrical shape used in the initial condition definition on the TICEUL1 entry. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
CYLINDR	VID		XC1	YC1	ZC1	XC2	YC2	ZC2	
	RAD								

Example:

CYLINDR	4		0.	0.	0.	1.	1.	1.	
	.5								

Descriptor	Meaning
VID	Unique cylinder number. (Integer > 0; Required)
XC1, YC1, ZC1	Coordinates of point 1, See Remark 1. (Real; Required)
XC2, YC2, ZC2	Coordinates of point 2. See Remark 1. (Real; Required)
RAD	Radius of the cylinder. (Real; Required)

Remarks:

1. A cylinder is defined by the two end points of the cylinder axis and a radius.
2. Initial conditions are defined for the elements that are fully or partially inside the cylinder.
3. See also the TICEUL1 Bulk Data entry.

CYSUP

Fictitious Supports for Cyclic Symmetry

Defines fictitious supports for cyclic symmetry analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CYSUP	GID	C							

Example:

CYSUP	16	1245							
-------	----	------	--	--	--	--	--	--	--

Descriptor	Meaning
GID	Grid point identification number. (Integer > 0)
C	Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks.)

Remarks:

- Components of motion defined on this entry may not appear on SPC, SPC1, OMIT, OMIT1 entries, or in rigid elements or multipoint constraints as dependent degrees-of-freedom.
- Supports are applied at the grid point identified in field 2 to prevent rigid body motions in static analysis, or to define rigid body modes in eigenvalue analysis. All degrees-of-freedom should be at a single grid point. In other words, there can only be one such supported grid point in the model. The supports are applied only to the cyclic components of order k=0 or k=1. In order to satisfy conditions of symmetry, certain restrictions are placed on the location of the grid point and the orientation of its displacement coordinate system, as shown in the following table:

Symmetry Option (STYPE on the CYSYM entry)	ROT	ROT	DIH	DIH	DIH
Number of Segments, N	2	≥ 3	1	2	≥ 3
Location of Grid Point	See Note c.	See Note d.	Side 1	Side 1	Side 1
Special Restrictions on Displacement Coordinate System	See Notes a. and e.	See Note b.	None	See Note a.	See Note b.

Notes:

- T3 axis must be parallel to axis of symmetry.
- Displacement coordinate system at the referenced grid point must be cylindrical with z-axis along the axis of symmetry.

- c. Any location except on side 2.
 - d. Any location except on the axis of symmetry or on side 2.
 - e. If the grid point is on the axis of symmetry, the displacement coordinate system must be rectangular.
3. If the number of segments, N, is 1 (in the case of DIH symmetry) or 2 (in the case of ROT or AXI symmetry), it is important that the rotational components referenced in field 3 be elastically connected to the structure. If $N \geq 2$ (in the case of DIH symmetry) or $N \geq 3$ (in the case of ROT or AXI symmetry), it is not important, because in this case the supports for rigid body rotation are actually applied to translational motions.
 4. If $N \geq 3$, supports will be applied to both the 1 and 2 (inplane-translational) components, if either is referenced, and to both the 4 and 5 (out-of-plane rotational) components, if either is referenced. If component 6 is supported, component 2 should not appear on OMIT or OMIT1 entries.
 5. The restrictions noted in Remarks 2. and 4. are related to symmetry requirements. For $N \geq 3$, symmetry requires that the supports be symmetrical (or antisymmetrical), with respect to any plane passing through the axis of symmetry. For the DIH options, $N = 1$ and $N = 2$, symmetry requires that the supports be symmetrical (or antisymmetrical) with respect to the plane(s) of symmetry. For the ROT option, $N = 2$, symmetry requires that a support be either parallel or perpendicular to the axis of symmetry.
 6. GID must be a grid point and not a scalar point.

CYSYM

Cyclic Symmetry Parameters

Defines parameters for cyclic symmetry analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CYSYM	NSEG	STYPE							

Example:

CYSYM	6	ROT							
-------	---	-----	--	--	--	--	--	--	--

Descriptor	Meaning
NSEG	Number of segments. (Integer > 0)
STYPE	Symmetry type. (Character: "ROT", "DIH", or "AXI")

Remarks:

1. STYPE = "AXI" is a special case of STYPE = "ROT" used to model axisymmetric structures.
2. If STYPE = "AXI", then all grid points must lie on side 1, side 2, or the axis. Also, plate elements with midside grid points may not be defined.

+Entries D - E

DAMPING

Parameter and Hybrid Damping Specification

Specifies the values for parameter damping and/or selects optional HYBRID damping.

Format:

1	2	3	4	5	6	7	8	9	10
DAMPING	ID	G	ALPHA1	ALPHA2	HYBRID	GEFACT	ROTSEP		
	W3	W4	WH						

Example:

DAMPING	100	.02					NO		
---------	-----	-----	--	--	--	--	----	--	--

Descriptor	Meaning
ID	Damping entry identification number. (Integer > 0; no Default)
G	Structural damping coefficient, see Remark 1. (Real; Default = 0.0)
ALPHA1	Scale factor for mass portion of Rayleigh damping, see Remark 4. (Real; Default = 0.0)
ALPHA2	Scale factor for stiffness portion of Rayleigh damping, see Remark 4. (Real; Default = 0.0)
HYBRID	Identification number of HYBDAMP entry for hybrid damping, see Remarks 5. and 6. (Integer \geq 0; Default = 0)
GEFACT	Scale factor for material damping. See Remark 1. and Remark 3. (Real; Default = 1.0)
ROTSEP	Remove rotor stiffness, mass, and structural damping from the hybrid damping calculation (Character: YES or NO; Default=YES)
W3	Average frequency for calculation of structural damping in transient response, see Remark 7. (Real \geq 0.0; Default = 0.0)
W4	Average frequency for calculation of material damping in transient response, see Remark 7. (Real \geq 0.0; Default = 0.0)
WH	Average frequency for calculation of hybrid ‘structural’ damping in transient response, see Remark 7. (Real \geq 0.0; Default = 0.0)

Remarks:

1. The DAMPING entry is referenced by the RSDAMP or SEDAMP Case Control commands. If a DAMPING entry is selected in the Case Control, the DAMPING values, including defaults, will override parameter inputs.
2. All damping selections are cumulative.
3. Structural damping specified by the G and GEFACT entries will replace any structural damping by:

$$\text{new}[K4] = G[K] + \text{GEFACT}[K4]$$

The GFACT value used is $\text{GEFACT} = \text{GE} (\text{default}=0.0) * \text{GEFACT} (\text{default}=1.0)$ where GE is the structural damping value on the material entry.

4. Rayleigh viscous damping is calculated as:

$$[B]_{\text{Rayleigh}} = \alpha_1[M] + \alpha_2[K]$$

5. Hybrid damping on the residual structure is only active for direct solution sequences. For modal solution sequences, the SDAMP Case Control request should be used.
6. Hybrid damping for superelements uses modes that are calculated using the superelement mass and stiffness matrices before upstream superelements are added and before SPC or MPC constraints are imposed. These matrices are known as the JJ-type matrices.
7. The W3 and W4 values are used in transient response only. A zero value (default) will result in no damping. The equivalent viscous damping is calculated as:

$$[B]_{\text{equiv}} = \left(\frac{G}{W3}\right)[K] + \left(\frac{\text{GEFACT}}{W4}\right)[K4] + \left(\frac{1}{WH}\right)[KH]$$

8. For rotordynamic analyses (RGYRO in the Case Control Section), the DAMPING calculations for the residual structure (RSDAMP) are performed without any rotor contributions (support structure only).
9. Rayleigh damping is designed to be applied only at Superelement, Part Superelement, or External Superelement residual assembly time. It is not applied in individual Parts or External superelements.
10. If Modules are present then this entry may only be specified in the main Bulk Data section.

Examples:

1. Specify a structural damping coefficient of 0.03 for the residual structure for frequency response.

Case Control:

RSDAMP=100

Bulk Data:

DAMPING, 100, 0.03

2. Specify a structural damping coefficient of 0.03 for the residual structure for transient response (use a “natural” frequency of 100 Hz)

Case Control:

RSDAMP=100

Bulk Data:

```
DAMPING, 100, 0.03,  
, 628.3
```

3. Specify hybrid damping for superelement 1. Use modal damping of two percent critical for the first 6 modes.

Case Control:

```
SUBCASE 1  
SUPER=1  
SEDAMP=100
```

Bulk Data:

```
DAMPING, 100, , , , 101  
HYBDAMP, 101, 102, 1001  
EIGRL, 102, , , 6  
TABDMP1, 1001, CRIT,  
, 0.0, 0.02, 1000.0, 0.02, ENDT
```

DAREA**Load Scale Factor**

Defines scale (area) factors for static and dynamic loads. In dynamic analysis, DAREA is used in conjunction with ACSRCE, RLOADi and TLOADi entries.

Format:

1	2	3	4	5	6	7	8	9	10
DAREA	SID	P1	C1	A1	P2	C2	A2		

Example:

DAREA	3	6	2	8.2	15	1	10.1		
-------	---	---	---	-----	----	---	------	--	--

Descriptor	Meaning
SID	Identification number. (Integer > 0)
Pi	Grid, extra, or scalar point identification number. (Integer > 0)
Ci	Component number. (Integer 1 through 6 for grid point; blank or 0 for extra or scalar point.)
Ai	Scale (area) factor. (Real)

Remarks:

1. One or two scale factors may be defined on a single entry.
2. Refer to RLOAD1, RLOAD2, TLOAD1, TLOAD2, or ACSRCE entries for the formulas that define the scale factor Ai in dynamic analysis.
3. Component numbers refer to the displacement coordinate system.
4. In dynamic analysis, DAREA entries may be used with LSEQ Bulk Data entries if LOADSET is specified in Case Control. The LSEQ and static load entries will be used to internally generate DAREA entries.
5. If DAREA is referenced by a GUST entry, Pi must be defined. However, it is only used if selected through a DLOAD Case Control command. WG from the GUST entry is used instead of Ai when requested via a GUST entry.
6. All DAREA entries corresponding to all grid and scalar points are automatically converted internally by the program to equivalent FORCE/MOMENT/SLOAD entries (as appropriate) *if there are no LSEQ Bulk Data entries*.
7. In superelement analysis, DAREA may be used to specify loads not only on the interior points of the residual, but also on the interior points of upstream superelements *if there are no LSEQ Bulk Data entries*.
8. In static analysis, DAREA entries may be used *only if there are no LSEQ Bulk Data entries*. They are ignored if there are any LSEQ Bulk Data entries.

DCONADD**Design Constraint Set Combination**

Defines the design constraints for a subcase as a union of DCONSTR entries.

Format:

1	2	3	4	5	6	7	8	9	10
DCONADD	DCID	DC1	DC2	DC3	DC4	DC5	DC6	DC7	
	DC8	-etc.-							

Example:

DCONADD	10	4	12						
---------	----	---	----	--	--	--	--	--	--

Descriptor	Meaning
DCID	Design constraint set identification number. (Integer > 0)
DCi	DCONSTR entry identification number. (Integer > 0)

Remarks:

1. The DCONADD entry is selected by a DESSUB or DESGLB Case Control command.
2. All DCi must be unique from other DCi.
3. For PART SE, DCi from different PART SEs can be referenced on a single DCONADD and only DCONADD in the main Bulk Data Section, starts with 'BEGIN BULK' or 'BEGIN SUPER=0', will be considered as part of design constraints. Note that DCONADD entries in 'BEGIN SUPER=seid' where seid>0 will be ignored.

DCONSTR

Design Constraints

Defines design constraints.

Format:

1	2	3	4	5	6	7	8	9	10
DCONSTR	DCID	RID	LALLOW/LID	UALLOW/UID	LOWFQ	HIGHFQ			

Example:

DCONSTR	10	4	1.25						
---------	----	---	------	--	--	--	--	--	--

Descriptor	Meaning
DCID	Design constraint set identification number. (Integer > 0)
RID	DRESPi entry identification number. (Integer > 0)
LALLOW/LID	Lower bound on the response quantity or the set identification ID of a number of TABLEDi entry that supplies the lower bound as a function of frequency. (Real; Default = -1.0E20)
UALLOW/UID	Upper bound on the response quantity or the set identification ID of a number of TABLEDi entry that supplies the upper bound as a function of frequency. (Real; Default = 1.0E20)
LOWFQ	Low end of frequency range in Hertz (Real \geq 0.0; Default = 0.0). See Remark 8.
HIGHFQ	High end of frequency range in Hertz (Real \geq LOWFQ; Default = 1.0E+20). See Remark 8.

Remarks:

1. The DCONSTR entry may be selected in the Case Control Section by the DESSUB or DESGLB command.
2. DCID may be referenced by the DCONADD Bulk Data entry.
3. For a given DCID, the associated RID can be referenced only once.
4. The units of LALLOW and UALLOW must be consistent with the referenced response defined on the DRESPi entry. If RID refers to an “EIGN” response, then the imposed bounds must be expressed in units of eigenvalue, (radian/time)². If RID refers to a “FREQ” response, then the imposed bounds must be expressed in cycles/time.
5. LALLOW and UALLOW are unrelated to the stress limits specified on the MATi entry.
6. Constraints are computed as follows:

$$g = \frac{LALLOW - r}{GNORM} \text{ for lower bound constraints}$$

$$g = \frac{r - UALLOW}{GNORM} \text{ for upper bound constraints}$$

$$GNORM = \begin{cases} |LALLOW| \text{ for lower bounds if } |LALLOW| > GSCAL \\ |UALLOW| \text{ for upper bounds if } |UALLOW| > GSCAL \\ GSCAL \text{ otherwise} \end{cases}$$

GSCAL is specified on the DOPTPRM entry (Default = 0.001)

7. As Remark 6. indicates, small values of UALLOW and LALLOW require special processing and should be avoided. Bounds of exactly zero are particularly troublesome. This can be avoided by using a DRESP2 entry that offsets the constrained response from zero.
8. LOWFQ and HIGHFQ fields are functional only for RTYPE with a 'FR' or a 'PSD' prefix, e.g., FRDISP or on DRESP2 or DRESP3 entries that inherit the frequency value from these RTYPES. The bounds provided in LALLOW and UALLOW are applicable to a response only when the value of forcing frequency of the response falls between the LOWFQ and HIGHFQ. If the ATTB field of the DRESP1 entry is not blank and these responses are subsequently referenced on a DRESP2 or DRESP3, the LOWFQ and HIGHFQ are ignored when the DCONSTR refers to these DRESP2/DRESP3 responses. LOWFQ/HIGHFQ are not ignored when the DCONSTR refers to a DRESP1 that has a real number in the ATTB field.
9. LID and UID are optional inputs that identify tabular input to specify the lower and upper bounds as a function of frequency. They are applicable to the 'FR' and 'PSD' responses of Remark 8. and to DRESP2 and DRESP3 responses that inherit the frequency value from these RTYPES.
10. For PART SE, DCONSTR entries can reside in each individual PART SE Bulk Data Section starting with 'BEGIN SUPER=seid'. If DCID is different from a PART SE to the next, DCONADD in the residual (or main Bulk Data Section) can be utilized to group DCONSTR entries together for a single subcase.

DDVAL**Discrete Design Variable Values**

Defines real, discrete design variable values for use in discrete variable optimization, topometry optimization or rotordynamics.

Format:

1	2	3	4	5	6	7	8	9	10
DDVAL	ID	DVAL1	DVAL2	DVAL3	DVAL4	DVAL5	DVAL6	DVAL7	

Alternate Format:

DDVAL	ID	DVAL1	THRU	DVAL2	BY	INC			
-------	----	-------	------	-------	----	-----	--	--	--

The Continuation Entry formats may be used more than once and in any order. They may also be used with either format above.

Continuation Entry Format 1:

	DVAL8	DVAL9	DVAL10	DVAL11	-etc.-				
--	-------	-------	--------	--------	--------	--	--	--	--

Continuation Entry Format 2:

	DVAL8	THRU	DVAL9	BY	INC				
--	-------	------	-------	----	-----	--	--	--	--

Example:

1	2	3	4	5	6	7	8	9	10
DDVAL	110	0.1	0.2	0.3	0.5	0.6	0.4		
	.7	THRU	1.0	BY	0.05				
	1.5	2.0							

Descriptor	Meaning
ID	Unique set identification number (Integer > 0) See Remark 1.
DVALi	Discrete values (Real, or THRU or BY) See Remarks 2. through 5.
INC	Discrete value increment (Real) See Remarks 2., 4., 5.

Remarks:

1. DDAVAL entries may be referenced by a DESVAR entry in its DDVAL field (field 8), a TOMVAR entry in its DSVID field, a entry in its DDVALID field (field 4) or a RSPINR entry in its SPTID field (field 6).

2. Trailing fields on a DDVAL record can be left blank if the next record is of type DVALi THRU DVALj BY INC. Also fields 7 - 9 must be blank when the type DVALi THRU DVALj BY INC is used in fields 2 - 6 and fields 8 - 9 must be blank when the type DVALi "THRU" DVALj "BY" INC is used in fields 3 - 7 for the first record. Embedded blanks are not permitted in other cases.
3. The DVALi values in a sequence need not be in an ascending or descending order. They can be in any order.
4. If the format DVALi THRU DVALj BY INC is employed, INC must be a positive real value if DVALi < DVALj and INC must be a negative real value if DVALi > DVALj. The program terminates the execution with an appropriate fatal error if this requirement is not satisfied.
5. The format DVALi THRU DVALj BY INC defines a list of discrete values given by DVALi, DVALi+INC, DVALi+2.0*INC, ..., DVALj. The last discrete value of DVALj is always included, even if the range is not evenly divisible by INC.
6. If Modules are present then this entry may only be specified in the main Bulk Data section.

DEACTEL**Elements To Be Deactivated for SOL 400 Analysis**

This entry identifies the elements that do not participate in the analysis or do not participate in a particular physics pass of a coupled analysis. SOL 400 for NLSTATIC, and NLTRAN, as well Perturbation analyses only.

Format:

1	2	3	4	5	6	7	8	9	10
DEACTEL	ID				ISET				

Example:

DEACTEL	2				200				
---------	---	--	--	--	-----	--	--	--	--

Descriptor	Meaning
ID	ID of the DEACTEL entry. This is referenced by a DEACTEL=ID Case Control command. (Integer; no Default)
ISET	SID of a list of elements defined by a SET3 entry. (Integer; no Default)

Remarks:

1. DEACTEL can only be referenced before the first subcase and/or inside the first subcase and/or inside the first step of the first subcase and/or inside each substep of the first step of the first subcase. References that are made anywhere else are currently ignored. It is referenced by the Case Control command DEACTEL=ID.
2. The description field (DES) of the SET3 entry referenced by ISET can only be of type ELEM.
3. Element IDs in the list originating from some set that do not exist are ignored.

DEACTEL

Define Elements that Should be Deactivated for a Particular Subcase in SOL 600

This entry allows the user to deactivate elements that have failed or are no longer necessary in a particular subcase. Some or all of these elements can be re-activated in a subsequent subcase using the ACTIVAT entry.

Format:

1	2	3	4	5	6	7	8	9	10
DEACTEL	ID	STRESS	STRAIN	IGEOM	ISET				

Example:

DEACTEL	2	1	1	0	200				
---------	---	---	---	---	-----	--	--	--	--

Descriptor	Meaning
ID	Identification number of a matching DEACTEL Case Control command defining the subcase to which these elements should be deactivated. (Integer; no Default)
STRESS	Flag to control output stresses after the elements have been deactivated. (Integer; Default = 0) <ul style="list-style-type: none"> 0 Stresses have the same value as just prior to deactivation 1 Stresses are set to zero
STRAIN	Flag to control output strains after the elements have been deactivated. (Integer; Default = 0) <ul style="list-style-type: none"> 0 Strains have the same value as just prior to deactivation 1 Strains are set to zero
IGEOM	Flag to control whether to update the geometry on the post file. (Integer; Default = 0) <ul style="list-style-type: none"> 0 Update the geometry. 1 Do not update the geometry.
ISET	ID of a list of elements described by SET3 is ID=ISET to be deactivated. (Integer; no Default)

Remarks:

1. This entry maps to Marc's DEACTIVATE (option A) History definition (option B is not supported in SOL 600).

DEFORM**Static Element Deformation**

Defines enforced axial deformation for one-dimensional elements for use in statics problems.

Format:

1	2	3	4	5	6	7	8	9	10
DEFORM	SID	EID1	D1	EID2	D2	EID3	D3		

Example:

DEFORM	1	535	.05	536	-.10				
--------	---	-----	-----	-----	------	--	--	--	--

Descriptor	Meaning
SID	Deformation set identification number. (Integer > 0)
EID _i	Element number. (Integer > 0)
D _i	Deformation. (Real; positive value represents elongation.)

Remarks:

1. The referenced element must be one-dimensional (CROD, CONROD, CTUBE, CBAR, CBEAM).
2. Deformation sets must be selected in the Case Control Section with DEFORM = SID.
3. One to three enforced element deformations may be defined on a single entry.
4. The DEFORM entry, when called by the DEFORM Case Control command, is applicable to linear static, inertia relief, differential stiffness, and buckling (Solutions 101, 105, 114, and 200) and will produce fatal messages in other solution sequences. Use SPCD to apply enforced displacements in solution sequences for which DEFORM does not apply.

DEFUSET

Degree-of-Freedom Set Name Definition

Defines new names for degree-of-freedom sets.

Format:

1	2	3	4	5	6	7	8	9	10
DEFUSET	OLD1	NEW1	OLD2	NEW2	OLD3	NEW3	OLD4	NEW4	

Example:

DEFUSET	U2	X	U4	Y	U3	Z			
---------	----	---	----	---	----	---	--	--	--

Descriptor	Meaning
OLDi	Default set name. (One to four characters)
NEWi	New set name. (One to four characters)

Remarks:

1. From one to four set names may be specified on a single entry.
2. OLDi must refer to any of the set names given in [Degree-of-Freedom Sets, 1111](#). It is recommended that OLDi refer only to the set names U1 through U6. If sets PA or PS are referenced, a user fatal message is issued.
3. All NEWi names must be unique with respect to all other set names.
4. The DEFUSET entry is optional since default set names exist for all displacement sets.
5. The DEFUSET entry must be present in the Bulk Data Section in all restarts.

DELAY**Dynamic Load Time Delay**

Defines the time delay term τ in the equations of the dynamic loading function.

Format:

1	2	3	4	5	6	7	8	9	10
DELAY	SID	P1	C1	T1	P2	C2	T2		

Example:

DELAY	5	21	6	4.25	7	6	8.1		
-------	---	----	---	------	---	---	-----	--	--

Descriptor	Meaning
SID	Identification number of the DELAY entry. (Integer > 0)
Pi	Grid, extra, or scalar point identification number. (Integer > 0)
Ci	Component number. (Integer 1 through 6 for grid point, blank or 0 for extra point or scalar point.)
Ti	Time delay τ for designated point Pi and component Ci. (Real)

Remarks:

1. One or two dynamic load time delays may be defined on a single entry.
2. SID must also be specified on a RLOAD1, RLOAD2, TLOAD1, TLOAD2, or ACSRCE entry. See those entry descriptions for the formulas that define the manner in which the time delay τ is used.
3. A DAREA, LSEQ or static load entry should be used to define a load at Pi and Ci.
4. In superelement analysis, DELAY entries may only be applied to loads on points in the residual structure.

DEQATN

Equation Definition

Defines one or more equations for use in analysis.

Format:

1	2	3	4	5	6	7	8	9	10
DEQATN	EQID				EQUATION				
					EQUATION (Cont.)				

Example:

DEQATN	14	F1(A, B, C, D, R) = A + B · C - (D ** 3 + 10.0) + sin(PI(1) · R)	
		+ A**2/(B - C) ; F = A + B - F1 · D	

Descriptor	Meaning
EQID	Unique equation identification number. (Integer > 0)
EQUATION	Equation(s). See Remarks. (Character)

Remarks:

1. EQUATION is a single equation or a set of nested equations and is specified in fields 3 through 9 on the first entry and may be continued on fields 2 through 9 on the continuation entries. On the continuation entries, no commas can appear in columns 1 through 8. All data in fields 2 through 9 must be specified in columns 9 through 72. The large-field format is not allowed.

A single equation has the following format:

variable-1 (x1, x2, ..., xn)=expression-1

A set of nested equations is separated by semicolons and has the format:

variable-1 (x1, x2, ..., xn)=expression-1;
variable-2=expression-2;variable-3=expression-3;

etc.

variable-m=expression-m

Expression-i is a collection of constants, real variables, and real functions, separated by operators, and must produce a single real value. (x1, x2, ..., xn) is the list of all the variable names (except variable-i) that appear in all expressions. Variable-i may be used in subsequent expressions. The last equation, variable-m=expression-m, provides the value that is returned to the Bulk Data entry that references EQID; e.g., DRESP2. The example above represents the following mathematical equations:

$$F1 = A + B \cdot C - (D^3 + 10) + \sin(\pi \cdot R) + \frac{A^2}{B - C}$$

$$F = A + B + F1 \cdot D$$

where SIN and PI are intrinsic functions. See Remark 4.

2. EQUATION may contain embedded blanks. EQUATION must contain less than 32,000 nonblank characters. If more characters are required for use with a DRESP2 entry, the DRESP2 can be divided into two or more DRESP2 entries with a master DRESP2 referencing subsequent DRESP2s.
3. The syntax of the expressions follows FORTRAN language standards. The allowable arithmetic operations are shown in [Table 8](#) in the order of execution precedence. Parenthesis are used to change the order of precedence. Operations within parentheses are performed first with the usual order of precedence being maintained within the parentheses.

Table 8 DEQATN Entry Operators

Operator	Operation	Sample Expressions	Interpreted As
-, +	Negative or Positive immediately preceded by exponentiation	X ** -Y	X ** (-Y)
**	Exponentiation	-X ** Y	(-X) ** Y
-, +	Negative or Positive	-X-Y	(-X)-Y
*, /	Multiplication or Division	X * Y-Z	(X * Y)-Z
+, -	Addition or Subtraction	X+Y	X+Y

4. The expressions may contain intrinsic functions. [Table 9](#) contains the format and descriptions of functions that may appear in the expressions. The use of functions that may be discontinuous must be used with caution because they can cause discontinuous derivatives. These are ABS, DIM, MAX, MIN, and MOD. For examples and further details see the [MSC Nastran DMAP Programmer's Guide](#).

Table 9 DEQATN Entry Functions

Format	Description	Mathematical Expressions
ABS(x)	absolute value	$ x $
ACOS(x)	arccosine	$\cos^{-1} x$
ACOSH(x)	hyperbolic arccosine	$\cosh^{-1} x$
ASIN(x)	arcsine	$\sin^{-1} x$
ASINH(x)	hyperbolic arcsine	$\sinh^{-1} x$
ATAN(x)	arctangent	$\tan^{-1} x$
ATAN2(x,y)	arctangent of quotient	$\tan^{-1} (x/y)$
ATANH(x)	hyperbolic arctangent	$\tanh^{-1} x$
ATANH2(x,y)	hyperbolic arctangent of quotient	$\tanh^{-1} (x/y)$

Table 9 DEQATN Entry Functions (continued)

Format	Description	Mathematical Expressions
AVG(X ₁ , X ₂ , .., X _n)	average	$\frac{1}{n} \sum_{i=1}^n X_i$
COS(x)	cosine	cos x
COSH(x)	hyperbolic cosine	cosh x
DB(P, PREF)	sound pressure in decibel	$20.0 \cdot \log\left(\frac{P}{\text{PREF}}\right)$
DBA(P, PREF, F)	sound pressure in decibel (perceived)	$20.0 \cdot \log\left(\frac{P}{\text{PREF}}\right) + 10.0 \cdot \log(\text{Ta1}) + 10.0 \cdot \log(\text{Ta2})$
DIM(x,y)	positive difference	x-MIN(x,y)
EXP(x)	exponential	e ^x
INVDB(DB, PREF)	inverse Db	$10^{\left(\frac{DB}{20.0} + \log PREF\right)}$
INVDAB(DBA, PREF, F)	inverse Dba	$10^{\left(\frac{DBA - 10.0 \cdot \log(\text{Ta1}) - 10.0 \cdot \log(\text{Ta2})}{20.0}\right)}$
LOG(x)	natural logarithm	log _e x
LOG10(x)	common logarithm	log ₁₀ x
LOGX(x,y)	base x logarithm	log _x y
MAX(x ₁ , x ₂ , ...)	maximum	maximum of x ₁ , etc.
MIN(x ₁ , x ₂ , ...)	minimum	minimum of x ₁ , etc.
MOD(x,y)	remainder (modulo)	x - y · (INT(x/y))
PI(x)	multiples of pi (π)	x · π
RSS(X ₁ , X ₂ , ..., X _n)	square root of sum of squares	$\sqrt{\sum_{i=1}^n X_i^2}$
SIN(x)	sine	sin x
SINH(x)	hyperbolic sine	sinh x
SQRT(x)	square root	\sqrt{x}

Table 9 DEQATN Entry Functions (continued)

Format	Description	Mathematical Expressions
SSQ(X ₁ , X ₂ , ..., X _n)	sum of squares	$\sum_{i=1}^n X_i^2$
SUM(X ₁ , X ₂ , ..., X _n)	summation	$\sum_{i=1}^n X_i$
TAN(x)	tangent	$\tan x$
TANH(x)	hyperbolic tangent	$\tanh x$

where

X ₁ , X ₂ , .., X _n , P	=	structure responses or acoustic pressure
PREF	=	reference pressure
F	=	forcing frequency
DB	=	acoustic pressure in Decibel
DBA	=	perceived acoustic pressure in Decibel
Ta1	=	$\frac{K_3 \cdot F^4}{(F^2 + P_2^2)(F^2 + P_3^2)}$
Ta2	=	$\frac{K_1 \cdot F^4}{(F^2 + P_1^2)^2(F^2 + P_4^2)^2}$
K1	=	2.242882e+16
K3	=	1.562339
P1	=	20.598997
P2	=	107.65265
P3	=	737.86223
P4	=	12194.22

5. If the DEQATN entry is referenced by the:

- DVCREL2, DVMREL2, or DVPREL2 entry, then X_i represents the DVIDj and LABLk fields.
- DRESP2 entry, then X_i represents the DVIDj, LABLk, NRm, Gp, DPIPq, DCICr, DMIMs, DPI2Pt, DCI2Cu, DMI2Mv, and NRRw fields in that order.
- GMLOAD, GMBC, or TEMPF entries, then
 - X₁ represents x in the basic coordinate system,
 - X₂ represents y in the basic coordinate system, and
 - X₃ represents z in the basic coordinate system.

- GMCURV entry, then
 X_1 represents line parameter u.
 - GMSURF entry, then
 X_1 represents surface parameter u and
 X_2 represents surface parameter v.
6. If the DEQATN entry is referenced by the GMLOAD, GMBC, TEMPF, GMCURV, or GMSURF entries and your computer has a short word length (e.g., 32 bits/word), then EQUATION is processed with double precision and constants may be specified in double precision; e.g., 1.2D0. If your machine has a long word length (e.g., 64 bits/word) then EQUATION is processed in single precision and constants must be specified in single precision; e.g., 1.2. If the DEQATN entry is referenced by DRESP2, DVCREL2, DVMREL2 or DVPREL2 entries, constants must be specified in single precision regardless of your machine's word length.
 7. The DMAP logical operators NOT, AND, OR, XOR, and XQV cannot be used as X_i names.
 8. Input errors on the DEQATN entry often result in poor messages. Substituting a "[" for a parenthesis or violating the restriction against large field format are examples. Known messages are UFM 215, SFM 233 and UFM 5199. If any of these messages are encountered then review the DEQATN entry input.
 9. Intrinsic functions AVG, MAX, MIN, RSS, SSQ and SUM are limited to <97 arguments. If more arguments are desired, the functions may be divided up. For example, to perform a sum of squares on 170 items, use:
$$\text{SSQ}(X1,X2,\dots,X95) + \text{SSQ}(X96,X97,\dots,X170).$$
For MAX/MIN, the following concatenation can be used:
$$\text{MAX1} = \text{MAX}(X1,X2,\dots,X95), \text{MAX2} = \text{MAX}(X96,X97\dots X170); \text{MAXT} = \text{MAX}(\text{MAX1}, \text{MAX2}).$$
For AVG, an example to average 170 terms is:
$$\text{AVG1} = \text{AVG}(X1,X2,\dots,X95); \text{AVG2} = \text{AVG}(X96,X97\dots X170); \text{AVGT} = (95. * \text{AVG1} + 75. * \text{AVG2}) / 170.$$
 10. Arithmetic is carried out using the type of the input data. For example, in the expression:

$$X = A^{**}(1/2)$$

both values in the exponent are integers so that the value returned for the exponent is calculated using integer arithmetic or $1/2 = 0$. In this case $1/2$ should be replaced by $(.5)$.

DESVAR**Design Variable**

Defines a design variable for design optimization.

Format:

1	2	3	4	5	6	7	8	9	10
DESVAR	ID	LABEL	XINIT	XLB	XUB	DELXV	DDVAL		

Example:

DESVAR	2	BARA1	35.0	10.	100.	0.2			
--------	---	-------	------	-----	------	-----	--	--	--

Descriptor	Meaning
ID	Unique design variable identification number. (Integer > 0)
LABEL	User-supplied name for printing purposes. (Character)
XINIT	Initial value. (Real; $\text{XLB} \leq \text{XINIT} \leq \text{XUB}$)
XLB	Lower bound. (Real; Default = $-1.0\text{E}+20$)
XUB	Upper bound. (Real; Default = $+1.0\text{E}+20$)
DELXV	Fractional change allowed for the design variable during approximate optimization. (Real > 0.0; for Default see Remark 2.)
DDVAL	ID of a DDVAL entry that provides a set of allowable discrete values. (Blank or Integer > 0; Default=blank for continuous design variables. See Remark 3.)

Remarks:

1. DELXV can be used to control the change in the design variable during one optimization cycle.
2. If DELXV is blank, the default is taken from the specification of the DELX parameter on the DOPTPRM entry. If DELX is not specified, then the default is 0.5.
3. If the design variable is to be discrete (Integer > 0 in DDVAL field), and if either of the XLB and/or XUB bounds are wider than those given by the discrete list of values on the corresponding DDVAL entry, XLB and/or XUB will be replaced by the minimum and maximum discrete values.

DETSPH**Spherical Detonation Wave**

Defines the ignition point from which a spherical detonation wave travels, causing the reaction of high explosive materials. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
DETSPH	DID	MID	X	Y	Z	VEL	TIME		

Example:

DETSPH	100	10	96.5	177.6	37.4	2379.	1.7E-6		
--------	-----	----	------	-------	------	-------	--------	--	--

Descriptor	Meaning
DID	Unique detonation number. (Integer > 0; Required)
MID	References MATDEUL id of the exploding material. (Integer > 0; Required)
X, Y, Z	Coordinates of the ignition point. (Real, 0.0)
VEL	Velocity of the detonation wave. (Real ≥ 0.0 , 0.0)
TIME	Detonation time. (Real ≥ 0.0 , 0.0)

Remark:

1. An element detonates when a spherical detonation wave originating from the detonation point at the specified time reaches the element.

DIVERG**Divergence Analysis Data**

Defines Mach numbers (m) for a static aeroelastic divergence analysis.

Format:

1	2	3	4	5	6	7	8	9	10
DIVERG	SID	NROOT	M1	M2	M3	M4	M5	M6	
	M7	M8	-etc.-						

Example:

DIVERG	70	2	.5	.8	.9				
--------	----	---	----	----	----	--	--	--	--

Descriptor	Meaning
SID	Unique set identifier. (Integer > 0)
NROOT	Number of divergence roots that are to be output and their eigenvectors printed. (Integer; Default = 1)
Mi	Mach number. (Real ≥ 0.0)

Remarks:

1. The DIVERG entry is referenced in Case Control by “DIVERG = SID”.
2. The NROOT lowest divergence dynamic pressures are printed. If there are fewer than NROOT pressures, all available dynamic pressures are printed.
3. Mi values must be distinct.
4. A blank Mach number field terminates the input.

DLINK**Multiple Design Variable Linking**

Relates one design variable to one or more other design variables.

Format:

1	2	3	4	5	6	7	8	9	10
DLINK	ID	DDVID	C0	CMULT	IDV1	C1	IDV2	C2	
	IDV3	C3	-etc.-						

Example:

DLINK	10	2	0.1	0.33	4	2.0	6	-1.0	
	8	7.0							

Descriptor	Meaning
ID	Unique identification number. (Integer > 0)
DDVID	Dependent design variable identification number. (Integer > 0)
C0	Constant term. (Real; Default = 0.0)
CMULT	Constant multiplier. (Real; Default = 1.0)
IDVi	Independent design variable identification number. (Integer > 0)
Ci	Coefficient i corresponding to IDVi. (Real)

Remarks:

1. DLINK defines the relationship

$$\text{DDVID} = \text{C0} + \text{CMULT} * \sum_i \text{Ci} * \cdot \text{IDVi}$$

2. This capability provides a means of linking physical design variables such as element thicknesses to nonphysical design variables such as the coefficients of interpolating functions.
3. CMULT provides a simple means of scaling the Ci. For example if Ci = 1/7, 2/7, 4/7, etc. is desired, then CMULT = 1/7 and Ci = 1, 2, 4, etc., may be input.
4. An independent IDVi must not occur on the same DLINK entry more than once.
5. ID is for user reference only.
6. If a design variable is specified as dependent on a DLINK entry, then it cannot be specified as independent on another DLINK entry.

DLOAD

Dynamic Load Combination or Superposition

Defines a dynamic loading condition for frequency response or transient response problems as a linear combination of load sets defined via RLOAD1, RLOAD2, TLOAD1, TLOAD2 and ACSRCE entries for frequency response or TLOAD1 or TLOAD2 entries for transient response.

Format:

1	2	3	4	5	6	7	8	9	10
DLOAD	SID	S	S1	L1	S2	L2	S3	L3	
	S4	L4	-etc.-	*					

Examples:

DLOAD	17	1.0	2.0	6	-2.0	7	2.0	8	
	-2.0	9							

Descriptor	Meaning
SID	Load set identification number. See Remarks 1. and 4. (Integer > 0)
S	Scale factor. See Remarks 2. and 8. (Real)
Si	Scale factors. See Remarks 2., 7. and 8. (Real)
Li	Load set identification numbers of RLOAD1, RLOAD2, TLOAD1, TLOAD2, and ACSRCE entries. See Remarks 3. and 7. (Integer > 0)

Remarks:

1. Dynamic load sets must be selected in the Case Control Section with DLOAD = SID.
2. The load vector being defined by this entry is given by
$$\{P\} = S \sum_i S_i \{P_i\}$$
3. Each Li must be unique from any other Li on the same entry.
4. SID must be unique from all ACSRCE, RLOAD1, RLOAD2, TLOAD1 and TLOAD2 dynamic load entries.
5. Nonlinear transient load sets (NOLINI entries) may not be specified on DLOAD entries. NOLINI entries are selected separately in the Case Control Section by the NONLINEAR command.
6. A DLOAD entry may not reference a set identification number defined by another DLOAD entry.
7. The scale factor Si will apply to all dynamic load entries with load set identification number of Li.
8. For RC network solver in thermal analysis, the S and Si factors are always taken as 1.
9. If Modules are present then this entry may only be specified in the main Bulk Data section.

DMI**Direct Matrix Input**

Defines matrix data blocks. Generates a matrix of the following form:

$$[\text{NAME}] = \begin{bmatrix} X_{11} & X_{12} & \dots & X_{1n} \\ X_{21} & X_{22} & \dots & X_{2n} \\ \vdots & \vdots & \ddots & \vdots \\ X_{m1} & \dots & \dots & X_{mn} \end{bmatrix}$$

where the elements X_{ij} may be real ($X_{ij} = A_{ij}$) or complex ($X_{ij} = A_{ij} + iB_{ij}$). The matrix is defined by a single header entry and one or more column entries. Only one header entry is required. A column entry is required for each column with nonzero elements.

Header Entry Format:

1	2	3	4	5	6	7	8	9	10
DMI	NAME	"0"	FORM	TIN	TOUT		M	N	

Column Entry Format for Real Matrices:

DMI	NAME	J	I1	A(I1,J)	A(I1+1,J)		-etc.-	I2	
		A(I2,J)		-etc.-					

Column Entry Format for Complex Matrices:

DMI	NAME	J	I1	A(I1,J)	B(I1,J)	A(I1+1,J)	B(I1+1,J)	-etc.-	
		I2	A(I2,J)	B(I2,J)	-etc.-				

Example of a Real Matrix:

DMI	BBB	0	2	1	1		4	2	
DMI	BBB	1	1	1.	3.	5.			
DMI	BBB	2	2	6.	4	8.			

$$\text{BBB} = \begin{bmatrix} 1.0 & 0.0 \\ 3.0 & 6.0 \\ 5.0 & 0.0 \\ 0.0 & 8.0 \end{bmatrix}$$

Example of a Complex Matrix:

DMI	QQQ	0	2	3	3		4	2	
DMI	QQQ	1	1	1.0	2.0	3.0	0.0	3	

	5.0	6.0							
DMI	QQQ	2	2	6.0	7.0	4	8.0	9.0	

$$[QQQ] = \begin{bmatrix} 1.0 + 2.0i & 0.0 + 0.0i \\ 3.0 + 0.0i & 6.0 + 7.0i \\ 5.0 + 6.0i & 0.0 + 0.0i \\ 0.0 + 0.0i & 8.0 + 9.0i \end{bmatrix}$$

Descriptor	Meaning
NAME	Name of the matrix. See Remark 1. Name is used to reference the data block in the DMAP sequence. (One to eight alphanumeric characters, the first of which must be alphabetic.)
FORM	Form of matrix, as follows: (Integer)
	1 Square matrix (not symmetric) 2 General rectangular matrix 3 Diagonal matrix (M=number of rows, N = 1) 4 Lower triangular factor 5 Upper triangular factor 6 Symmetric matrix 8 Identity matrix (M=number of rows, N = M)
TIN	Type of matrix being input, as follows: (Integer)
	1 Real, single precision (one field used/element) 2 Real, double precision (one field used/element) 3 Complex, single precision (two fields used/element) 4 Complex, double precision (two fields used/element)

Descriptor	Meaning
TOUT	Type of matrix being output, as follows: (Integer) <ul style="list-style-type: none"> 0 Set by precision cell 1 Real, single precision 2 Real, double precision 3 Complex, single precision 4 Complex, double precision
M	Number of rows in NAME. (Integer > 0)
N	Number of columns in NAME. Except for FORM 3 and 8. (Integer > 0)
"0"	Indicates the header entry.
J	Column number of NAME. (Integer > 0)
I1, I2, etc.	Row number of NAME, which indicates the beginning of a group of nonzero elements in the column. See Remark 13. (Integer > 0)
A(Ix,J)	Real part of element (see TIN). (Real)
B(Ix,J)	Imaginary part of element (see TIN). (Real)

Remarks:

1. In order to use the DMI feature, the user must write a DMAP, or make alterations to a solution sequence that includes the DMIIN module. See the [MSC Nastran DMAP Programmer's Guide](#). All of the rules governing the use of data blocks in DMAP sequences apply.
2. The total number of DMIs and DTIs may not exceed 1000.
3. Field 3 of the header entry must contain an integer of zero (0).
4. For symmetric matrices, the entire matrix must be input.
5. Only nonzero terms need be entered.
6. Leading and trailing zeros in a column do not have to be entered. However, a blank field between nonzero fields on this entry is not equivalent to a zero. If a zero input is required, the appropriate type zero must be entered (i.e., 0.0 or 0.0D0).
7. Complex input must have both the real and imaginary parts entered if either part is nonzero; i.e., the zero component must be input explicitly.
8. If A(Ix,J) is followed by "THRU" in the next field and an integer row number "IX" after the THRU, then A(Ix,J) will be repeated in each row through IX. The "THRU" must follow an element value. For example, the entries for a real matrix RRR would appear as follows:

1	2	3	4	5	6	7	8	9	10
DMI	NAME	J	I1	A(I1,J)			I1	A(I2,J)	
DMI	RRR	1	2	1.0	THRU	10	12	2.0	

These entries will cause the first column of the matrix RRR to have a zero in row 1, the values 1.0 in rows 2 through 10, a zero in row 11, and 2.0 in row 12.

9. Each column must be a single logical entry. The terms in each column must be specified in increasing row number order.
10. The “FORM” options 4, 5, and 8 are nonstandard forms and may be used only in conjunction with the modules indicated in [Table 10](#).

Table 10 DMI FORM Options

FORM	Matrix Description	Modules			
		ADD	FBS	MATPRN	MPYAD
4	Lower Triangular Factor		X	X	
5	Upper Triangular Factor		X	X	
8	Identity	X	X	X	X

11. Form 3 matrices are converted to Form 6 matrices, which may be used by any module.
12. Form 7 matrices may not be defined on this entry.
13. I1 must be specified. I2, etc. are not required if their matrix elements follow the preceding element in the next row of the matrix. For example, in the column entry for column 1 of QQQ, neither I2 nor I3 is specified.
14. The DMIG entry is more convenient for matrices with rows and columns that are referenced by grid or scalar point degrees-of-freedom.
15. If Modules are present then this entry may only be specified in the main Bulk Data section.

DMDIA**Direct Matrix Input for Axisymmetric Analysis**

Defines axisymmetric (fluid or structure) related direct input matrix terms.

The matrix is defined by a single header entry and one or more column entries. Only one header entry is required. A column entry is required for each column with nonzero elements.

Header Entry Format:

1	2	3	4	5	6	7	8	9	10
DMDIA	NAME	"0"	IFO	TIN	TOUT				

Column Entry Format:

DMDIA	NAME	GJ	CJ	NJ					
	G1	C1	N1	A1	B1				
	G2	C2		-etc.-					

Example:

DMDIA	B2PP	0	1	3	4				
DMDIA	B2PP	32							
	1027	3		4.25+6	2.27+3				

Descriptor	Meaning
NAME	Name of the matrix. See Remark 2. (One to eight alphanumeric characters, the first of which is alphabetic.)
IFO	Form of matrix: (Integer)
	1 Square matrix
	2 General rectangular matrix
	6 Symmetric matrix
TIN	Type of matrix being input: (Integer)
	1 Real, single precision (One field is used per element.)
	3 Complex, single precision (Two fields are used per element.)

Descriptor	Meaning
TOUT	Type of matrix that will be created: (Integer) <ul style="list-style-type: none"> 1 Real, single precision 2 Real, double precision 3 Complex, single precision 4 Complex, double precision
GJ, Gi	Grid, scalar, RINGFL fluid point, PRESPT pressure point, FREEPT free surface displacement, or extra point identification number. (Integer > 0)
CJ, Ci	Component number for GJ or Gi grid point ($0 \leq \text{Integer} \leq 6$; Blank or zero if GJ or Gi is a scalar, fluid, or extra point.)
NJ, Ni	Harmonic number of RINGFL point. Must be blank if a point type other than RINGFL is used. A negative number implies the “sine” series; a positive number implies the “cosine” series. (Integer)
Ai, Bi	Real and imaginary parts of matrix element; row (Gi, Ci, Ni) column (GJ, CJ, NJ). If the matrix is real (TIN = 1), then Bi must be blank.

1. This entry is allowed only if an AXIF entry is also present.
2. Matrices defined on this entry may be used in dynamics by selection with the Case Control commands K2PP = NAME, B2PP = NAME, or M2PP = NAME for $[K_{pp}^2]$, $[B_{pp}^2]$, or $[M_{pp}^2]$, respectively. See [Superelement Analysis](#) in the *MSC Nastran Reference Guide*.
3. Field 3 or the header entry must contain an integer 0.
4. For symmetric matrices, either the upper or the lower triangle terms may be specified, but not both.
5. Only nonzero terms need be entered.
6. If any DMIAX entry is changed or added on restart then a complete re-analysis may be performed. Therefore, DMIAX entry changes or additions are not recommended on restart.

DMIG**Direct Matrix Input at Points**

Defines direct input matrices related to grid, extra, and/or scalar points. The matrix is defined by a single header entry and one or more column entries. A column entry is required for each column with nonzero elements.

Header Entry Format:

1	2	3	4	5	6	7	8	9	10
DMIG	NAME	"0"	IFO	TIN	TOUT	POLAR		NCOL	

Column Entry Format:

DMIG	NAME	GJ	CJ		G1	C1	A1	B1	
		G2	C2	A2	B2	-etc.-			

Example:

DMIG	STIF	0	1	3	4				
DMIG	STIF	27	1		2	3	3.+5	3.+3	
		2	4	2.5+10	0.	50		1.0	0.

Descriptor	Meaning
NAME	Name of the matrix. See Remark 1. (One to eight alphanumeric characters, the first of which is alphabetic.)
IFO	Form of matrix input. IFO = 6 must be specified for matrices selected by the K2GG, M2GG, and B2GG Case Control commands. (Integer)
	1 Square
	2 or 9 Rectangular
	6 Symmetric
TIN	Type of matrix being input: (Integer)
	1 Real, single precision (One field is used per element.)
	2 Real, double precision (One field is used per element.)
	3 Complex, single precision (Two fields are used per element.)
	4 Complex, double precision (Two fields are used per element.)
TOUT	Type of matrix that will be created: (Integer)
	0 Set by precision system cell (Default)
	1 Real, single precision
	2 Real, double precision
	3 Complex, single precision

Descriptor	Meaning
	4 Complex, double precision
POLAR	Input format of Ai, Bi. (Integer=blank or 0 indicates real, imaginary format; Integer > 0 indicates amplitude, phase format.)
NCOL	Number of columns in a rectangular matrix. Used only for IFO = 9. See Remarks 5. and 6. (Integer > 0)
GJ	Grid, scalar or extra point identification number for column index. (Integer > 0)
CJ	Component number for grid point GJ. (0 < Integer \leq 6; blank or zero if GJ is a scalar or extra point.)
Gi	Grid, scalar, or extra point identification number for row index. (Integer > 0)
Ci	Component number for Gi for a grid point. (0 < CJ \leq 6; blank or zero if Gi is a scalar or extra point.)
Ai, Bi	Real and imaginary (or amplitude and phase) parts of a matrix element. If the matrix is real (TIN = 1 or 2), then Bi must be blank. (Real)

Remarks:

1. Matrices defined on this entry may be used in dynamics by selection in the Case Control with K2PP = NAME, B2PP = NAME, M2PP = NAME for $[K_{pp}]$, $[B_{pp}]$, or $[M_{pp}]$, respectively. Matrices may also be selected for all solution sequences by K2GG = NAME, B2GG = NAME, and M2GG = NAME. The g-set matrices are added to the structural matrices before constraints are applied, while p-set matrices are added in dynamics after constraints are applied. Load matrices may be selected by P2G = NAME for dynamic and superelement analyses.
2. The header entry containing IFO, TIN and TOUT is required. Each nonnull column is started with a GJ, CJ pair. The entries for each row of that column follows. Only nonzero terms need be entered. The terms may be input in arbitrary order. A GJ, CJ pair may be entered more than once, but input of an element of the matrix more than once will produce a fatal message.
3. Field 3 of the header entry must contain an integer 0.
4. For symmetric matrices (IFO = 6), a given off-diagonal element may be input either below or above the diagonal. While upper and lower triangle terms may be mixed, a fatal message will be issued if an element is input both below and above the diagonal.
5. The recommended format for rectangular matrices requires the use of NCOL and IFO = 9. The number of columns in the matrix is NCOL. (The number of rows in all DMIG matrices is always either p-set or g-set size, depending on the context.) The GJ term is used for the column index. The CJ term is ignored.
6. If NCOL is not used for rectangular matrices, two different conventions are available:
 - If IFO = 9, GJ and CJ will determine the sorted sequence, but will otherwise be ignored; a rectangular matrix will be generated with the columns submitted being in the 1 to N positions, where N is the number of logical entries submitted (not counting the header entry).

- If IFO = 2, the number of columns of the rectangular matrix will be equal to the index of the highest numbered non-null column (in internal sort). Trailing null columns of the g- or p-size matrix will be truncated.
7. The matrix names must be unique among all DMIGs.
 8. TIN should be set consistent with the number of decimal digits required to read the input data adequately. For a single-precision specification on a short-word machine, the input will be truncated after about eight decimal digits, even when more digits are present in a double-field format. If more digits are needed, a double precision specification should be used instead. However, note that a double precision specification requires a “D” type exponent even for terms that do not need an exponent. For example, unity may be input as 1.0 in single precision, but the longer form 1.0D0 is required for double precision.
 9. On long-word machines, almost all matrix calculations are performed in single precision and on short-word machines, in double precision. It is recommended that DMIG matrices also follow these conventions for a balance of efficiency and reliability. The recommended value for TOUT is 0, which instructs the program to inspect the system cell that measures the machine precision at run time and sets the precision of the matrix to the same value. TOUT = 0 allows the same DMIG input to be used on any machine. If TOUT is contrary to the machine type specified (for example, a TOUT of 1 on a short-word machine), unreliable results may occur.
 10. If any DMIG entry is changed or added on restart then a complete re-analysis is performed. Therefore, DMIG entry changes or additions are not recommended on restart.

DMIG,UACCEL**Direct Matrix Input of Enforced Static Acceleration**

Defines rigid body accelerations in the basic coordinate system.

Format:

1	2	3	4	5	6	7	8	9	10
DMIG	UACCEL	"0"	"9"	TIN				NCOL	
DMIG	UACCEL	L			G1	C1	X1		
	G2	C2	X2		G3	C3	X3		

Example:

DMIG	UACCEL	0	9	1				4	
DMIG	UACCEL	2			2	3	386.4		
DMIG	UACCEL	3			2	4	3.0		
DMIG	UACCEL	4			2	6	1.0		

Descriptor	Meaning
TIN	Type of matrix being input. (Integer 1 or 2)
1	Real, single precision (One field is used per element.)
2	Real, double precision (One field is used per element.)
NCOL	Number of columns, see Remark 2. Default is the number of columns specified. (Integer > 0)
L	Load sequence number. (Integer > 0)
Gi	Grid point identification number of a single reference point. (Integer > 0)
Ci	Component number for Gi in the basic coordinate system. See Remark 4. ($0 < \text{Integer} \leq 6$)
Xi	Value of enforced acceleration term in the basic coordinate system. (Real)

Remarks:

1. DMIG,UACCEL is an optional entry when PARAM,INREL,-1 is specified in SOLs 101 or 200. If DMIG,UACCEL is present, the loads applied to the structure are the sum of the conventional applied loads plus the inertia loads resulting from the rigid body accelerations defined on this entry. If it is not present, conventional inertia relief calculations are performed.
2. The load sequence number interpretation depends on the value of the NCOL field. The recommended method is to set it equal to the number of loading conditions. The load sequence number L is then the sequence number of the subcase to which the applied acceleration will be applied.

3. The grid point identification number listed on Gi defines a single grid point on the model where loads will be applied to cause the enforced acceleration state. Gi must also appear on a SUPPORT Bulk Data entry. It must also appear on a PARAM,GRDPNT entry. In superelement analysis, it must be a residual structure point exterior to all superelements.
4. The Xi value is the enforced acceleration at grid point Gi. The translation and rotation components are in consistent units and will be applied in the basic coordinate system regardless of the displacement coordinate system specified for Gi (CD field on GRID entry).
5. Only nonzero terms need be entered.
6. See [Superelement Analysis](#) in the *MSC Nastran Reference Guide* for the theoretical basis of inertia relief with superelements.
7. If any DMIG entry is changed or added on restart then a complete re-analysis is performed.
Therefore, DMIG entry changes or additions are not recommended on restart.

DMIGOUT**DMIG Matrices to be Output from the Marc Portion of SOL 600**

Defines DMIG matrices to be output from the Marc Portion of SOL 600.

Header Entry Format:

1	2	3	4	5	6	7	8	9	10
DMIGOUT	ID	ISTIFF	IDIFF	IMASS	IDAMP	ICOND	ISPECIF	ISOL	
		ICTRL	IFREQ	ICORD	KIND	AMIN	IUSEK	IUSEM	
		IE1	THRU	IE2		IE3	THRU	IE4	

Example:

DMIGOUT	100	1	1	0	0	0	0	103	
		2	-1	2	2	1.0E-16			

Descriptor	Meaning
ID	Subcase for which the reduced matrices will be output. ID must correlate to a SUBCASE Case Control ID, for example if the Case Control contains SUBCASE 20, ID would be 20. To output matrices in Marc's phase zero, etc ID=0. (Integer; Default = 1, see Remark 11.)
ISTIFF	Flag to output stiffness matrices. (Normally contains the differential stiffness and all other stiffness terms.) (Integer; Default = 0)
	0 Do not output the matrix
	1 Output the matrix
IDIFF	Flag to output differential stiffness matrices. (Only available for buckling analysis or subcases.) (Integer; Default = 0)
	0 Do not output the matrix
	1 Output the matrix
IMASS	Flag to output mass matrices. (Integer; Default = 0)
	0 Do not output the matrix
	1 Output the matrix (must be a dynamic analysis)
IDAMP	Flag to output damping matrices. (Integer; Default = 0)
	0 Do not output the matrix
	1 Output the matrix (must be a dynamic analysis)
ICOND	Flag to output conductivity matrices. (Integer; Default = 0)
	0 Do not output the matrix
	1 Output the matrix
ISPECIF	Flag to output specific heat matrices. (Integer; Default = 0)

Descriptor	Meaning
	0 Do not output the matrix 1 Output the matrix
ISOL	Solution sequence to run using the DMIG matrices. (Integer absolute value > 100; Default = 0, which means do not run any solution sequence using the DMIG's created by Marc in this execution). To speed up the solution, use DOMAINsolver ACMS (PARTOPT=DOF) for eigenvalues and set ISOL to the negative value of the solution sequence desired (-103, -111 or -112).
ICTRL	Controls type of matrix. (Integer; Default = 2) 1 Element matrices 2 Global matrices 3 Element and global matrices
IFREQ	Controls how often the matrices are output (Integer; no Default) 1 Output at every increment 2 Output every other increment, etc. 3 Output every 3rd increment, etc. -1 Output only at start of the subcase. (See Remark 12.)
ICORD	Controls matrix output coordinate system (Integer; no Default) 1 Nastran basic coordinate system 2 Current transformed coordinate system (at the start of the run, this is the Nastran global coordinate system)
KIND	Controls which elements are written (Integer; Default = 2) 1 A list of elements starting with the 3 rd entry will be specified 2 All elements will be written
AMIN	Values below AMIN will not be written (Real; Default = 1.0E-15). Values below AMIN will be skipped.
IUSEK	Increment to use in SOL 600 CONTINUE option for the stiffness matrix (if CONTINUE is specified on the SOL 600 entry). (Integer; Default = -1) Increments 0 to 9999 may be specified. If -1 is specified, the last increment output by Marc will be used. As an example, the Marc DMIG file for the global stiffness matrix for increment 10 will have the name jid.marc_cglsti_0020.
IUSEM	Increment to use in SOL 600 CONTINUE option for the mass matrix (If CONTINUE is specified on the SOL 600 entry). (Integer, Default = -1) Increments 0 to 9999 may be specified. If -1 is specified, the last increment output by Marc will be used. As an example, the Marc DMIG file for the global mass matrix for increment 20 will have the name jid.marc_cgmas_0020.

Descriptor	Meaning
IE1, IE3	Starting element number of a range of elements (Integer; no Default) Only enter IE1 if KIND=1
IE2, IE4	Ending element number of a range of elements (Integer; no Default) Only enter IE2 if KIND=1 (IE2 is required if IE1 is entered).

Remarks:

1. The first continuation line is required.
2. The second continuation line should only be entered if KIND=1 and may be repeated as many times as necessary to define all applicable elements.
3. ICTRL, IFREQ, ICORD, KIND and AMIN apply to all types of matrices to be written for the subcase.
4. Only one DMIGOUT entry can be entered per subcase. If more than one is entered, only the first encountered will be used.
5. DMIGOUT entries may be made for each subcase desired.
6. Marc DMIG element output will be in files named jid.marc_dmigXX_inc where XX is shown below and inc is the increment number.

ST	Stiffness Matrices
DF	Differential stiffness matrices
MS	Mass matrices
DM	Damping
CO	Conductivity matrices
SP	Specific heat matrices

Global matrices are named jid_marc_cglsti_inc.

7. If the SOL 600 CONTINUE options is invoked, Case Control commands and a Bulk Data include statement to receive the matrices will be automatically added to the original input data file. A second Nastran execution will be spawned from the original Nastran execution after completion of the Marc execution.
8. ID may not be 600 or 700 in the Executive Control statement, SOL 600, ID.
9. The following Bulk Data parameters are usually required in addition to the DMIGOUT entry:

\$2345678x234567890123456x34567890123456

param* mrspawn2 nastcmd
param,mrmtxnam,KAAK
param,marcfile,nastb.rc

where



- a. nastcmd is the name of the command to run the primary and continuation jobs (examples are nastran, nast2006t1, nast2006t2, etc.)
 - b. nastb.rc should be changed to the name of the rc file to be used for the continuation run. It usually will specify mem= with a larger value than that of the primary run and also include a line bat=no (except for windows systems).
 - c. PARAM,MARCFILi should not be used.
10. For standard nonlinear static or dynamic analyses the stiffness matrix contains all contributions including the differential stiffness matrix and it is not possible to obtain the differential stiffness matrix separately. For a buckling analysis, the differential stiffness matrix may be obtained separately.
11. To obtain stiffness and mass matrices for SOL 600,103, ID in field 2 of this entry must be zero.
12. If IFREQ is 1 or a small number, the number of matrices output for dynamic analyses (SOL 600,109 or SOL 600,120) can be extremely large.
13. If ID=0, IFREQ should be 1.
14. Setting IFREQ to a value larger than the actual number of increments in a subcase will produce no matrices.

DMIGROT**DMIG Matrices for which large rotation in SOL 600**

Defines large rotation and other characteristics of a matrix entered using DMIG in SOL 600.

Format for K2GG and K2PP Stiffness DMIG Matrices:

1	2	3	4	5	6	7	8	9	10
DMIGROT	Name	G1	G2	G3	SCALE	ITRAN	IACT	IDEACT	

Format for M2GG and M2PP Mass DMIG Matrices:

DMIGROT	Name				SCALE		IACT		
---------	------	--	--	--	-------	--	------	--	--

Example:

DMIGROT	KAAX	375							
DMIGROT	KAAY	401	402	403					
DMIGROT	KAAX				1.25	1	2	5	
DMIGROT	MAAX				0.001		7	8	

Descriptor	Meaning
Name	Name of the DMIG matrix for which large rotation is specified. See Remark 1. (Character; Required field; no Default)
G1	Grid ID specifying the rotation of the matrix. See Remark 2. (Integer; Required field; no Default)
G2	Grid ID which combined with G1 and G3 specifies a plane whose rotation provides the rigid body rotation of the matrix. See Remark 2. (Integer; Default = 0;)
G3	Grid ID which combined with G1 and G2 specifies a plane whose rotation provides the rigid body rotation of the matrix. See Remark 2. (Integer; Default = 0)
SCALE	Scale factor. Each term in the DMIG matrix will be scaled by this value. (Real; Default = 1.0)
ITRAN	Flag indicating whether to transform the DMIG's or not. (Integer, Default = 0) <ul style="list-style-type: none"> 0 Do not transform the DMIG matrix. 1 Transform the DMIG using G1, G2, G3 to form the transformation matrix.
IACT	Subcase ID for which this DMIG matrix will be activated. See Remark 5. (Integer; Default = 0). <ul style="list-style-type: none"> 0 The matrix is active for all subcases. N The matrix is activated starting with subcase N
IDEACT	Subcase ID for which this DMIG matrix will be deactivated. See Remark 5. (Integer; no Default; leave blank if the matrix should be active fro the entire analysis or starting with subcase IACT)

Remarks:

1. All DMIG matrices with the name specified in field 2 will rotate with grids G1.
2. If G1 has 6 dof it is not necessary to specify G2 or G3 and the DMIG matrix will rotate with the same rotation as G1. If G1 has fewer than 6 dof, or if it is desired that the DMIG matrix rotate with a plane defined by G1, G2 and G3, G2 and G3 must be entered and may not all be collinear.
3. If a transformation is to be applied to the stiffness matrix, the DMIG **must** contain all of the degrees of freedom associated with the node to which the transformation is applied.
4. This bulk data entry maps to Marc's K2GG (and K2PP) model definition entry.
5. IACT and IDEACT are ID's of the Case Control SUBCASE entry.
6. G1, G2, G3 and ITRAN may not be entered for mass matrices.

DMIJ**Direct Matrix Input at js-Set of the Aerodynamic Mesh**

Defines direct input matrices related to collation degrees-of-freedom (js-set) of aerodynamic mesh points for CAERO1, CAERO3, CAERO4 and CAERO5 and for the slender body elements of CAERO2. These include W2GJ, FA2J and input pressures and downwashes associated with AEPRESS and AEDW entries. The matrix is described by a single header entry and one or more column entries. A column entry is required for each column with nonzero elements. For entering data for the interference elements of a CAERO2, use DMIJI or DMI.

Header Entry Format:

1	2	3	4	5	6	7	8	9	10
DMIJ	NAME	"0"	IFO	TIN	TOUT	POLAR		NCOL	

Column Entry Format:

DMIJ	NAME	GJ	CJ		G1	C1	A1	B1	
		G2	C2	A2	B2	-etc.-			

Example:

DMIJ	ALPH1	0	9	2	0			1	
DMIJ	ALPH1	1	1		1	1	.1		
		2	1	.1					

Descriptor	Meaning
NAME	Name of the matrix. See Remark 1. (One to eight alphanumeric characters, the first of which is alphabetic.)
IFO	Form of matrix being input. (Integer)
	1 Square
	9 or 2 Rectangular
	6 Symmetric
TIN	Type of matrix being input: (Integer)
	1 Real, single precision (One field is used per element)
	2 Real, double precision (One field is used per element)
	3 Complex, single precision (Two fields are used per element)
	4 Complex, double precision (Two fields are used per element)
TOUT	Type of matrix being created: (Integer)
	0 Set by precision system cell (Default)
	1 Real, single precision

Descriptor	Meaning
2	Real, double precision
3	Complex, single precision
4	Complex, double precision
POLAR	Input format of Ai, Bi. (Integer = blank or 0 indicates real, imaginary format. Integer > 0 indicated magnitude, phase format.)
NCOL	Number of columns in a rectangular matrix. Used only for IFO = 9. (Integer > 0)
GJ	Grid, scalar or extra point identification number for column index. (Integer > 0)
CJ	Component number for grid point GJ. ($0 < \text{Integer} \leq 6$; blank or zero if GJ is a scalar or extra point.)
Gi	Grid, scalar, or extra point identification number for row index. (Integer > 0)
Ci	Component number for Gi for a grid point. ($0 < CJ \leq 6$; blank or zero if Gi is a scalar or extra point.)
Ai, Bi	Real and imaginary (or amplitude and phase) parts of a matrix element. If the matrix is real (TIN = 1 or 2), then Bi must be blank. (Real)

Remarks:

1. Matrices defined on this entry are referenced in static aeroelastic analysis by reference on AEDW and/or AEPRESS entries. In that paradigm, a single column is required. Also, DMIJ may also be used for the W2GJ and FA2J entries. Again, a single column is required. If both DMI and DMIJ are specified for W2GJ or FA2J, the DMI entry will be used. DMI may NOT be used for AEDW and AEPRESS.
2. The js-set DOF's for each aerodynamic theory are limited to the six-DOF paradigm (3 translations and 3 rotations). However, particular DOF's are permanently SPC'd based on the theory's ability to support those degrees-of-freedom. Unlike the DMIG entry, DMIJ data will be partitioned to the j-set, not reduced. No warnings are issued about truncated data. The j-set DOF's for each aerodynamic element/grid are highly method dependent. The following data define the known set, but the j-set definition is somewhat arbitrary in the general (external aerodynamics) paradigm.

Entry Type	COMP				
	1	2	3	5	6
CAERO1			X		
CAERO2-Y		X			
CAERO2-Z			X		
CAERO2-ZY		X	X		

3. The header entry containing IFO, TIN and TOUT is required. Each nonnull column is started with a GJ, CJ pair. The entries for each row of that column then follow. Only nonzero terms need be entered. The terms may be input in arbitrary order. A GJ, CJ pair may be entered more than once, but the input of an element of the matrix more than once results in a fatal message.
4. Field 3 of the header entry must contain an integer 0.
5. For symmetric matrices (very rare in the j-set!) (IFO=6), a given off-diagonal element may be input either below or above the diagonal. While upper and lower triangle terms may be mixed, a fatal message will be issued if an element is input both above and below the diagonal.
6. The recommended format for rectangular matrices requires the use of NCOL and IFO = 9. the number of columns in the matrix is NCOL. (The number of rows in all DMIJ matrices is always the js-set size--the union of the j-set and the permanently SPC'd partition). The GJ term is used for the column index. the CJ term is ignored.
7. If NCOL is not used for rectangular matrices, two different conventions are available:
 - If IFO = 9, GJ and CJ will determine the sorted sequence, but will otherwise be ignored; a rectangular matrix will be generated with the columns submitted being in the 1 to N positions, where N is the number of logical entries submitted (not counting the header entry).
 - If IFO = 2, the number of columns of the rectangular matrix will be equal to the index of the highest numbered nonnull column (in internal sort). Trailing null columns of the js-size matrix will be truncated.
8. The matrix names must be unique among all DMIJ.
9. TIN should be consistent with the number of decimal digits required to read the input data adequately. For a single precision specification on a short word machine, the input will be truncated after about eight decimal digits, even when more digits are present in a double field format. If more digits are needed, a double precision specification should be used instead. However, note that a double precision specification requires a "D" type exponent even for terms that do not need an exponent. For example, unity may be input as 1.0 in single precision, but the longer form 1.0D0 is required for double precision.
10. On long word machines, almost all matrix calculations are performed in single precision and, on short word machines, in double precision. It is recommended that DMIJ matrices also follow these conventions for a balance of efficiency and reliability. The recommended value for TOUT is 0, which instructs the program to inspect the system cell that measures the machine precision at run time and sets the precision of the matrix to the same value. TOUT = 0 allows the same DMIJ input to be used on any machine. If TOUT is contrary to the machine type specified, unreliable results may occur.
11. If any DMIJ entry is changed or added on restart then a complete reanalysis is performed. Therefore, DMIJ entry changes or additions are not recommended on restart.

DMIJI

Direct Matrix Input at js-Set of the Interference Body

Defines direct input matrices related to collation degrees-of-freedom (js-set) of aerodynamic mesh points for the interference elements of CAERO2. These include W2GJ, FA2J and input pressures and downwashes associated with AEPRESS and AEDW entries. The matrix is described by a single header entry and one or more column entries. A column entry is required for each column with nonzero elements. For entering data for the slender elements of a CAERO2, or a CAERO1, 3, 4 or 5 use DMIJ or DMI.

Header Entry Format:

1	2	3	4	5	6	7	8	9	10
DMIJI	NAME	"0"	IFO	TIN	TOUT	POLAR		NCOL	

Column Entry Format:

DMIJ	NAME	GJ	CJ		G1	C1	A1	B1	
		G2	C2	A2	B2	-etc.-			

Example:

DMIJI	ALPH1	0	9	2	0			1	
DMIJI	ALPH1	1	1		1	1	.1		
		2	1	.1					

Descriptor	Meaning
NAME	Name of the matrix. See Remark 1. (One to eight alphanumeric characters, the first of which is alphabetic.)
IFO	Form of matrix being input. (Integer)
	1 Square
	9 or 2 Rectangular
	6 Symmetric
TIN	Type of matrix being input: (Integer)
	1 Real, single precision (One field is used per element)
	2 Real, double precision (One field is used per element)
	3 Complex, single precision (Two fields are used per element)
	4 Complex, double precision (Two fields are used per element)
TOUT	Type of matrix being created: (Integer)
	0 Set by precision system cell (Default)
	1 Real, single precision
	2 Real, double precision

Descriptor	Meaning
3	Complex, single precision
4	Complex, double precision
POLAR	Input format of Ai, Bi. (Integer = blank or 0 indicates real, imaginary format. Integer > 0 indicated magnitude, phase format.)
NCOL	Number of columns in a rectangular matrix. Used only for IFO = 9. (Integer > 0)
GJ	Grid, scalar or extra point identification number for column index. (Integer > 0)
CJ	Component number for grid point GJ. (0 < Integer ≤ 6; blank or zero if GJ is a scalar or extra point.)
Gi	Grid, scalar, or extra point identification number for row index. (Integer > 0)
Ci	Component number for Gi for a grid point. (0 < CJ ≤ 6; blank or zero if Gi is a scalar or extra point.)
Ai, Bi	Real and imaginary (or amplitude and phase) parts of a matrix element. If the matrix is real (TIN = 1 or 2), then Bi must be blank. (Real)

Remarks:

1. Matrices defined on this entry are referenced in static aeroelastic analysis by reference on AEDW and/or AEPRESS entries. In that paradigm, a single column is required. DMI may NOT be used for AEDW and AEPRESS.
2. The js-set DOF's for each aerodynamic theory are limited to the six-DOF paradigm (3 translations and 3 rotations). However, particular DOF's are permanently SPC'd based on the theory's ability to support those degrees-of-freedom. Unlike the DMIG entry, DMIJI data will be partitioned to the j-set, not reduced. No warnings are issued about truncated data. The j-set DOF's for each aerodynamic element/grid are highly method dependent. The following data define the known set, but the j-set definition is somewhat arbitrary in the general (external aerodynamics) paradigm.

Entry Type	COMP				
	1	2	3	5	6
CAERO2-Y		X			
CAERO2-Z			X		
CAERO2-ZY		X	X		

3. The header entry containing IFO, TIN and TOUT is required. Each nonnull column is started with a GJ, CJ pair. The entries for each row of that column then follow. Only nonzero terms need be entered. The terms may be input in arbitrary order. A GJ, CJ pair may be entered more than once, but the input of an element of the matrix more than once results in a fatal message.
4. Field 3 of the header entry must contain an integer 0.

5. For symmetric matrices (very rare in the j-set!) (IFO=6), a given off-diagonal element may be input either below or above the diagonal. While upper and lower triangle terms may be mixed, a fatal message will be issued if an element is input both above and below the diagonal.
6. The recommended format for rectangular matrices requires the use of NCOL and IFO = 9. the number of columns in the matrix is NCOL. (The number of rows in all DMIJI matrices is always the js-set size--the union of the j-set and the permanently SPC'd partition). The GJ term is used for the column index. the CJ term is ignored.
7. If NCOL is not used for rectangular matrices, two different conventions are available:
 - If IFO = 9, GJ and CJ will determine the sorted sequence, but will otherwise be ignored; a rectangular matrix will be generated with the columns submitted being in the 1 to N positions, where N is the number of logical entries submitted (not counting the header entry).
 - If IFO = 2, the number of columns of the rectangular matrix will be equal to the index of the highest numbered nonnull column (in internal sort). Trailing null columns of the js-size matrix will be truncated.
8. The matrix names must be unique among all DMIJI.
9. TIN should be consistent with the number of decimal digits required to read the input data adequately. For a single precision specification on a short word machine, the input will be truncated after about eight decimal digits, even when more digits are present in a double field format. If more digits are needed, a double precision specification should be used instead. However, note that a double precision specification requires a "D" type exponent even for terms that do not need an exponent. For example, unity may be input as 1.0 in single precision, but the longer form 1.0D0 is required for double precision.
10. On long word machines, almost all matrix calculations are performed in single precision and, on short word machines, in double precision. It is recommended that DMIJ matrices also follow these conventions for a balance of efficiency and reliability. The recommended value for TOUT is 0, which instructs the program to inspect the system cell that measures the machine precision at run time and sets the precision of the matrix to the same value. TOUT = 0 allows the same DMIJI input to be used on any machine. If TOUT is contrary to the machine type specified, unreliable results may occur.
11. If any DMIJ entry is changed or added on restart then a complete reanalysis is performed. Therefore, DMIJ entry changes or additions are not recommended on restart.

DMIK**Direct Matrix Input at ks-Set of the Aerodynamic Mesh**

Defines direct input matrices related to physical (displacement) degrees-of-freedom (ks-set) of aerodynamic grid points. These include WKK, WTFAC and input forces associated with AEFORCE entries. The matrix is described by a single header entry and one or more column entries. A column entry is required for each column with nonzero elements.

Header Entry Format:

1	2	3	4	5	6	7	8	9	10
DMIK	NAME	"0"	IFO	TIN	TOUT	POLAR		NCOL	

Column Entry Format:

DMIK	NAME	GJ	CJ		G1	C1	A1	B1	
		G2	C2	A2	B2	-etc.-			

Example:

DMIK	ALPH1	0	9	2	0			1	
DMIK	ALPH1	1	1		1	1	1.0		
		2	1	1.0					

Descriptor	Meaning
NAME	Name of the matrix. See Remark 1. (One to eight alphanumeric characters, the first of which is alphabetic.)
IFO	Form of matrix being input. (Integer)
	1 Square
	9 or 2 Rectangular
	6 Symmetric
TIN	Type of matrix being input: (Integer)
	1 Real, single precision (One field is used per element)
	2 Real, double precision (One field is used per element)
	3 Complex, single precision (Two fields are used per element)
	4 Complex, double precision (Two fields are used per element)
TOUT	Type of matrix being created: (Integer)
	0 Set by precision system cell (Default)
	1 Real, single precision
	2 Real, double precision
	3 Complex, single precision

Descriptor	Meaning
4	Complex, double precision
POLAR	Input format of Ai, Bi. (Integer = blank or 0 indicates real, imaginary format. Integer > 0 indicated magnitude, phase format.)
NCOL	Number of columns in a rectangular matrix. Used only for IFO = 9. (Integer > 0)
GJ	Grid, scalar or extra point identification number for column index. (Integer > 0)
CJ	Component number for grid point GJ. (0 < Integer \leq 6; blank or zero if GJ is a scalar or extra point.)
Gi	Grid, scalar, or extra point identification number for row index. (Integer > 0)
Ci	Component number for Gi for a grid point. (0 < CJ < 6; blank or zero if Gi is a scalar or extra point.)
Ai, Bi	Real and imaginary (or amplitude and phase) parts of a matrix element. If the matrix is real (TIN = 1 or 2), then Bi must be blank. (Real)

Remarks:

1. Matrices defined on this entry are referenced in static aeroelastic analysis by reference on AEFORCE entries. In that paradigm, a single column is required. Also, DMIK may also be used for the WKK and WTFACT entries. Matrices defined on this entry are referenced in static aeroelastic analysis by reference on AEFORCE entries. In that paradigm, a single column is required. Also, DMIK may also be used for the WKK and WTFACT entries. If both DMI and DMIK are specified for WKK or WTFACT, the DMI entry will be used. DMI may NOT be used for AEFORCE.
2. The ks-set DOF's for each aerodynamic theory are limited to the six-DOF paradigm (3 translations and 3 rotations). However, particular DOF's are permanently SPC'd based on the theory's ability to support those degrees-of-freedom. Unlike the DMIG entry, DMIK data will be partitioned to the k-set, not reduced. No warnings are issued about truncated data. The k-set DOF's for each aerodynamic element/grid are highly method dependent. The following data define the known set, but the j-set definition is somewhat arbitrary in the general (external aerodynamics) paradigm.

Entry Type	COMP				
	1	2	3	5	6
CAERO1			X	X	
CAERO2-Y		X		X	
CAERO2-Z			X		X
CAERO2-ZY		X	X	X	X

3. The header entry containing IFO, TIN and TOUT is required. Each nonnull column is started with a GJ, CJ pair. The entries for each row of that column then follow. Only nonzero terms need be entered. The terms may be input in arbitrary order. A GJ, CJ pair may be entered more than once, but the input of an element of the matrix more than once results in a fatal message.

4. Field 3 of the header entry must contain an integer 0.
5. For symmetric matrices (IFO=6), a given off-diagonal element may be input either below or above the diagonal. While upper and lower triangle terms may be mixed, a fatal message will be issued if an element is input both above and below the diagonal.
6. The recommended format for rectangular matrices requires the use of NCOL and IFO = 9. The number of columns in the matrix is NCOL. (The number of rows in all DMIK matrices is always the ks-set size--the union of the k-set and the permanently SPC'd partition). The GJ term is used for the column index. The CJ term is ignored.
7. If NCOL is not used for rectangular matrices, two different conventions are available:
 - If IFO = 9, GJ and CJ will determine the sorted sequence, but will otherwise be ignored; a rectangular matrix will be generated with the columns submitted being in the 1 to N positions, where N is the number of logical entries submitted (not counting the header entry).
 - If IFO = 2, the number of columns of the rectangular matrix will be equal to the index of the highest numbered nonnull column (in internal sort). Trailing null columns of the js-size matrix will be truncated.
8. The matrix names must be unique among all DMIK.
9. TIN should be consistent with the number of decimal digits required to read the input data adequately. For a single precision specification on a short word machine, the input will be truncated after about eight decimal digits, even when more digits are present in a double field format. If more digits are needed, a double precision specification should be used instead. However, note that a double precision specification requires a "D" type exponent even for terms that do not need an exponent. For example, unity may be input as 1.0 in single precision, but the longer form 1.0D0 is required for double precision.
10. On long word machines, almost all matrix calculations are performed in single precision and, on short word machines, in double precision. It is recommended that DMIK matrices also follow these conventions for a balance of efficiency and reliability. The recommended value for TOUT is 0, which instructs the program to inspect the system cell that measures the machine precision at run time and sets the precision of the matrix to the same value. TOUT = 0 allows the same DMIK input to be used on any machine. If TOUT is contrary to the machine type specified, unreliable results may occur.
11. If any DMIK entry is changed or added on restart then a complete reanalysis is performed. Therefore, DMIK entry changes or additions are not recommended on restart.

DOPTPRM

Design Optimization Parameters

Overrides default values of parameters used in design optimization.

Format:

1	2	3	4	5	6	7	8	9	10
DOPTPRM	PARAM1	VAL1	PARAM2	VAL2	PARAM3	VAL3	PARAM4	VAL4	
	PARAM5	VAL5	-etc.-						

Example:

DOPTPRM	IPRINT	5	DESMAX	10					
---------	--------	---	--------	----	--	--	--	--	--

Descriptor	Meaning
PARAMi	Name of the design optimization parameter. Allowable names are given in Table 11 . (Character)
VALi	Value of the parameter. (Real or Integer, see Table 11 .)

Table 11 PARAMi Names and Descriptions

Name	Description, Type, and Default Value	
APRCOD	Approximation method to be used. (Integer 1, 2, or 3; Default = 2)	
	1	Direct Linearization. APRCOD = 1 is recommended for shape, topology and topography optimization problems.
	2	Mixed Method based on response type
	3	Convex Linearization
AUTOSE	Flag to request an AESO job. AUTOSE = 1 activates an AESO creation run. (Integer ≥ 0 ; Default = 0)	
CONV1	Relative criterion to detect convergence. If the relative change in objective between two optimization cycles is less than CONV1, then optimization is terminated. Topology optimization can be terminated with CONV1<5.00E-3 (Real > 0.0; Default = 0.001).	
CONV2	Absolute criterion to detect convergence. If the absolute change in objective between two optimization cycles is less than CONV2, then optimization is terminated. (Real > 0.0; Default = 1.0E-20)	
CONVDV	Relative convergence criterion on design variables. (Real > 0.0; Default = 0.001 for non-topology; Default = 0.0001 for topology optimization)	
CONVPR	Relative convergence criterion on properties. (Real > 0.0; Default = 0.001)	

Table 11 PARAMi Names and Descriptions (continued)

Name	Description, Type, and Default Value	
CT	Constraint tolerance. Constraint is considered active if current value is greater than CT. (Real < 0.0; Default = -0.03)	
CTMIN	Constraint is considered violated if current value is greater than CMIN. (Real > 0.0; Default = 0.003)	
DABOBJ	Maximum absolute change in objective between two consecutive iterations to indicate convergence at optimizer level. F0 is the initial objective function value. (Real > 0.0; Default = MAX[0.001*ABS(F0), 0.0001])	
DELB	Relative finite difference move parameter. (Real > 0.0; Default = 0.0001; 0.001 for fatigue responses)	
DELOBJ	Maximum relative change in objective between two consecutive iterations to indicate convergence at optimizer level. (Real > 0.0; Default = 0.001)	
DELP	Fractional change allowed in each property during any optimization design cycle. This provides constraints on property moves. (Real, 0.0<DELP<1.0, default=0.2)	
DELX	Fractional change allowed in each design variable during any optimization cycle. (Real > 0.0; Default = .5 for sizing/shape/topometry optimization; Default = 0.2 for topology and topography optimization)	
DELXESL	Fractional change allowed in each design variable during the ESLNRO loop. (Real > 0.0, Default = 0.5)	
DESMAX	Maximum number of design cycles (not including FSD cycle) to be performed. (Integer ≥ 0 ; Default = 5 for sizing/shape optimization; Default = 30 for topology, topography, and topometry optimization)	
DISCOD	Discrete Processing Method: (Integer 0, 1, 2, 3 or 4; Default = 1)	
	0	No Discrete Optimization
	1	Design of Experiments
	2	Conservative Discrete Design
	3	Round up to the nearest design variable
	4	Round off to the nearest design variable
DISBEG	Design cycle ID for discrete variable processing initiation. Discrete variable processing analysis is carried out for every design cycle after DISBEG. (Integer ≥ 0 ; Default = 0=the last design cycle)	
DPMAX	Maximum fraction of change on designed property (Default = 0.5), used by Trust Region Method.	
DPMIN	Minimum move limit imposed. (Real > 0.0; Default = 0.01)	
DRATIO	Threshold value that can be used to turn off an active AESO job. An AESO job is terminated if the ratio of the size of a design model to that of an analysis model is greater than DRATIO. (Real > 0.; Default = 0.1)	

Table 11 PARAMi Names and Descriptions (continued)

Name	Description, Type, and Default Value	
DSMXESL	Maximum number of design cycles applied to the ESLNRO loop. (Integer ≥ 0 ; Default = 20)	
DXMAX	Maximum fraction of change on design variable (Default = 1.0), used by Trust Region Method.	
DXMIN	Minimum design variable move limit. (Real > 0.0 ; Default = 0.05 for sizing/shape/topometry optimization; Default = 1.0E-5 for topology and topography optimization)	
ETA1 (η_1)	the cutting ratio 1 (Default = 0.01), used by Trust Region Method.	
ETA2 (η_2)	the cutting ratio 2 (Default = 0.25), used by Trust Region Method.	
ETA3 (η_3)	the cutting ratio 3 (Default = 0.7), used by Trust Region Method.	
FSDALP	Relaxation parameter applied in Fully Stressed Design. (Real, $0.0 < FSDALP \leq 1.0$; Default = 0.9)	
FSDMAX	Specifies the number of Fully Stressed Design Cycles that are to be performed. (Integer; Default = 0)	
GMAX	Maximum constraint violation allowed at the converged optimum. (Real > 0.0 ; Default = 0.005)	
GSCAL	Constraint normalization factor. See Remarks under the DSCREEN and DCONSTR entries. (Real > 0.0 ; Default = 0.001)	
IGMAX	If IGMAX = 0, only gradients of active and violated constraints are calculated. If IGMAX > 0 , up to IGMAX gradients are calculated including active, violated, and near active constraints. For many constraint problems, a reasonable value of IGMAX = 2*NDV (number of design variables). A smaller IGMAX has a faster performance of MSCADS. (Integer > 0 ; Default = 0)	
IPRINT	Print control during approximate optimization phase. Increasing values represent increasing levels of optimizer information. ($0 \leq \text{Integer} \leq 7$; Default = 0)	
	0	No output (Default)
	1	Internal optimization parameters, initial information, and results
	2	Same, plus objective function and design variables at each iterations
	3	Same, plus constraint values and identification of critical constraints
	4	Same, plus gradients
	5	Same, plus search direction
	6	Same, plus scaling factors and miscellaneous search information
	7	Same, plus one dimensional search information
ISCAL	Design variables scaling. (Integer ≥ -1 ; Default = 0) = -1 no scaling; ≥ 0 scaling	

Table 11 PARAMi Names and Descriptions (continued)

Name	Description, Type, and Default Value	
METHOD	Optimization Method: (Integer ≥ 0 ; Default = 0)	
	0	Automatic selection for a better performance based on number of design variables, number of constraints, number of active/violated constraints and computer memory.
	1	Modified Method of Feasible Directions for MSCADS.
	2	Sequential Linear Programming for MSCADS
	3	Sequential Quadratic Programming for MSCADS
	4	SUMT method for MSCADS
NASPR0	IJK	See Remark 3.
	First cycle analysis output control. (Integer 0 or 1)	
	0	Print analysis output of first cycle. (Default)
OBJMOD	1	Do NOT print analysis output of first cycle.
	Objective function modification. (Integer; Default = 0)	
	0	Objective function will not be modified.
OPTCOD	1	Objective function will be reset to 0.0. Subsequently, printed objective function value represents the change of objective function.
	OPTCOD. See Remark 2. (Character; Default = Blank)	
	Blank	Taken from system cell number 413
	“MSCADS”	MSCADS is used
	“IPOPT”	IPOPT is used
	“External Optimizer SCA Service Identifier”	
P1		
	Must be defined in the CONNECT Service statement (see <i>MSC Nastran User Defined Services User's Guide</i>)	
	Print control items specified for P2. (Integer ≥ 0 ; Default = 0) Initial results are always printed prior to the first approximate optimization. If an optimization task is performed, final results are always printed for the final analysis unless PARAM,SOFTEXIT,YES is specified. These two sets of print are not controllable.	
		n Print at every n-th design cycle.

Table 11 PARAMi Names and Descriptions (continued)

Name	Description, Type, and Default Value	
P2	Items to be printed according to P1: (Integer; Default = 1)	
0	No print.	
1	Print objective and design variables. (Default for sizing/shape optimization) Print objective. (Default for topology optimization) $P2 \geq 13$ Print design variables for topology, topography, and topometry optimization	
2	Print properties.	
4	Print constraints.	
8	Print responses.	
16	Print weight as a function of a material ID (note that this is not a design quantity so that only inputs to the approximate design are available).	
n	Sum of desired items. For example, P2 = 10 means print properties and responses.	
P2CALL	Maximum number of retained constraints of all categories to be printed per category. This single parameter can be used in place of the individual parameters P2CBL, P2CC, P2CDDV, P2CM, P2CP and P2CR. If any of these six parameters are non-zero, the P2CALL value is overridden for that constraint type. (Integer > 0 ; default is to print all retained constraints.)	
P2CBL	Maximum number of constraints on beam library dimensions to be printed. (Integer ≥ 0 ; default is to print all beam library constraints.)	
P2CC	Maximum number of constraints on connectivity properties to be printed. (Integer ≥ 0 ; default is to print all connectivity property constraints.)	
P2CDDV	Maximum number of constraints on dependent design variables to be printed. (Integer ≥ 0 ; default is to print all dependent design variable constraints.)	
P2CM	Maximum number of constraints on material properties to be printed. (Integer ≥ 0 ; default is to print all material property constraints.)	
P2CP	Maximum number of constraints on element properties to be printed. (Integer ≥ 0 ; default is to print all element property constraints.)	
P2CR	Maximum number of constraints on design responses to be printed. (Integer ≥ 0 ; default is to print all retained design response constraints.)	
P2RSET	ID of a SET1 Bulk Data entry to identify the set of retained responses (DRESP1, DRESP2 and/or DRESP3) to be printed. (Integer; Default is to print all responses associated with printed constraints. If P2CR is > 0 , the set associated P2RSET > 0 will be printed independent of the responses associated with the printed constraint. If P2CR > 0 and PR2SET = -1, all retained responses will be printed.)	

Table 11 PARAMi Names and Descriptions (continued)

Name	Description, Type, and Default Value	
PENAL	Penalty parameter used to transform an infeasible approximate optimization task to a feasible one. Setting this parameter to; e.g., 2.0 may improve optimizer performance when the starting design is infeasible. (Real; Default = 0.0)	
PLVIOL	Flag for handling of property limit violation. By default, the job will terminate with a user fatal message if the property derived from design model (DVPRELi, DVMRELi, DVCRELi) exceeds the property limits. Setting PLVIOL to a non-zero number will cause the program to issue a user warning message by ignoring the property limits violation and proceed with the analysis. (Integer; Default = 0)	
PTOL	Maximum tolerance on differences allowed between the property values on property entries and the property values calculated from the design variable values on the DESVAR entry (through DVPRELi relations). PTOL is provided to trap ill-posed design models. (The minimum tolerance may be specified on user parameter DPEPS. See Parameters, 783) (Real > 0.0; Default = 1.0E+35)	
STPSCL	Scaling factor for shape finite difference step sizes, to be applied to all shape design variables. (Real > 0.0; Default = 1.0)	
TCHECK	Topology Checkerboarding/minimum member size control option. (Integer ≥ -1)	
	1	Filtering algorithm
	2	Density constraint
	0	No control
	-1	Automatic selection of filtering or density constraint algorithm for a better performance. (Default)
TDMIN	Minimum diameter of members in topology optimization. This option is applied on 2 and 3D elements only. (Real > 0.0)	
TREGION	Flag to invoke Trust Region method.	
	0	Don't employ trust regions (Default)
	1	Turn Trust Region on
UPDFAC1	Updating factor 1 (Default = 2.0), used by Trust Region Method.	
UPDFAC2	Updating factor 2 (Default = 0.5), used by Trust Region Method.	

Remarks:

- Only one DOPTPRM entry is allowed in the Bulk Data Section. All defaults recommended.
- OPTCOD specifies which optimization code to be used in SOL 200 and METHOD specifies which optimization method to be used. The default is recommended.
- METHOD = IJK enables a user selectable optimization strategy as documented in Vanderplaats, G. N., ADS -- A Fortran Program for Automated Design Synthesis -- Version 1.10, NASA CR 177985, 1985. The default is recommended.

The I selects one of ten available strategy options:

0	None -- Go directly to the optimizer
1	Sequential unconstrained minimization using the exterior penalty function method
2	Sequential unconstrained minimization using the linear extended interior penalty function method
3	Sequential unconstrained minimization using the quadratic extended interior penalty function method
4	Sequential unconstrained minimization using the cubic extended interior penalty function method
5	Augmented Lagrange multiplier method
6	Sequential linear programming
7	Method of centers
8	Sequential quadratic programming
9	Sequential convex programming

The J selects one of five available optimizer options:

1	Fletcher-Reeves algorithm for unconstrained minimization
2	Davidon-Fletcher-Powell (DFP) variable metric method for unconstrained minimization
3	Broydon-Fletcher-Goldfarb-Shanno (BFGS) variable metric method for unconstrained minimization
4	Method of feasible directions for constrained minimization
5	Modified method of feasible directions for constrained minimization

And K selects one of eight available one-dimensional search strategies:

1	Find the minimum of an unconstrained function using the Golden Section method
2	Find the minimum of an unconstrained function using the Golden Section method followed by polynomial interpolation
3	Find the minimum of an unconstrained function by first finding bounds and then using the Golden Section method followed by polynomial interpolation
4	Find the minimum of an unconstrained function by polynomial interpolation/extrapolation without first finding bounds on the solution

5	Find the minimum of a constrained function using the Golden Section method
6	Find the minimum of a constrained function using the Golden Section method followed by polynomial interpolation
7	Find the minimum of a constrained function by first finding bounds and then using polynomial interpolation
8	Find the minimum of a constrained function by polynomial interpolation/extrapolation without first finding bounds on the solution

DPHASE**Dynamic Load Phase Lead**

Defines the phase lead term θ in the equation of the dynamic loading function.

Format:

1	2	3	4	5	6	7	8	9	10
DPHASE	SID	P1	C1	TH1	P2	C2	TH2		

Example:

DPHASE	4	21	6	2.1	8	6	7.2		
--------	---	----	---	-----	---	---	-----	--	--

Descriptor	Meaning
SID	Identification number of DPHASE entry. (Integer > 0)
Pi	Grid, extra, or scalar point identification number. (Integer > 0)
Ci	Component number. (Integers 1 through 6 for grid points; zero or blank for extra or scalar points)
THi	Phase lead θ in degrees. (Real)

Remarks:

1. One or two dynamic load phase lead terms may be defined on a single entry.
2. SID must be referenced on a RLOADi entry. Refer to the RLOAD1 or RLOAD2 entry for the formulas that define how the phase lead θ is used.
3. A DAREA, LSEQ or static load entry should be used to define a load at Pi and Ci.
4. In superelement analysis, DPHASE entries may only be applied to loads on points in the residual structure.
5. RC network solver does not support DPHASE for thermal analysis.

DRESP1**Design Sensitivity Response Quantities**

Defines a set of structural responses that is used in the design either as constraints or as an objective.

Format:

1	2	3	4	5	6	7	8	9	10
DRESP1	ID	LABEL	RTYPE	PTYPE	REGION	ATTA	ATTB	ATT1	
	ATT2	-etc.-							

Example:

DRESP1	1	DX1	STRESS	PROD	2	3		102	
	103								

DRESP1	10	BMWW	WMPID	PSHELL		10		10	
	20	30							

Descriptor	Meaning
ID	Unique entry identifier. (Integer > 0)
LABEL	User-defined label. (Character, no default)
RTYPE	Response type. See Table 12 . (Character)
PTYPE	Element flag (PTYPE = "ELEM") or property entry name. Used with element type responses (stress, strain, force, etc.) to identify the property type, since property entry IDs are not unique across property types. (Character: "ELEM", "PBAR", "PSHELL", etc.)
REGION	Region identifier for constraint screening. See Remark 10. for defaults. (Integer > 0)
ATTA, ATTB, ATTi	Response attributes. See Table 12 . (Integer > 0 or Real or blank)

Table 12 Design Sensitivity Response Attributes

Response Type (RTYPE)	Response Attributes		
	ATT A (Integer > 0)	ATT B (Integer > 0 or Real > 0.0)	ATT I (Integer > 0)
WEIGHT	Row Number ($1 \leq \text{ROW} \leq 6$) See Remark 24.	Column Number ($1 \leq \text{COL} \leq 6$)	SEIDi or All or blank. See Remark 12.
VOLUME	Blank	Blank	SEIDi or ALL or blank. See Remark 12.
FRMASS	Blank	Blank	Blank or Property ID (PID). See Remark 37.
COMP	Blank	Blank	Blank
See Remark 28.			
EIGN	Normal Modes Mode Number. See Remark 33.	Approximation Code. See Remark 19.	Blank
CEIG	Complex Eigenvalue Mode Number. (Integer > 0)	ALPHA or OMEGA (Default = ALPHA)	Blank
FREQ	Normal Modes Mode Number. See Remarks 18. and 33.	Approximation Code. See Remark 19.	Blank
LAMA	Buckling Mode Number	Approximation Code. See Remark 19.	Blank
DISP	Displacement Component	Blank or Mode Number	Grid ID
STRAIN	Strain Item Code	Blank or Mode Number	Property ID (PID) or Element ID (EID)
ESE	Strain Energy Item Code See Remark 21.	Blank or Mode Number	Property ID (PID) or Element ID (EID)
STRESS	Stress Item Code	Blank or Mode Number	Property ID (PID) or Element ID (EID)
FORCE	Force Item Code	Blank or Mode Number	Property ID (PID) or Element ID (EID)
FATIGUE (pseudo-static) See Remark 39.	Fatigue Item Code. See Remark 43.	ID of a FATIGUE case control. See Remark 40.	Property ID (PID) or Element ID (EID)

Table 12 Design Sensitivity Response Attributes

Response Type (RTYPE)	Response Attributes		
	ATTA (Integer > 0)	ATTB (Integer > 0 or Real > 0.0)	ATTI (Integer > 0)
FRFTG (frequency response - random vibration fatigue) See Remark 39.	Fatigue Item Code. See Remark 43.	ID of a FATIGUE case control. See Remark 40.	Property ID (PID) or Element ID (EID)
SPCFORCE	SPC Force Component	Blank	Grid ID
CSTRAIN See Remark 2.	Strain Item Code	LAMINA Number or GPLYIDi (Integer; Default = 1)	Property ID (PID) or Element ID (EID)
CSTRESS See Remark 2.	Stress Item Code	LAMINA Number or GPLYIDi (Integer; Default = 1)	Property ID (PID) or Element ID (EID)
CFailure See Remark 2.	Failure Indices Item Code	LAMINA Number or GPLYIDi (Integer; Default = 1)	Property ID (PID) or Element ID (EID)
CSTRAT See Remark 2.	Composite Strength Ratio Item Code	LAMINA Number or GPLYIDi (Integer; Default = 1)	Property ID (PID) or Element ID (EID)
TOTSE (Total Strain Energy)	Blank	Blank or Mode Number	SEIDi or All or blank. See Remark 12.
GPFORCE	GPFORCE Component Code (1-6; see Remark 25.)	Blank	Element ID
GPFORCP	Grid Point (see Remark 26.)	Blank	Grid ID connected to ATTA grid to specify orientation.
ABSTRESS	Arbitrary Beam Stress Item Code (see Remark 30.)	Blank	Property ID (PID) or Element ID (EID)
FRDISP	Displacement Component	Frequency Value. (Blank; Real ≥ 0.0 or Character) See Remarks 15. and 20.	Grid ID
PRES	Acoustic Pressure Component (= 1 or 7)	Frequency Value. (Blank; Real ≥ 0.0 or Character) See Remarks 15. and 20.	Grid ID

Table 12 Design Sensitivity Response Attributes

Response Type (RTYPE)	Response Attributes		
	ATT A (Integer > 0)	ATT B (Integer > 0 or Real > 0.0)	ATT I (Integer > 0)
FRVELO	Velocity Component	Frequency Value. (Blank; Real ≥ 0.0 or Character) See Remarks 15. and 20.	Grid ID
FRACCL	Acceleration Component	Frequency Value. (Blank; Real ≥ 0.0 or Character) See Remarks 15. and 20.	Grid ID
FRSPCF	SPC Force Component	Frequency Value. (Blank; Real ≥ 0.0 or Character) See Remarks 15. and 20.	Grid ID
FRSTRE	Stress Item Code	Frequency Value. (Blank; Real ≥ 0.0 or Character) See Remarks 15. and 20.	Property ID (PID) or Element ID (EID)
FRFORC	Force Item Code	Frequency Value. (Blank; Real ≥ 0.0 or Character) See Remarks 15. and 20.	Property ID (PID) or Element ID (EID)
PSDDISP	Displacement Component (see Remarks 27. and 31.)	Frequency Value. (Blank; Real ≥ 0.0 or Character). See Remarks 15. and 20.	Grid ID
PSDVELO	Velocity Component (see Remarks 27. and 31.)	Frequency Value (Blank; Real ≥ 0.0 or Character). See Remarks 15. and 20.	Grid ID
PSDACCL	Acceleration Component (see Remarks 27. and 31.)	Frequency Value. (Blank; Real ≥ 0.0 or Character). See Remarks 15. and 20.	Grid ID

Table 12 Design Sensitivity Response Attributes

Response Type (RTYPE)	Response Attributes		
	ATTA (Integer > 0)	ATTB (Integer > 0 or Real > 0.0)	ATTI (Integer > 0)
RMSDISP	Displacement Component (see Remark 31.)	RANDPS ID	Grid ID
RMSVELO	Velocity Component (see Remark 31.)	RANDPS ID	Grid ID
RMSACCL	Acceleration Component (see Remark 31.)	RANDPS ID	Grid ID
ACPWR (Acoustic Power radiated through a panel) See Remark 34.	Blank	Frequency value. (Blank for all forcing frequency; Real ≥ 0.0)	Blank
ACINTS (Acoustic Intensity)	Blank	Frequency value. (Blank for all forcing frequency; Real ≥ 0.0)	Grid ID of wetted surface.
AFPRES (Acoustic Pressure for AFPM) See Remark 35.	Acoustic Pressure Component (Integer = 1 or 7)	Frequency value (Blank for all forcing frequency; Real ≥ 0.0)	Grid ID of AFPMID.
AFINTS (Acoustic Intensity for AFPM) See Remark 35.	Component Code - 0 - normal to AFPM, 1 - x-dir 2 - y-dir 3 - z-dir	Frequency value. (Blank for all forcing frequency; Real ≥ 0.0)	Grid ID of AFPMID.
AFVELO Velocity for AFPM) See Remark 35.	Component Code - 11 - Real/Mag in x-dir 12 - Real/Mag in y-dir 13 - Real/Mag in z-dir 71 - Img/Ph in x-dir 72 - Img/Ph in y-dir 73 - Img/Ph in z-dir	Frequency value. (Blank for all forcing frequency; Real ≥ 0.0)	Grid ID of AFPMID.
AFPWR (Acoustic Power for AFPM) See Remark 35.	Blank	Frequency value. (Blank for all forcing frequency; Real ≥ 0.0)	Blank
ERP See Remarks 41. and 42.	ERP Item Code	Frequency Value. (Blank; Real ≥ 0.0 or Character) See Remarks 15. and 20.	Set3 ID or Blank

Table 12 Design Sensitivity Response Attributes

Response Type (RTYPE)	Response Attributes		
	ATTI (Integer > 0)	ATTB (Integer > 0 or Real > 0.0)	ATTI (Integer > 0)
TDISP	Displacement Component	Time Value. (Blank; Real; or Character) See Remarks 16. and 20.	Grid ID
TVELO	Velocity Component	Time Value. (Blank; Real, or Character) See Remarks 16. and 20.	Grid ID
TACCL	Acceleration Component	Time Value. (Blank; Real, or Character) See Remarks 16. and 20.	Grid ID
TSPCF	SPC Force Component	Time Value. (Blank; Real, or Character) See Remarks 16. and 20.	Grid ID
TSTRE	Stress Item Code	Time Value. (Blank; Real, or Character) See Remarks 16. and 20.	Property ID (PID) or Element ID (EID)
TFORC	Force Item Code	Time Value. (Blank; Real, or Character) See Remarks 16. and 20.	Property ID (PID) or Element ID (EID)
STMONP1 Structural MONPNT1	Component See Remark 36.	Blank	Blank
STMOND1 Structural MONDSP1	Component See Remark 36.	Blank	Blank
MONPNT3	Component See Remark 36.	Blank	Blank
AEMONP1 Aerodynamic MONPNT1	Component See Remark 36.	Blank	Blank