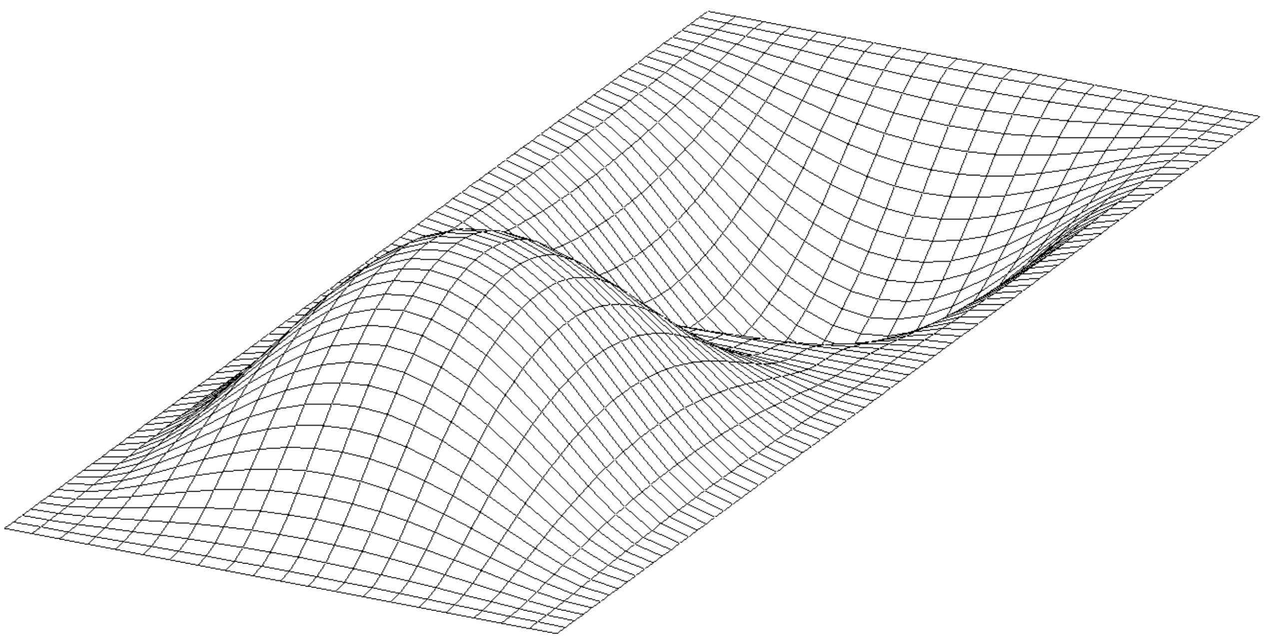
Autodesk Explicit 2021

Reference Manual

Version 2021



Printed and bound in the United States of America.

.

Contents

Section 1

AUTODESK EXPLICIT COMMAND LINE

Running Autodesk Explicit

Autodesk Explicit is run by executing the file *aexp_app.exe*. The syntax for this along with the optional command line arguments are shown below. The *aexp_app.exe* should have the following lines preceding it:

```
cscript /nologo
```

This tells the script to output the information to the console. The entire line would appear as follows:

```
cscript /nologo  “C:\Program Files\Autodesk\Inventor Nastran
       2021\aexp_app.exe” -i inputfile.nas -dc

aexp_app.exe” -i inputFile -dc
```

The command line arguments are defined as follows:

`[-i inputFile]`	(REQUIRED) The name of an input file for the application. The input file should be a NASTRAN input file. The file extension ".nas" or “.NAS” is required.
`[-dc]`	(OPTIONAL) Data check flag: If this flag appears, the application will perform a data check run only. In this case the application will run the mesh translator through the point where the explicit module has read all of the input file, computed the stable time increment for the model, and checked the contact conditions. The explicit module will then stop without any time integration.

(Continued)

Section 2

CASE CONTROL

The Case Control Section

The Case Control Section performs the following basic functions:

- Selects loads and constraints.
- Defines the contents of the Model Results Output File.
- Defines the output coordinate system for element and grid point results.
- Defines the subcase structure for the analysis.

Case Control Command Descriptions

Case Control commands may be abbreviated down to the first four characters provided the abbreviation is unique relative to all other commands. Each command is described as follows:

Description

A brief **Description** is given to state the function of the Case Control command.

Format

The command syntax is defined under **Format**. Listed options are further described under **Option**. The following conventions are used:

- Options in uppercase are keywords that must be specified as shown.
- Options in lowercase indicate that the user must provide a value.
- Parentheses ( ) must be included if an option requiring them is specified.
- Brackets [ ] indicate that specifying an option is not required.
- Braces { } indicate that specifying an option is required.
- If the command line is longer than 80 columns, then it may be continued to the next line with a comma. For example:

```
SET 12 = 15, 16, 17, 28, 39,
         100 THRU 556
```

Example

A typical example is given under **Example**.

Option, Definition, and Type

Each option is listed under **Option** and briefly discussed under **Definition**. The option’s type (e.g., Integer, Real, or Character) and allowable range are specified under **Type**. The default option is annotated with a ✓ symbol.

Remarks

Additional information about the command is given under **Remarks**.

$ Comment

Description: The $ symbol is used to add comments to the Model Input File.

Format:

$ followed by any characters out to column 80.

Example:

```
$ Nitrogen Tank Model Version 10.0, 17 Feb 2010
```

Remarks:

1. Comments are ignored by the program and may appear anywhere within the Model Input File.
2. Comments will not appear in either the sorted or unsorted echo of the Bulk Data.

ACCELERATION

Acceleration Output Request

Description: The ACCELERATION Case Control command requests acceleration vector output.

Format:

ACCELERATION

Example:

ACCELERATION

Remarks:

1. ACCELERATION results are output in the global coordinate system.
2. The translation components are in the same units of measure as the model. For example, if the length unit of the model is inches, and the time unit for the model is seconds, the acceleration will be inches per second squared. The rotation components are in radians per second squared.
3. ACCELERATION output is output for all grid points in the model.

APPLIEDLOADS — Applied Load Output Request

Description: The APPLIELOADS Case Control command requests applied load vector output.

Format:

APPLIEDLOADS

Example:

APPLIEDLOADS

Remarks:

1. APPLIEDLOADS results are output in the global coordinate system.
2. Kinematic boundary conditions, which are specified with SPCD and SPCR Bulk Data entries, generate reaction forces. These reaction forces are not included in the APPLIEDLOADS output. Reaction forces are output by use of the REACTIONS Case Control command.

BEGIN BULK

Case Control Delimiter

Description: The BEGIN BULK entry designates the end of the Case Control Section and the beginning of the Bulk Data Section.

Format:

BEGIN BULK

Remarks:

1. Only one occurrence of BEGIN BULK is allowed.
2. A BEGIN BULK entry and an associated ENDDATA entry are required even if there are no Bulk Data entries.

CONTACTFORCES

Contact Force Output Selection

Description: The CONTACTFORCES Case Control command requests contact force vector output.

Format:

CONTACTFORCES

Example:

CONTACTFORCES

Remarks:

1. The output from the CONTACTFORCES command is in the global coordinate.

CONTACTDELETE — Delete Contact Pairs

Description: The CONTACTDELETE Case Control command is used in conjunction with the CONTACTGENERATE command. The CONTACTGENERATE command automatically generates all the possible contact surface pairs in a model. Use the CONTACTDELETE command to manually remove pairs of contact surfaces that you know will never come into contact. The description of the CONTACTGENERATE command gives the convention for numbering generated contact surfaces. You can eliminate the case of self-contact for a surface by listing the surface identifier twice on the CONTACTDELETE command. See the following examples.

Format:

CONTACTDELETE, csid1, csid2

Examples:

CONTACTDEL, 88,77
CONTACTDEL, 5,5

CONTACTGENERATE — Automated Surface Contact Generation

Description: The CONTACTGENERATE Case Control command automatically generates contact surfaces for all “parts” of the model and an automatic pairing of all surfaces as potential contact pairs. This contact surface generation is referred to as Automated Surface Contact Generation (ASCG). (We define the term “parts” in the following discussion.)

The numbering for generated contact surfaces is determined by property identification values (PID’s that appear on property entries). Suppose, for example, we have a set of elements that reference a property entry with a PID of 25. The ASCG will examine all of the elements that reference this property entry and determine which element faces associated with this set of elements are exterior faces – faces that can come into contact with other exterior faces on different element sets. The set of exterior faces from the elements that reference the property entry with a PID of 25 constitutes a contact surface with an identifier of 25. The identifier of 25 is used as the CSID (contact surface identifier) of the generated contact surface. The generated contact surface is a list of faces (segments in BSSEG terminology) that are exterior faces for all of the elements that reference a property entry with a PID of 25. The generated contact surface is treated like a user defined BSSEG entry and is equivalent to an actual BSSEG entry. The generated contact surface does not appear as an actual BSSEG entry in the Bulk Data.

A common occurrence is to have two or more blocks of solid elements come into contact. The ASCG will automatically “skin” any block of solid elements to obtain the contact surface for the block of elements. If a block of solid elements references a property entry with a PID of 37, the skinning of the block will produce a generated contact surface with a CSID (contact surface identifier) of 37.

A set of blocks that references the same property entry will be referred to as a “part”. Suppose a set of hexahedral elements defined by CHEXA entries all reference a PSOLID entry with a PID of 115. We will have a set of elements that all have the same topology and all reference a single property entry. (A part will not be a mixture of elements such as solid elements plus shell elements.) The generated contact surface associated with this part will have a CSID of 115.

The ASCG automatically generates contact surfaces for all parts of the model. Every part of the model that has a set of exterior faces and is associated with a property entry, and hence a unique PID, will generate a contact surface associated with the PID. The generated contact has a CSID that can be traced back to the PID on some property entry. You may refer to all the generated contact surfaces in the same manner as any contact surface defined with a BSSEG entry just by referencing the CSID of the generated surface

The matrix of all parts of the body that can contact each other is full. That is, if there are N unique parts in the model, then there is an N x N matrix that identifies all the contact pairs. This includes the diagonal of the matrix, which means that parts can contact themselves. The CONTACTDELETE command can be used to remove contact pairs from the matrix that you know in advance will not contact one another.

Format:

CONTACTGENERATE, *ptype*, *self, sfact, mu, maxad*, *capturetol*

Example:

CONTACTGENERATE, 2, OFF, 0.1

(Continued)

Option	Definition	Type	Default
ptype	Penetration type: 2 = Symmetric general contact 4 = Symmetric welded contact	Integer	2
self	Self contact flag: This flag indicates whether or not to allow surfaces to contact themselves (i.e., self contact).	Character, either ON or OFF	ON
sfact	Stiffness scaling factor: This factor is used to scale the penalty values. Penalty values are forces that are calculated internally to enforce contact. See Remarks 2 and 5.	$0.0 < \text{Real} \leq 1.0$	0.2
mu	Coefficient of static friction.	Real $\geq$ 0.0	0.0
maxad	Maximum activation parameter: See Remarks 3 and 4.	Real $\geq$ 0.0	0.5
capturetol	Capture tolerance parameter for tied contact pairs: This value is used for tied contact. Regions of opposing contact surfaces within a certain tolerance are tied together. See Remarks 4 and 5.	Real $\geq$ 0.0	0.05

Remarks:

1. The contact identification numbers are generated for each surface using PID's on the property entries. See the above description of CONTACTGENERATE.
2. The contact enforcement algorithm uses a penalty method approach. The penalty method automatically calculates a set of forces to eliminate overlap of any contact surfaces. The factor *sfact* may be used to scale the forces that are generated from the penalty method.
3. Values of *sfact* greater than 0.5 are not recommended; a value of *sfact* greater than 0.5 can lead to numerical instability.
4. The parameter *maxad* may be used to prevent unnecessary generation of contact. Contact is only generated if the distance from any contact surface primary node to the potential secondary node is less than $MAXAD * L_c$,, where L_c is a characteristic segment length determined from the faces on the contact surface.
5. The stiffness parameter, friction coefficient, penetration type, maximum activation distance, and capture tolerance specified on this line will be applied to all surface pairs generated by this command. Individual surface pair parameters may be re-defined using the BSCONP entry.
6. The parameter *capturetol* is used for tied contact. Regions of opposing contact surfaces that are within a distance $CONTACTOL * L_c$, where L_c is a characteristic segment length determined from the faces on the contact surfaces, are tied together.

DELETION — Element Deletion Criteria

Description: The DELETION Case Control command defines element deletion criteria for parts of the model. This command allows you to identify parts of the model that can be deleted due to excess strain, excess stress, negative element volume, etc. The DELETION command references the property identification value, PID, which appears on a property entry. Any elements referencing this PID are subject to deletion based on the criterion specified with the associated DELETION specification.

Format:

DELETION, *pid, type, val, numdel*

Example:

DELETION, 88, EFFSTRAIN, .01,10

Option	Definition		Type	Default
pid	Property identification: The parameter *pid* references a property identification value, PID, which appears on a property entry.		Integer > 0	Required
type	Deletion quantity: The choices for this parameter are:		Character	Required
	NEGVOLUME	Delete element if volume becomes negative.		
	EFFSTRAIN	Delete element if von Mises strain exceeds VALUE.		
	EQPS	Delete element if equivalent plastic strain exceeds VALUE.		
	PPFA	Delete element if progressive ply failure occurs in composite analysis.		
	PRINSTRAIN	Delete element if max principal strain exceeds VALUE.		
	BRITTLE	Delete element if brittle materials have two or more cracks form.		
	THINNING	Delete element if the ratio of current shell thickness to original shell thickness is less than VALUE.		
val	Deletion value. Deletion occurs if the criterion exceeds this value. Value is ignored for TYPE = NEGVOLUME, BRITTLE, and PPFA..		Real ≥ 0.0	0.0
numdel	Deletion interval. Once the element is deleted, the internal forces for the element are decremented to zero over *numdel* increments.		Integer > 0	10

DELSTOP — Stop Analysis if Based Upon Element Deletion

Description: The DELSTOP Case Control command defines limits based upon element deletion for which the analysis will stop when the limits are reached. This command allows you to identify parts of the model that you want to monitor for failure and stop the analysis when the failure limits are reached.. The DELSTOP command references the property identification value, PID, which appears on a property entry.

Format:

DELSTOP, *pid, val, type*

Example:

DELSTOP, 22,10,PERCENT

Option	Definition	Type	Default
pid	Property identification: The parameter *pid* references a property identification value, PID, which appears on a property entry.	Integer > 0	Required
val	Specifies the number of elements to required to be deleted at which time the analysis will terminate. A value of zero indicates that the analysis will terminate when the first element in the part is deleted.	Real ≥ 0.0	0.0
type	Specifies whether the number of elements is given as an absolute number or a percentage of all elements in the part. Allowable type specifications are PERCENT and NUMELEM.	Character	PERCENT

DISPLACEMENT — Displacement Output Request

Description: The DISPLACEMENT Case Control command requests displacement vector output.

Format:

DISPLACEMENT

Example:

DISPLACEMENT

Remarks:

1. DISPLACEMENT results are output in the global coordinate system.
2. The translation components are in the same units of measure as the model. For example, if the length unit of the model is inches, displacement will be inches. The rotation components are in radians.
3. Displacements are written for all grid points in the model.

DLOAD — Dynamic Load Set Selection

Description: The DLOAD Case Control command selects a dynamic load to be applied.

Format:

DLOAD = *n*

Example:

DLOAD = 10

Option	Definition	Type
n	Set identification. The parameter *n* references a set identification, SID, on a DLOAD or TLOAD1 Bulk Data entry.	Integer > 0

ELSTRAIN — Element Strain Output Request

Description: The ELSTRAIN Case Control command requests element strain output.

Format:

ELSTRAIN

Example:

ELSTRAIN

Remarks:

1. ELSTRAIN is an alternate form and is identical to STRAIN.
2. Strains are written at a single point for each element for solid elements. The top and bottom strain values are written for shell elements. Strains are written for a single point for each ply for composite shell elements.
3. This command will output all tensor strain components and the von Mises strain for all elements.
4. Equivalent plastic strain is automatically written for any element that has a material model with plasticity.
5. The von Mises strains are defined as follows:

For solid elements, the von Mises strain is:

$$\varepsilon_v = \left[\frac{2}{9}\left\{\left(\varepsilon_x - \varepsilon_y\right)^2 + \left(\varepsilon_y - \varepsilon_z\right)^2 + \left(\varepsilon_z - \varepsilon_x\right)^2\right\} + \frac{1}{3}\left(\gamma_{xy}^2 + \gamma_{yz}^2 + \gamma_{zx}^2\right)\right]^{\frac{1}{2}}$$

Shell elements are defined by a state of plane stress. The von Mises strain for shell elements is:

$$\varepsilon_v = \left[\frac{4}{9}\left(\varepsilon_x^2 + \varepsilon_y^2 - \varepsilon_x\varepsilon_y\right) + \frac{1}{3}\gamma_{xy}^2\right]^{\frac{1}{2}}$$

ELSTRESS — Element Stress Output Request

Description: The ELSTRESS Case Control command requests element stress output.

Format:

ELSTRESS

Example:

ELSTRESS

Remarks:

1. ELSTRESS is an alternate form and is identical to STRESS.
2. Stresses are written at a single point for each element for solid elements. The top and bottom stress values are written for shell elements. Strains are written for a single point for each ply for composite shell elements.
3. This command will output all stress tensor components, the von Mises stress, and pressure for all elements.
4. The invariant quantities, von Mises stress and pressure, are defined as follows:

For solid elements, the mean pressure is defined as:

$$p_o = -\frac{1}{3}\left(\sigma_x + \sigma_y + \sigma_z\right)$$

For solid elements the von Mises equivalent stress is defined as:

$$\tau_v = \left(\frac{3}{\sqrt{2}}\right)\tau_o$$

Shell elements are defined by a state of plane stress. The mean pressure is:

$$p_o = -\frac{1}{3}\left(\sigma_x + \sigma_y\right)$$

The von Mises stress for shell elements is:

$$\tau_v = \left[\sigma_x^2 + \sigma_y^2 - \sigma_x\sigma_y + 3\tau_{xy}^2\right]^{\frac{1}{2}}$$

IC — Transient Initial Condition Set Selection

Description: The IC Case Control command selects the initial conditions.

Format:

IC = *n*

Example:

IC = 15

Option	Definition	Type
n	Set identification. The parameter *n* references a set identification value, SID, on a TIC, INITVEL or ROTVEL Bulk Data entry.	Integer > 0

Remarks:

1. The above initial condition entries will not be used (no initial conditions) unless selected in the Case Control Section.

LOAD — External Static Load Set Selection

Description: The LOAD Case Control command selects the external load set to be applied to the model.

Format:

LOAD = *n*

Example:

LOAD = 15

Option	Definition	Type
n	Set identification. The parameter *n* references a set identification value, SID, on a FORCE, FORCE1, GRAV, MOMENT, MOMENT1, LOAD, PLOAD1, PLOAD2, PLOAD4, SPCD, or SPCDR entry in the Bulk Data Section. The SID referenced on this entry must appear on at least one of the above Bulk Data entries.	Integer $>$ 0

Remarks:

1. The above load entries will not be used unless selected in the Case Control Section.
2. The total load applied will be the sum of external (LOAD command) and constrained displacement (SPC command), loads should have unique set identification numbers.

LOADSET — Load Set Selection for Use in Dynamics

Description: The LOADSET Case Control command selects a sequence of load sets which can be referenced by dynamic load commands.

Format:

LOADSET = *n*

Example:

LOADSET = 100

Option	Definition	Type
n	Set identification. The parameter *n* references a set identification value, SID, which appears on at least one LSEQ Bulk Data Entry.	Integer $>$ 0

Remarks:

1. The number of load vectors created is the number of unique DAREA fields defined on all LSEQ Bulk Data entries.

PARAM — Parameter Specification

Description: The PARAM Case Control command specifies values for parameters to be used at certain places in the control sequence.

Format:

PARAM, *n, value*

Example:

PARAM, NLCOMPPLYFAIL,ON

Option	Definition	Type	Default
n	Parameter name. The parameter name is one to 16 alphanumeric characters, the first of which is alphabetic.	Character	Required
value	Parameter value. The parameter value is based on parameter type.	Character, real, or integer	Required

Remarks:

1. Parameters with names that are less than or equal to 8 characters can appear anywhere in the model input file prior to ENDDATA. Parameters with names greater than 8 characters must be specified in the Case Control Section.
2. For a list and detailed description of each parameter, see Section 4, *Parameters*.

PLASTICSTRAIN — Element Equivalent Plastic Strain Output Request

Description: The PLASTICSTRAIN Case Control command requests element equivalent plastic strain output. Any elements that reference a material definition with plasticity will write values of equivalent plastic strain.

Format:

PLASTICSTRAIN

Example:

PLASTICSTRAIN

REACTIONS — Reactions Output Request

Description: The REACTIONS Case Control command requests reactions vector output.

Format:

REACTIONS

Example:

REACTIONS

Remarks:

1. Indirect loads generated via the SPC, SPC1, SPCR, SPCD and SPCRD Bulk Data entries are output if the REACTIONS command is used.
2. Output for the reactions is in the global coordinate system.

RESTARTREAD — Restart File Reading Definition

Description: The RESTARTREAD Case Control command defines a restart file to be read for restart.

Formats:

RESTARTREAD [*d*:][*path*]*filename*[.*ext*]

Example:

RESTARTREAD ‘Bolt.rst.xml’

Option	Definition	Type	Default
d:	Device	Character	
path	Pathname for restart file	Character	
filename	Restart file name	Character	Required

Remarks:

1. Only one RESTARTREAD command may be used.
2. Maximum file specification length is 26 characters.
3. Quotation marks on the file specification are optional and are only required when the file name includes embedded blanks.
4. File names are case sensitive.
5. A file extension, *ext*, of “.rst.xml” is required on the filename.

SET — Set Definition

Description: The SET Control Case command defines a set of identification numbers (grid point or element) for processing and XY output requests.

Formats:

$$\text{SET}\, n = \{\text{i1}[\text{, i2, i3 THRU i4}]\}$$

$$\text{SET}\, n = \text{ALL}$$

Examples:

SET 15 = 7

SET 55 = 1 THRU 200000

SET 22 = 1, 5, 7, 8, 9, 15 THRU 66, 77, 79, 106 THRU 400,
544, 625, 1005 THRU 2067, 3005, 4020

SET 12 = 1.0, 2.0, 3.0, 4.0

SET 35 = 1.07-2, 8.05, 16.145, 2.456+2

Option	Definition	Type	Default
n	Set identification number.	Integer > 0	Required
i1, i2, etc.	Identification numbers. Identification numbers that do not exist are ignored.	Integer > 0	
i3 THRU i4	Identification number range (i3 $<$ i4). Identification numbers that do not exist are ignored.	Integer > 0	
ALL	All identification numbers are included.	Character	

Remarks:

1. Multiple SET commands with the same set identification number are allowed and will be treated as one set.
2. A comma at the end of the command signifies a continuation.
3. A THRU symbol may not be used for a continuation without the ending identification number.

SOLUTION — Solution Sequence

Description: The SOLUTION Case Control command selects the type of solution.

Format:

SOLUTION = *type*

Example:

SOLUTION = NLTRAN

Alternate Format and Example:

SOLUTION = 129

Option	Definition	Type
type	Type of solution sequence. Available solution types depend on the license purchased.	Character

Remarks:

1. The following table gives the solution number corresponding to each solution type. Either one may be used.

Solution Character Variable	Solution Number
NONLINEAR TRANSIENT RESPONSE	129
NLTRAN	129

SPC — Single-Point Constraint Set Selection

Description: The SPC Case Control command selects the single-point constraint set to be applied to the model.

Format:

SPC = *n*

Example:

SPC = 10

Option	Definition	Type
n	Set identification. The parameter *n* references the set identification value, SID, of a single-point constraint set. There must be a SPC or SPCADD Bulk Data entry with a SID corresponding to the value specified for *n* on the SPC Case Control command.	Integer > 0

Remarks:

1. SPC and SPCADD Bulk Data entries will not be used unless selected in Case Control.
2. SPCD and SPCR entries cannot be referenced with this command.

STRAIN — Element Strain Output Request

Description: The STRAIN Case Control command requests element strain output.

Format:

STRAIN

Example:

STRAIN

Remarks:

1. ELSTRAIN is an alternate form and is identical to STRAIN.
2. Strains are written at a single point for each element for solid elements. The top and bottom strain values are written for shell elements. Strains are written for a single point for each ply for composite shell elements.
3. This command will output all tensor strain components and the von Mises strain for all elements.
4. Equivalent plastic strain is automatically written for any element that has a material model with plasticity.
5. The von Mises strains are defined as follows:

For solid elements, the von Mises strain is:

$$\varepsilon_v = \left[\frac{2}{9}\left\{ \left(\varepsilon_x - \varepsilon_y\right)^2 + \left(\varepsilon_y - \varepsilon_z\right)^2 + \left(\varepsilon_z - \varepsilon_x\right)^2 \right\} + \frac{1}{3}\left(\gamma_{xy}^2 + \gamma_{yz}^2 + \gamma_{zx}^2\right) \right]^{\frac{1}{2}}$$

Shell elements are defined by a state of plane stress. The von Mises strain for shell elements is:

$$\varepsilon_v = \left[\frac{4}{9}\left(\varepsilon_x^2 + \varepsilon_y^2 - \varepsilon_x\varepsilon_y\right) + \frac{1}{3}\gamma_{xy}^2 \right]^{\frac{1}{2}}$$

STRESS — Element Stress Output Request

Description: The STRESS Case Control command requests element stress output.

Format:

STRESS

Example:

STRESS

Remarks:

1. ELSTRESS is an alternate form and is identical to STRESS.
2. Stresses are written at a single point for each element for solid elements. The top and bottom stress values are written for shell elements. Strains are written for a single point for each ply for composite shell elements.
3. This command will output all stress tensor components, the von Mises stress, and pressure for all elements.
4. The invariant quantities, von Mises stress and pressure, are defined as follows:

   For solid elements, the mean pressure is defined as:

   $$p_o = -\frac{1}{3}\left(\sigma_x + \sigma_y + \sigma_z\right)$$

   For solid elements, the von Mises equivalent stress is defined as:

   $$\tau_v = \left(\frac{3}{\sqrt{2}}\right)\tau_o$$

   Shell elements are defined by a state of plane stress. The mean pressure is:

   $$p_o = -\frac{1}{3}\left(\sigma_x + \sigma_y\right)$$

   The von Mises stress for shell elements is:

   $$\tau_v = \left[\sigma_x^2 + \sigma_y^2 - \sigma_x\sigma_y + 3\tau_{xy}^2\right]^{\frac{1}{2}}$$

TSTEPNL — Transient Time Integration Specification

Description: The TSTEPNL Case Control command selects integration and output time steps for nonlinear transient response problems.

Format:

TSTEPNL = *n*

Example:

TSTEPNL = 45

Option	Definition	Type
n	Set identification. The parameter *n* references the identification number, ID, of a TSTEPNL Bulk Data entry.	Integer $>$ 0

Remarks:

1. A TSTEPNL entry must be selected to perform nonlinear transient response analysis.

VELOCITY — Velocity Output Request

Description: The VELOCITY Case Control command requests velocity vector output.

Format:

VELOCITY

Example:

VELOCITY

Remarks:

1. VELOCITY results are output in the global coordinate.
2. The translation components are in the same units of measure as the model. For example, if the length unit of the model is inches, and the time unit for the model is seconds, the velocity will be inches per second. The rotation components are in radians per second.
3. VELOCITY output is written for all grid points in the model.

XYPLOT — Generate X-Y Plots for a Set of Grid Points or Elements

Description: The XYPLOT Case Control command requests the generation of results for x-y plots for a set of grid points or elements.

Format:

XYPLOT, *setID, stype, variable, component, reduction, localsystem*

Example:

```
XYPLOT,,GLOBAL,ENERGY BALANCE
XYPLOT,15,GRID,DISPL
XYPLOT,88,GRID,REACTION,SUM
XYPLOT,16,ELEM,STRESS
```

Option	Definition	Type	Default
setID	Set identification. For *stype* equal to GRID or GLOBAL, the set identification, *setID*, references a set created using the SET entry. For *stype* equal to RIGID, the *setID* references a Part ID for a rigid body. A set identification is not required for *stype* equal to GLOBAL	Integer > 0	Required. See Remark 1
stype	Output set identification type. The *stype* parameter is one of the following character variables: GRID, RIGID, ELEM, or GLOBAL.	Character	Required.
variable	Output variable. For *stype* equal to GRID or RIGID, the variable options are: 1. Spatial selections: ACCEL, VEL, DISPL, REACTION FORCE, APPLIED LOAD, or CONTACT FORCE 2. Rotational selections: ANG ACCEL, ANG VEL, ROTATION, REACTION MOMENT, or APLLIED MOMENT For *stype* equal to ELEM, the variable options are: STRESS, STRAIN, STRESS RESULTANT, MOMENT RESULTANT, or CURVATURE For *stype* equal to GLOBAL, the variable options are: ENERGY BALANCE or TIME STEP	Character	Required. See Remarks 2 and 3
component	Component(s) of variable. For *stype* equal to GRID or RIGID, the following component specifications are permissible: X, Y, Z, VECMAG, or ALL For *stype* equal to ELEM, the following component specifications are permissible: XX, YY, ZZ, XY, YZ, ZX, MISES, or ALL For *stype* equals GLOBAL, *component* is not used.	Character	Required.

(Continued)

Option	Definition	Type	Default
reduction	Reduction of output quantities. For *stype* equal to GRID, the following reductions are possible: NONE Write out history for all grid points in the set. AVG Write out only one history for the average of all the values in the set. SUM Write out only one history for summation of all the values in the set. The reduction parameter is not used for the case of *stype* equal to RIGID. For *stype* equal to ELEM, the following reductions are possible: NONE Write out history for all elements in the set. AVG Write out only one history for the average of all the values in the set. The reduction parameter is not used for the case of *stype* equal to GLOBAL.	Character	NONE
localsystem	Local coordinate system for output of GRID quantities. If the GRID option is used, specifying the CID for a local coordinate system defined by a CORDix entry will result in the output of the GRID nodal quantities in the local coordinate system.	Integer > 0 or blank	0

Remarks:

1. The *setID* can be zero or a negative value for GLOBAL data.
2. The variable strings can be truncated to a minimum number of characters to be unique. For example, ACCEL can be truncated to AC, VEL can be truncated to V.
3. The variable names for GRID, ELEM and GLOBAL are in separate name spaces.

BULK DATA

The Bulk Data Section

The Bulk Data Section contains entries that define the model. This consists of model geometry, element connectivity, element and material properties, constraints, and loads. Certain entries, such as loads and constraints, are not active unless selected by an appropriate Case Control command.

Bulk Data Entry Descriptions

Each Bulk Data entry is described using the following format:

Description

A **Description** is given which states the function of the Bulk Data entry.

Format

The entry syntax is defined under **Format**. The first field gives the entry name. The following fields are referenced under **Field** and **Definition**. Light shaded fields are optional. Dark shaded fields must be left blank. If field 10 is dark shaded, then no continuation entries are permitted.

Example

A typical example is given under **Example**.

Field, Definition, Type, and Default

Each of the fields 2 through 9 that are named under **Format** is briefly described under **Definition**. The field's type (e.g., Integer, Real, or Character) and allowable range are specified under **Type**. If the field has a default, then it will be given under **Default**. If user input is required, then "Required" will be specified.

Remarks

Additional information about the entry is given under **Remarks**.

$ Comment

Description: The $ symbol is used to add comments to the Model Input File.

Format:

$ followed by any characters out to column 80.

Example:

$ NITROGEN TANK PROPERTIES

Remarks:

1. Comments are ignored by the program and may appear anywhere within the Model Input File.
2. Comments will not appear in either the sorted or unsorted echo of the Bulk Data or in the Bulk Data File.

BSBOUNDS — Define Bounding Box for Contact

Description: The BSBOUNDS entry defines a bounding box in three-dimensional space outside of which contact will not occur. Use of this entry can dramatically reduce the computation times in certain classes of problems.

The BSBOUNDS entry allows you to define a “contact process zone” for all contact surfaces in the model. Any faces of the contact surfaces that are outside these bounds will automatically be rejected for consideration in the contact enforcement.

Consider a metal rolling problem in which a flat work piece of sheet metal is fed into a set of rollers to form some complex shape. All the contact in the problem will occur in a process zone between the rollers. Since we will feed the entire sheet into the rollers and the rollers may perform multiple full revolutions, the contact surfaces in the problem must include the entire perimeter of the rollers and all the faces of the sheet metal (both sides of shell elements for two-sided contact). At any point in the analysis, only a small fraction of the faces are in contact. The faces in contact are the faces in the process zone. The BSBOUNDS entry allows us to drastically reduce the required contact search and tracking computations because we can simply reject any face outside the bounds as a possible contact candidate.

If the bounding box has zero or negative volume, an error will be issued.

Format:

1	2	3	4	5	6	7	8	9	10
BSBOUNDS	xmin	ymin	zmin	xmax	ymax	zmax			

Example:

BSBOUNDS	-10	-10	0	10	20	20			

Field	Definition	Type	Default
xmin, ymin, zmin	Lower bounds definition of the bounding box.	Real	0.0
xmax, ymax, zmax	Upper bounds definition of the bounding box.	Real	0.0

BSCONP — Surface Contact Parameters

Description: The BSCONP entry defines the parameters for a surface contact region.

Format:

1	2	3	4	5	6	7	8	9	10
BSCONP	CRID	SECON DARY	PRIMAR Y	SFACT	CAPTOL	MU	PTYPE	MAXAD	

Example:

BSCONP	11	2	5			0.2	2		

Field	Definition	Type	Default
CRID	Contact region identification number.	Integer > 0	Required
SECONDARY	Secondary region identification number. This is the CSID for a BSSEG entry or a generated contact surface.	Integer > 0	Required
PRIMARY	Primary region identification number. This is the CSID for a BSSET entry or a generated contact surface.	Integer > 0	Required
SFACT	Stiffness scaling factor. The stiffness scaling factor is used to scale the penalty values. The penalty values are determined automatically. See Remark 4.	0. < Real ≤ 1.0	0.2
CAPTOL	Capture tolerance parameter for tied contact pairs. This value is used for tied contact. Regions of opposing contact surfaces within a distance controlled by this parameter are tied together. See Remark 6.	Real ≥ 0.0	0.05
MU	Coefficient of static friction.	Real ≥ 0.0	0.0
PTYPE	Penetration type: 1 = Unsymmetric general contact (secondary penetration only) 2 = Symmetric general contact 3 = Unsymmetric welded contact 4 = Symmetric welded contact	1 ≤ Integer ≤ 4	1
MAXAD	Maximum activation parameter: See Remark 5.	Real ≥ 0.0	0.5

Remarks:

1. The contact region identification number, CRID, must be unique with respect to all other BSCONP identification numbers.
2. The SECONDARY field defines the secondary surface by referencing a user defined BSSEG Bulk Data entry or a generated contact surface from a CONTACTGENERATE command. The generated contact surface from a CONTACTGENERATE command uses a CSID that references the PID on a property entry. See the discussion of the CONTACTGENERATE command.

(Continued)

3. The PRIMARY field defines the primary surface by referencing a user defined BSSEG Bulk Data entry or generated contact surface from a CONTACTGENERATE command. The generated contact surface from a CONTACTGENERATE command uses a CSID that references the PID on a property entry. See the discussion of the CONTACTGENERATE command.
4. SFACT may be used to scale the penalty values that are computed automatically. Values of SFACT greater than 0.5 are not recommended as they can lead to numerical instability.
5. MAXAD may be used to prevent unnecessary generation of contact. Contact is only generated if the distance from any contact surface primary node to the potential secondary node is less than $MAXAD * L_c$, where L_c is a characteristic segment length determined from the faces on the contact surface.
6. The value of CAPTOL is used to determine the extent of tied contact during initialization. Regions of opposing contact surfaces that are within a distance $CONTACTOL * L_c$, where L_c is a characteristic segment length determined from the faces on the contact surfaces, are tied together.

BSCREATE — Boundary Surface Creation from a Part ID

Description: The BSCREATE entry defines a surface which consists of only the exterior faces (segments) of a part in the mesh. Autodesk Explicit automatically skins the part and determines all of the exterior faces. If the surface is defined by shell elements, the surface is two-sided. See the discussion of the CONTACTGENERATE command.

Format:

1	2	3	4	5	6	7	8	9	10
BSCREATE	PID								

Example:

BSCREATE	2								

Field	Definition	Type	Default
PID	The part identification, PID, on the BSCREATE references the PID on a property entry. The property entry defines the list of elements from which the exterior faces are generated for a contact surface. This list of faces (segments in BSSEG terminology) is referenced as a generated contact surface with a CSID derived from the PID on the property entry. See the discussion of the CONTACTGENERATE command.	Integer > 0	Required

BSNSET — Boundary Surface Creation as a Cloud of Nodes

Description: The BSNET entry defines a cloud of nodes which will interact with contact surfaces. The nodes will not penetrate a contact surface defined by faces (segments in BSSEG terminology). The set of nodes are treated as secondary quantities. The set of nodes is paired with contact surfaces using a BSCONP entry.

Format:

1	2	3	4	5	6	7	8	9	10
BSNSET	CSID	TYPE							

Example 1: Create a set of contact nodes from a node set defined by a SET entry.

BSNSET	17	GRID							

Example 2: Create a set of contact nodes from all the unique nodes in an element set defined by a SET entry.

BSNSET	88	ELEM							

Example 3: Create a set of contact nodes from all the unique nodes in a part defined by a property ID.

BSNSET	2	PROP							

Field	Definition	Type	Default
CSID	The contact node set is assigned to an internal list which is referenced via this CSID. The grid points in the set will be determined by the TYPE specification.	Integer > 0	Required
TYPE	The choices for TYPE are as follows: GRID — The CSID refers to a SET command in the Case Control that defines a list of grid points. ELEM — The CSID refers to a SET command in the Case Control that defines a list of elements. Autodesk Explicit determines all of the unique nodes found in the element set and creates an internal contact node set for these nodes. PROP — The CSID refers to all the elements in a part. Autodesk Explicit determines all of the unique nodes found in the elements of this part and creates an internal contact node set for these nodes.	Character	Required

BSORIENT — Orient a Two-Sided Surface and Make it Single-Sided

Description: The BSORIENT entry defines the surface orientation for a two-sided contact surface. This entry makes the two-sided surface single-sided.

Only two-sided surfaces can use this option. If the contact surface identifier value, CSID, on a BSORIENT entry references a surface derived from solid elements, an error occurs. Solid elements generate single sided faces.

Format:

1	2	3	4	5	6	7	8	9	10
BSORIENT	CSID	REV	USEXYZ	X	Y	Z			

Example 1: Make a contact surface single-sided.

1	2	3	4	5	6	7	8	9	10
BSORIENT	21								

Example 2: Reverse the orientation of a contact surface and make it single-sided.

1	2	3	4	5	6	7	8	9	10
BSORIENT	21	TRUE							

Example 3: Orient a contact surface using the coordinates of a point in space. The surface will be oriented so that all positive normals point toward the point in space.

1	2	3	4	5	6	7	8	9	10
BSORIENT	2	FALSE	TRUE	7.3	-4.2	-0.5			

Field	Definition	Type	Default
CSID	The value for CSID is the contact surface identification number. See Remark 1.	Integer > 0	Required
REV	The REV flag set to TRUE indicates the surface orientation should be reversed. See Remark 2.	Character, either TRUE or FALSE	FALSE
USEXYZ	The USEXYZ flag set to TRUE indicates that the surface should be oriented using the coordinates defined by X, Y, & Z. See Remarks 3 and 4.	Character, either TRUE or FALSE	FALSE
X	X coordinate value of optional orientation point.	Real or blank	
Y	Y coordinate value of optional orientation point.	Real or blank	
Z	Z coordinate value of optional orientation point.	Real or blank	

(Continued)

Remarks:

1. The contact surface identifier references the CSID on a BSSEG entry or the CSID for a contact surface generated with a CONTACTGENERATE command. See the description of the CONTACTGENERATE command.
2. If the USEXYZ flag is true, the surface is first oriented using the coordinates and then reversed if the REV flag is set to TRUE.
3. All surfaces have a natural orientation defined by the positive normals of the faces on the surface. The positive normal for a face follows the right-hand rule for node number on the face. When you view the face, a counter clock-wise number of the nodes produces a normal that points towards you.
4. The coordinates of the orientation point will attempt to orient all the faces on the surface so that the positive normals point towards the orientation point. Consider a cylinder meshed using all CQAUD4 elements. Depending upon on how the mesh was generated, the surface orientation could either point into the cylinder or out of the cylinder. Let us assume that it points out of the cylinder but we want to define contact only on the interior of the cylinder. Then Example 2 above shows how to accomplish this. We simply define a point on the axis of the cylinder and the BSORIENT option selects the inside of the cylinder as the surface (all the normals point towards the centerline of the cylinder).

BSSEG — Boundary Surface Segments

Description: The BSSEG entry defines a contact surface comprised of quadrilateral or triangular segments (faces) that can come in contact with another contact surface. Each segment (face) is defined by a set of grid points.

Format:

1	2	3	4	5	6	7	8	9	10
BSSEG	CSID	G1A	G2A	G3A	G4A	G1B	G2B	G3B	
	G4B	G1C	G2C	G3C	G4C	- etc.-			

Example:

BSSEG	2	3	5	7	9	11	13	15	
		21	27	33	38				

Alternate Format and Example:

BSSEG	CSID	G1	THRU	G2	BY	INC			

BSSEG	10	23	THRU	55	BY	2			

Field	Definition	Type	Default
CSID	Contact surface identification number.	Integer > 0	Required
Gi	Grid point identification numbers. Grid points define quadrilateral or triangular segments (faces) of a surface and must be ordered so that the normal to the segment points outward. See Remark 3.	Integer > 0	Required

Remarks:

1. Contact surface identifiers, CSID, must be unique with respect to all other BSSEG entries. They must also be unique with respect to the CSID's generated with a CONTACTGENERATE command. See the description of the CONTACTGENERATE command.
2. A triangular segment is defined by specifying a zero or blank for the fourth node.
3. The normal to the segment is determined by the ordering of the segment nodes using the right hand rule. Each segment normal of a contact surface must define the outward normal of the face.

BSTHICK — Surface Thickness and Offset

Description: The BSTHICK entry defines the thickness for a two-sided surfaces and an optional surface offset.

In Autodesk Explicit, shell elements give rise to two-sided surfaces. Each surface arising from a single shell element is offset from the mid-plane of the shell by the half-thickness of the element. This thickness for a shell element is defined by the PSHELL or PCOMP option. Two-sided contact can occur only if there is a shell thickness defined. A set of contiguous shell elements, each of which has a defined thickness, defines a two-sided surface. The SPEC flag is used to define the thickness model for the two-sided surface. If the NONE flag is set, the surface will have no thickness and will be made a single-sided surface. The default value of ORIG uses the original half thickness of the shells for the offset of the positive and negative side of the shells.

This option can be used with the BSORIENT option to turn off the thickness (which results in a single-sided surface) and re-orient the surface.

The CUR value for the SPEC declares that the top and bottom surface offsets should be computed from the current thickness of the shell as it deforms.

SPEC = CUR is not yet implemented and is included here for compatibility with future releases.

The OFFSET option is not yet implemented and is included here for compatibility with future releases.

Format:

1	2	3	4	5	6	7	8	9	10
BSTHICK	CSID	SPEC	OFFSET						

Example: Make a surface single-sided with no thickness.

BSTHICK	21	NONE							

Field	Definition	Type	Default
CSID	Contact surface identification number. See Remark 1.	Integer > 0	Required
SPEC	Model to be used for surface. The options are: ORIG Use the original thickness for each element CUR Use the current thickness for each element NONE The surface is single-sided See Remark 2.	Character, either ORIG, CUR or NONE	ORIG
OFFSET	Surface offset: See Remark 3.	Real or blank	0.0

Remarks:

1. The contact surface identifier references the CSID on a BSSEG entry or the CSID for a contact surface generated with a CONTACTGENERATE command. See the description of the CONTACTGENERATE command.

(Continued)

2. Currently, only the ORIG and NONE options are implemented.
3. The OFFSET option is currently not used. The OFFSET option will allow the user to set the top and bottom surfaces at some distance other than half the element thickness from the mid-plane of the element.

BSTYPE — Over-ride Surface Pair PTYPE Specification

Description. The BSTYPE entry defines the PTYPE parameter for a surface contact region (i.e. a set of surfaces paired together for contact). This specification will over-ride the value specified on a BSCONP option that contains the same two surface IDs.

When this option is specified, a search is made through all the surface pairs, whether generated using CONTACTGENERATE or BSCONP to find a match to the secondary and primary IDs. If a match is found, the PTYPE specified here will over-ride the value specified on the CONTACTGENERATE and/or BSCONP option.

Format:

1	2	3	4	5	6	7	8	9	10
BSTYPE	SECONDARY	PRIMARY	PTYPE						

Example:

BSTYPE	2	5	4

Field	Definition	Type	Default
SECONDARY	Secondary region identification number. This is the CSID for a BSSEG entry or a generated contact surface.	Integer > 0	Required
PRIMARY	Primary region identification number. This is the CSID for a BSSET entry or a generated contact surface.	Integer > 0	Required
PTYPE	Penetration type: 1 = Unsymmetric general contact (secondary penetration only) 2 = Symmetric general contact 3 = Unsymmetric welded contact 4 = Symmetric welded contact	$1 \le$ Integer ≤ 4	1

(Continued)

CBAR Simple Beam Element Connection

Description: The CBAR entry defines a simple beam element.

Format:

1	2	3	4	5	6	7	8	9	10
CBAR	EID	PID	GA	GB	X1	X2	X3		

Example:

CBAR	10	100	101	102	0.0	0.0	1.0		

Alternate Format and Example:

1	2	3	4	5	6	7	8	9	10
CBAR	EID	PID	GA	GB	G0/X1	X2	X3		

CBAR	2	39	7	6	105				

Field	Definition	Type	Default
EID	Element identification number..	Integer > 0	Required
PID	Property identification number of a PBAR entry.	Integer > 0	Required
GA, GB	Grid point identification numbers of connection points.	Integer > 0; GA ≠ GB	Required
X1, X2, X3	Components of vector $\bar{\mathbf{v}}$, from GA, in the displacement coordinate system at GA (see Figure 1).	Real or blank	
G0	Grid point identification number to optionally supply X1, X2, and X3. Direction of orientation vector is GA to G0.	Integer or blank	

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. If field 6 is an integer, then G0 is used. If field 6 is blank or real, then X1, X2, X3 are used.
3. G0 cannot be located at GA or GB.

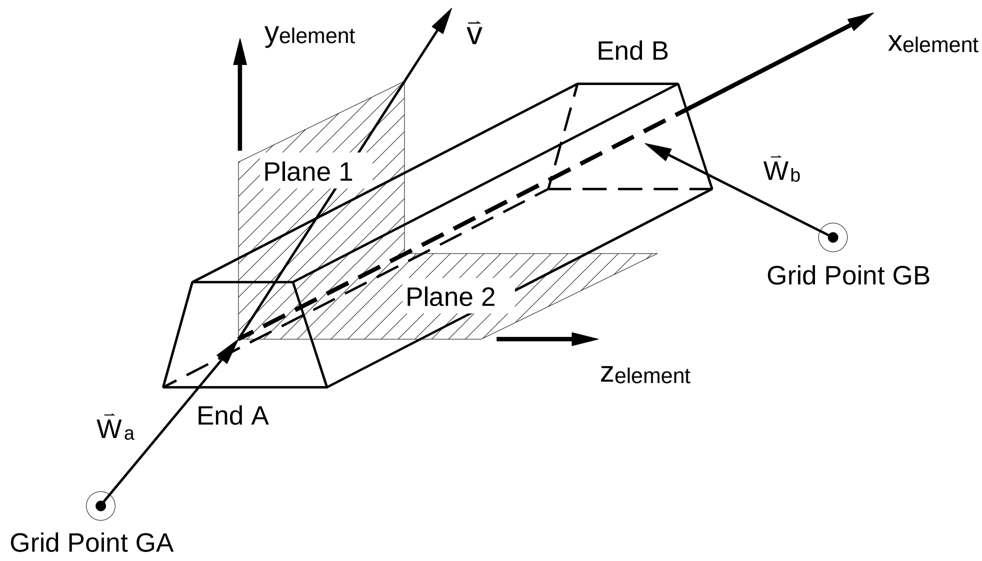


Figure 1. CBAR Element Geometry.

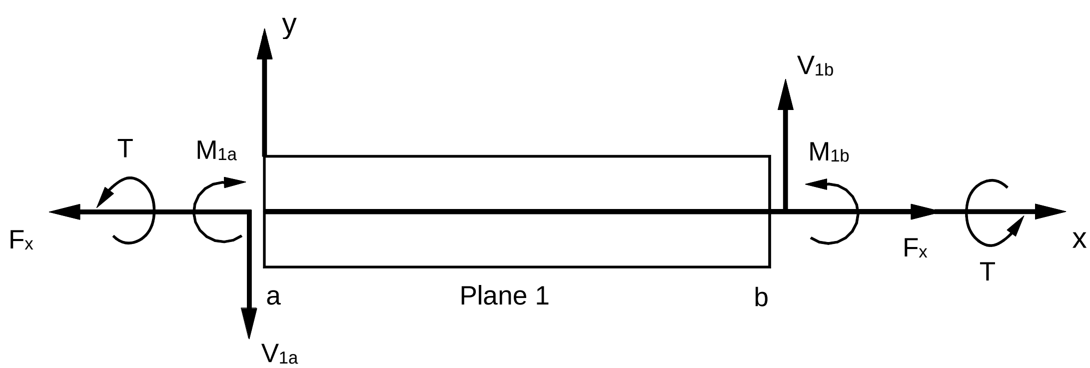


Figure 2. CBAR Element Internal Forces and Moments (xy-Plane).

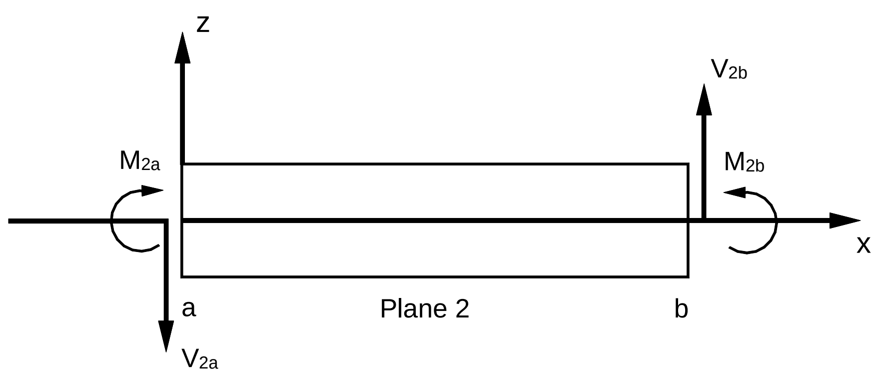


Figure 3. CBAR Element Internal Forces and Moments (xz-Plane).

CBEAM — Beam Element Connection

Description: The CBEAM entry defines a beam element.

Format:

1	2	3	4	5	6	7	8	9	10
CBEAM	EID	PID	GA	GB	G0/X1	X2	X3		

Example:

CBEAM	10	45	5	21	0.5	7.0	-1.3		

Alternate Format:

CBEAM	EID	PID	GA	GB	G0				

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Property identification number of a PBEAM entry.	Integer > 0	Required
GA, GB	Grid point identification numbers of connection points.	Integer > 0; GA $\neq$ GB	Required
X1, X2, X3	Components of vector $\vec{\mathbf{v}}$, from GA, in the displacement coordinate system at GA.	Real or blank	
G0	Grid point identification number to optionally supply X1, X2, and X3. Direction of orientation vector is GA to G0.	Integer or blank	

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. The following figure defines beam element geometry:

(Continued)

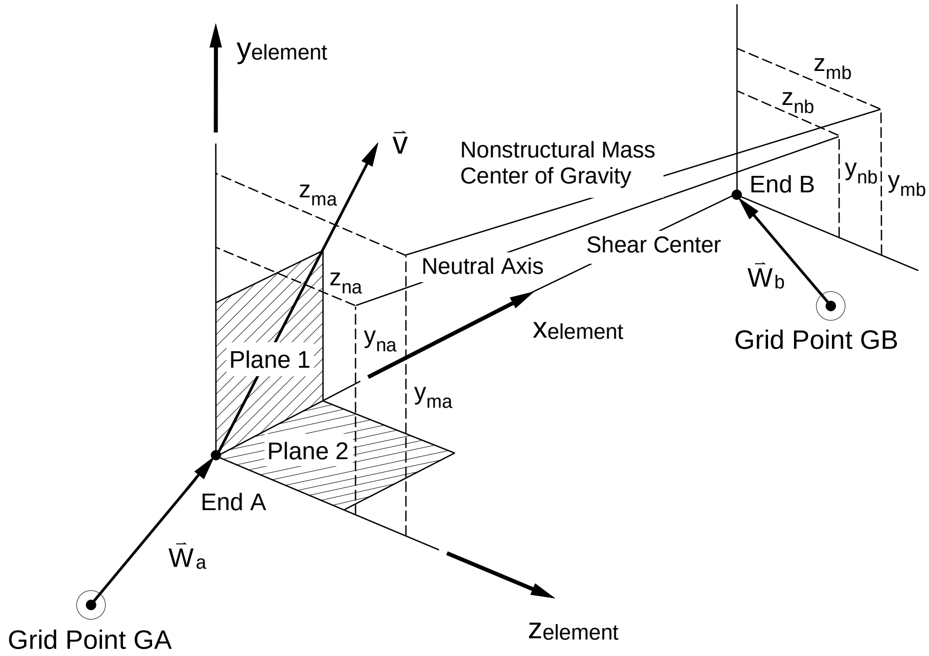


Figure 1. CBEAM Element Geometry System.

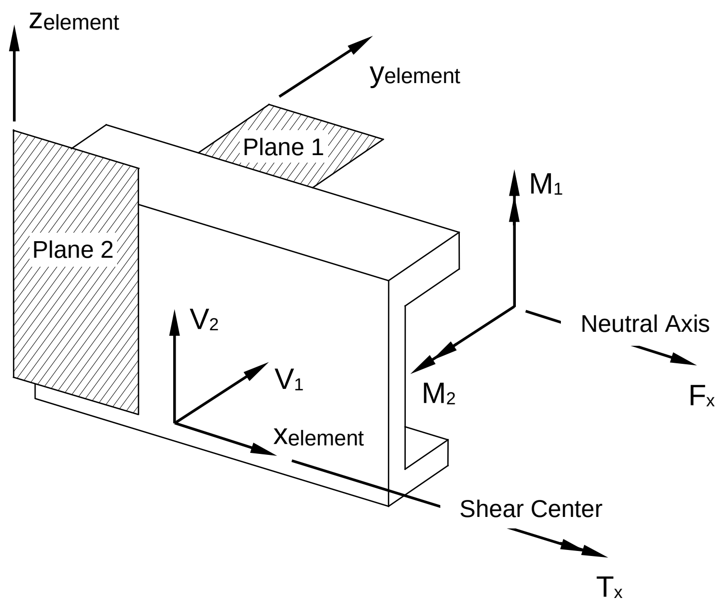


Figure 2. CBEAM Internal Element Forces and Moments.

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 continuation may be omitted if there are no pin flags or offsets.

CDAMP1 — Scalar Damper Connection

Description: The CDAMP1 entry defines a scalar damper element.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP1	EID	PID	G1	C1	G2	C2			

Example:

1	2	3	4	5	6	7	8	9	10
CDAMP1	12	101	G1	C1	22	4			

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Property identification number of a PDAMP entry.	Integer > 0	Required
G1, G2	Grid point identification numbers of connection points.	Integer > 0	See Remark 2
C1, C2	Component numbers.	0 ≤ Integer ≤ 6	See Remark 2

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. A 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 whose displacement is constrained to zero.
3. The two connection points (G1, C1) and (G2, C2), must be distinct.
4. If Gi refers to a grid point, then Ci refers to component numbers in the displacement coordinate system specified by CD on the GRID entry.

CDAMP2 — Scalar Damper Property and Connection

Description: The CDAMP2 entry defines a scalar damper element without reference to a property entry

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP2	EID	B	G1	C1	G2	C2	GE	S	

Example:

1	2	3	4	5	6	7	8	9	10
CDAMP2	124	2.98	32	1	131	1			

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
B	Value of scalar damper.	Real	Required
G1, G2	Grid point identification numbers of connection points.	Integer > 0	See Remark 2
C1, C2	Component numbers.	0 ≤ Integer ≤ 6	See Remark 2

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. A 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 whose displacement is constrained to zero.
3. This single entry completely defines the element since no material or geometric properties are required.
4. The two connection points (G1, C1) and (G2, C2) must be distinct.
5. If Gi refers to a grid point, then Ci refers to component numbers in the displacement coordinate system specified by CD on the GRID entry.

CELAS1 — Scalar Spring Connection

Description: The CELAS1 entry 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	12	101			22	4			

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Property identification number of a PELAS entry.	Integer > 0	Required
G1, G2	Grid point identification numbers of connection points.	Integer > 0	See Remark 2
C1, C2	Component numbers.	0 ≤ Integer ≤ 6	See Remark 2

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. A 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 whose displacement is constrained to zero.
3. The two connection points (G1, C1) and (G2, C2), must be distinct.
4. If Gi refers to a grid point, then Ci refers to component numbers in the displacement coordinate system specified by CD on the GRID entry.

CELAS2 — Scalar Spring Property and Connection

Description: The CELAS2 entry 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			

Example:

CELAS2	124	1.0+4	44	5	45	5			

Field	Definition	Type	Default
EID	Element identification number.	Integer $>$ 0	Required
K	Stiffness value.	Real	Required
G1, G2	Grid point identification numbers of connection points.	Integer $>$ 0	See Remark 2
C1, C2	Component numbers.	$0 \le$ Integer ≤ 6	See Remark 2

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. A 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 whose displacement is constrained to zero.
3. This single entry completely defines the element since no material or geometric properties are required.
4. The two connection points (G1, C1) and (G2, C2) must be distinct.
5. If Gi refers to a grid point, then Ci refers to component numbers in the displacement coordinate system specified by CD on the GRID entry.

CHEXA — Six-Sided Solid Element Connection

Description: The CHEXA entry defines the connections of a six-sided solid 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							

Example:

CHEXA	71	4	3	4	5	6	7	8	
	9	10							

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Property identification number of a PSOLID entry.	Integer > 0	Required
Gi	Grid point identification numbers of connection points.	Integer ≥ 0 or blank, all unique	Required

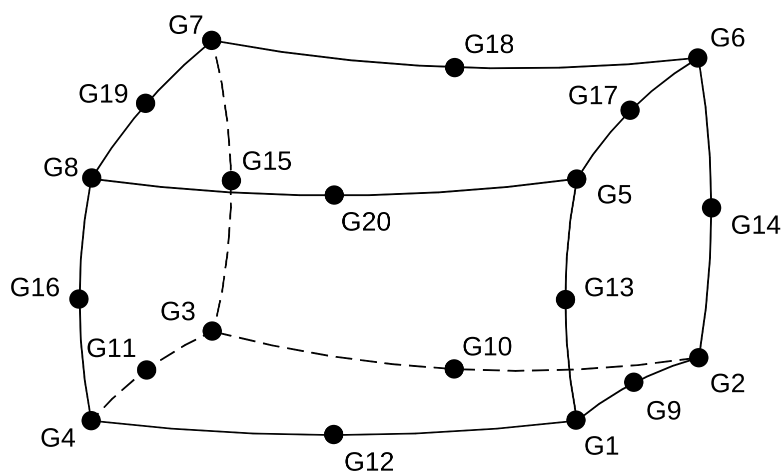


Figure 1. CHEXA Element Connection

(Continued)

Remarks:

1. Element identification numbers must 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. Grid points G5 through G8 must be in order in the same direction around the opposite face with G5 opposite G1, G6 opposite G2, etc.
3. Components of stress are output in the global coordinate system unless a material coordinate system is defined on the PSOLID entry.
4. Figure 1 shows the numbering for both an eight-node and a twenty-node hexahedral element. Autodesk Explicit supports only an eight-node hexahedral element.

CONM2 — Concentrated Mass Element Connection

Description: The CONM2 entry defines a concentrated mass at a grid point. The CONM2 entry is also used to define mass properties for a rigid body

The use of the CONM2 entry for Autodesk Explicit has been modified from the original CONM2 entry for Nastran. If the grid point, G, listed on the entry is the same as a reference node for a rigid body, the mass properties on the CONM2 entry are applied to a rigid body.

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	1	2	12						
	23.5		32.6		12.8				

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
G	Grid point identification number. See Remark 2.	Integer > 0	Required
CID	Coordinate system identification number.	Integer ≥ 0	0
M	Mass value.	Real	Required
X1, X2, X3	Offset distances. These are currently not used by Autodesk Explicit.	Real or blank	0.0
Iij	Mass moments of inertia added to the calculated inertia tensor for the rigid body. The inertia terms are defined in terms of the coordinate system referenced by the coordinate system identification number CID. See Remark 5.	I11, I22, and I33, Real ≥ 0.0; I21, I31, and I32, Real ≥ 0.0.	0.0

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. If the grid point identification number, G, is the same as the reference node for a rigid body, the mass and inertia terms are added to the mass and inertia calculated from the rigid body definition. If the grid point identification number, G, is not the reference node for a rigid body, the mass is simply added to the grid point defined by G.
3. The coordinate system identification is used only if the CONM2 entry references a rigid body.
4. The X1, X2 and X3 offsets are not used by Autodesk Explicit.
5. The continuation entry may be omitted.
6. The form of the inertia matrix about its center of gravity is taken as:

(Continued)

$$\begin{bmatrix} M & & & & & \\ & M & & & & \\ & & M & & & \\ & & & I_{11} & & \\ & & & -I_{21} & I_{22} & \\ & & & -I_{31} & -I_{32} & I_{33} \end{bmatrix}$$

In the above equations:

$$M = \int \rho dV$$

$$I_{11} = \int \rho\left(x_2^2 + x_3^2\right)dV$$

$$I_{22} = \int \rho\left(x_1^2 + x_3^2\right)dV$$

$$I_{33} = \int \rho\left(x_1^2 + x_2^2\right)dV$$

$$I_{21} = \int \rho x_1 x_2 dV$$

$$I_{31} = \int \rho x_1 x_3 dV$$

$$I_{32} = \int \rho x_2 x_3 dV$$

If the CONM2 entry references a rigid body, the mass M is added to the mass that is initially computed for the rigid body, M_C. The total mass for the rigid body, **M**, is $M+M_C$.

The x_1, x_2, x_3 are components of distance from the center of gravity in the coordinate system defined in field 4. Only the magnitude of I_{ij} should be supplied; the negative signs for the off-diagonal terms are supplied automatically. The inertia terms are transformed to the principal coordinate system for the inertia tensor for the rigid body.

CORD1C — Cylindrical Coordinate System Definition, Form 1

Description: The CORD1C entry defines a cylindrical coordinate system by referencing to three grid point identification numbers.

Format:

1	2	3	4	5	6	7	8	9	10
CORD1C	CIDA	G1A	G2A	G3A	CIDB	G1B	G2B	G3B	

Example:

CORD1C	4	2	44	67					

Field	Definition	Type	Default
CID	Coordinate system identification number.	Integer > 0	Required
GiA, GiB	Grid point identification numbers.	Integer > 0, G1A ≠ G2A ≠ G3A, G1B ≠ G2B ≠ G3B	Required

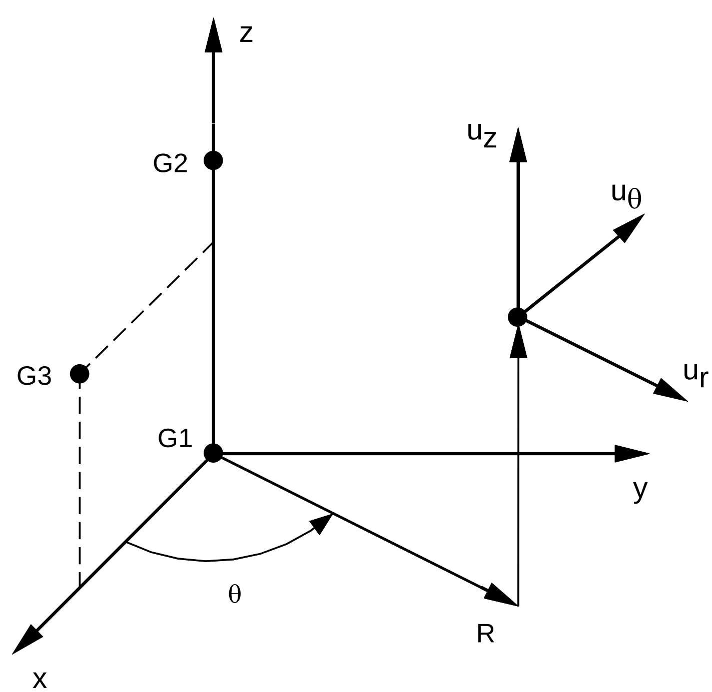


Figure 1. CORD1C Definition.

(Continued)

Remarks:

1. Coordinate system identification numbers on all CORD1C, CORD1R, CORD1S, CORD2C, CORD2R, and CORD2S entries must all be unique.
2. One or two coordinate systems may be defined on a single entry.
3. GiA and GiB must be defined in coordinate whose definition 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 azimuth origin. The three grid points GiA (or GiB) must be noncolinear and not coincident.
4. Coordinate systems defined using CORD1C, CORD1R, and CORD1S entries can not be used as reference coordinate systems on CORD2C, CORD2R, and CORD2S entries.
5. The location of a grid point (P in the sketch) in this coordinate system is given by (R, θ, Z) where θ is measured in degrees.
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. Points on the z-axis will not have their displacement directions defined in this coordinate system since a unique value of θ cannot be defined on the z-axis. For the case of points on the z-axis, the basic rectangular system will be used for displacement output.

CORD1R — Rectangular Coordinate System Definition, Form 1

Description: The CORD1R entry defines a rectangular coordinate system by referencing three grid point identification numbers.

Format:

1	2	3	4	5	6	7	8	9	10
CORD1R	CID	G1A	G2A	G3A	CID	G1B	G2B	G3B	

Example:

CORD1R	3	16	32	19					

Field	Definition	Type	Default
CID	Coordinate system identification number.	Integer > 0	Required
GiA, GiB	Grid point identification numbers.	Integer > 0, G1A ≠ G2A ≠ G3A, G1B ≠ G2B ≠ G3B	0

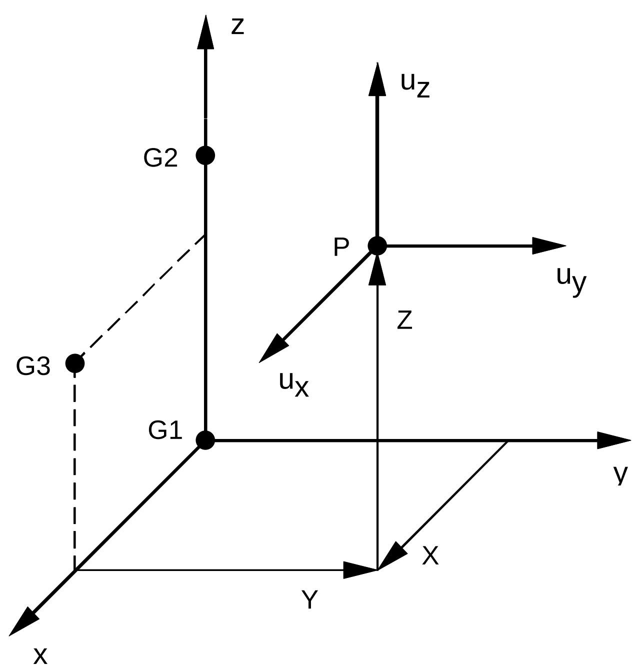


Figure 1. CORD1R Definition.

(Continued)

Remarks:

1. Coordinate system identification numbers on all CORD1C, CORD1R, CORD1S, CORD2C, CORD2R, and CORD2S entries must all be unique.
2. One or two coordinate systems may be defined on a single entry.
3. GiA and GiB must be defined in coordinate whose definition 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 azimuth origin. The three grid points GiA (or GiB) must be non-colinear and not coincident.
4. Coordinate systems defined using CORD1C, CORD1R, and CORD1S entries can not be used as reference coordinate systems on CORD2C, CORD2R, and CORD2S entries.
5. The location of a grid point (P in the sketch) in this coordinate system is given by (X, Y, Z).
6. The displacement coordinate directions at P are dependent on the location of P as shown above by (u_X, u_Y, u_Z).

CORD1S — Spherical Coordinate System Definition, Form 1

Description: The CORD1S entry defines a spherical coordinate system by referencing three grid point identification numbers.

Format:

1	2	3	4	5	6	7	8	9	10
CORD1S	CID	G1A	G2A	G3A	CID	G1B	G2B	G3B	

Example:

CORD1S	4	5	43	55					

Field	Definition	Type	Default
CID	Coordinate system identification number.	Integer > 0	Required
GiA, GiB	Grid point identification numbers.	Integer > 0, G1A $\neq$ G2A $\neq$ G3A, G1B $\neq$ G2B $\neq$ G3B	Required

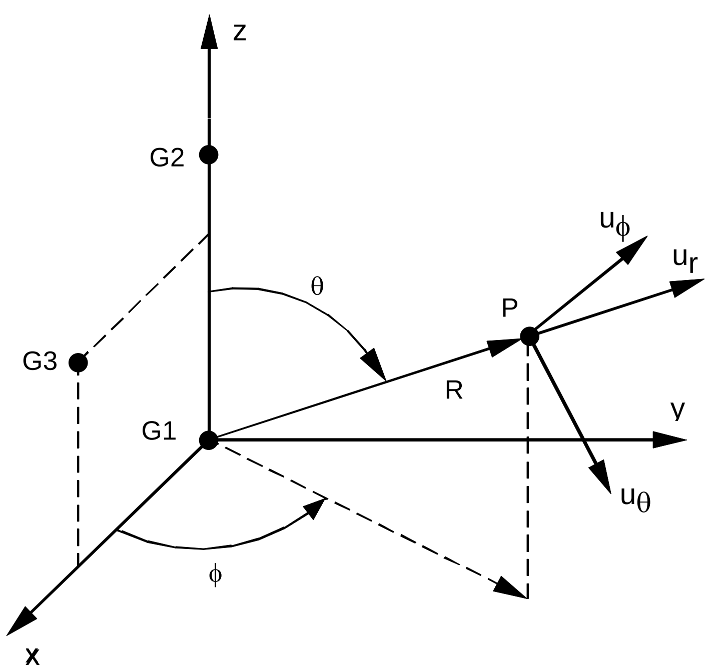


Figure 1. CORD1S Definition.

(Continued)

Remarks:

1. Coordinate system identification numbers on all CORD1C, CORD1R, CORD1S, CORD2C, CORD2R, and CORD2S entries must all be unique.
2. One or two coordinate systems may be defined on a single entry.
3. GiA and GiB must be defined in coordinate whose definition 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 azimuth origin. The three grid points GiA (or GiB) must be noncolinear and not coincident.
4. Coordinate systems defined using CORD1C, CORD1R, and CORD1S entries can not be used as reference coordinate systems on CORD2C, CORD2R, and CORD2S entries.
5. The location of a grid point (P in the sketch) in this coordinate system is given by (R, θ, ϕ) where θ and ϕ are measured in degrees.
6. The displacement coordinate directions at P are dependent on the location of P as shown above by (u_r, u_θ, u_ϕ).
7. Points on the z-axis will not have their displacement directions defined in this coordinate system since unique values of θ and ϕ can not be defined on the z-axis. For the case of points on the z-axis, the basic rectangular system will be used for displacement output.

CORD2C — Cylindrical Coordinate System Definition, Form 2

Description: The CORD2C entry defines a cylindrical coordinate system with 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	5		0.0	0.0	0.0	0.0	0.0	1.0	
	1.0	1.0	0.0						

Field	Definition	Type	Default
CID	Coordinate system identification number.	Integer > 0	Required
RID	Identification number of a coordinate system that is defined independently from this coordinate system.	Integer ≥ 0	0
Ai, Bi, Ci	Coordinates of three points in coordinate system defined in field 3.	Real	Required

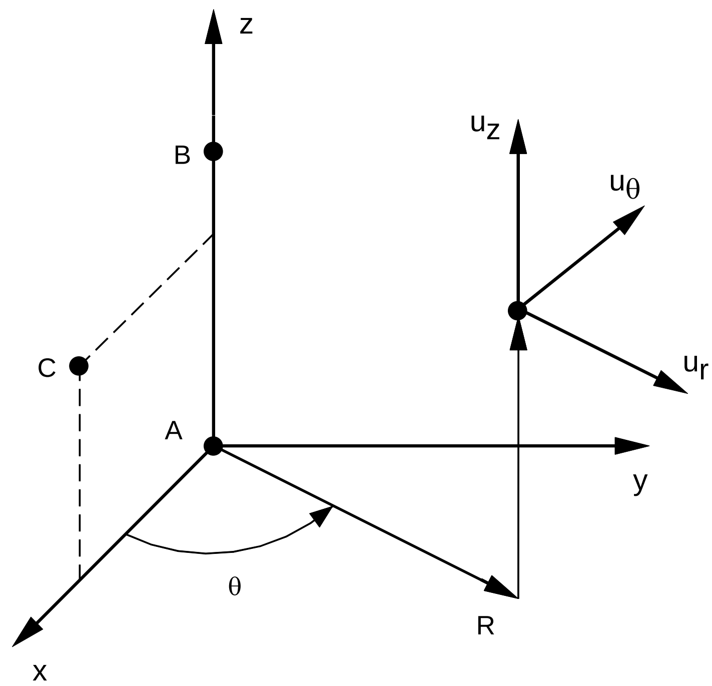


Figure 1. CORD2C Definition.

Remarks:

1. Continuation entry must be present.
2. The three points (A1, A2, A3), (B1, B2, B3), (C1, C2, C3) must be unique and non-colinear. The model translator checks for non-colinearity.
3. Coordinate system identification numbers on all CORD1C, CORD1R, CORD1S, CORD2C, CORD2R, and CORD2S entries must all be unique.
4. The reference coordinate system must be independently defined.
5. A RID of zero (or blank) references the basic coordinate system.
6. The location of a grid point (P in the sketch) in this coordinate system is given by (R, θ, Z) where θ is measured in degrees.
7. The displacement coordinate directions at P are dependent on the location of P as shown above by (u_r, u_θ, u_z).
8. Points on the z-axis will not have their displacement directions defined in this coordinate system since a unique value of θ can not be defined on the z-axis. For the case of points on the z-axis, the basic rectangular system will be used for displacement output.

CORD2R — Rectangular Coordinate System Definition, Form 2

Description: The CORD2R entry defines a rectangular coordinate system with 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	5		0.0	0.0	0.0	0.0	0.0	1.0	
	1.0	1.0	0.0						

Field	Definition	Type	Default
CID	Coordinate system identification number.	Integer > 0	Required
RID	Identification number of a coordinate system that is defined independently from this coordinate system.	Integer ≥ 0	0
Ai, Bi, Ci	Coordinates of three points in coordinate system defined in field 3.	Real	Required

(Continued)

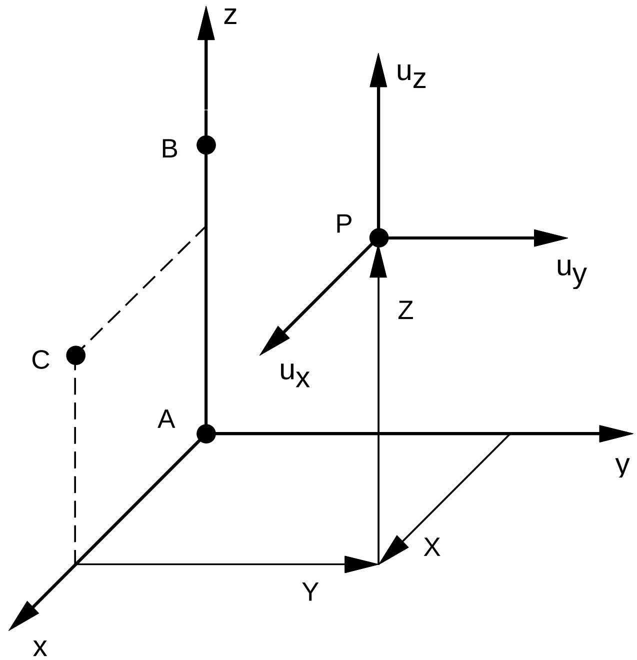


Figure 1. CORD2R Definition.

Remarks:

1. Continuation entry must be present.
2. The three points (A1, A2, A3), (B1, B2, B3), (C1, C2, C3) must be unique and non-colinear. The model translator checks for non-colinearity.
3. Coordinate system identification numbers on all CORD1C, CORD1R, CORD1S, CORD2C, CORD2R, and CORD2S entries must all be unique.
4. The reference coordinate system must be independently defined.
5. A RID of zero (or blank) references the basic coordinate system.
6. The location of a grid point (P in the sketch) in this coordinate system is given by (X, Y, Z).
7. The displacement coordinate directions at P are dependent on the location of P as shown above by (u_X, u_Y, u_Z).

CORD2S Spherical Coordinate System Definition, Form 2

Description: The CORD2S entry defines a spherical coordinate system with 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	5		0.0	0.0	0.0	0.0	0.0	1.0	
	1.0	1.0	0.0						

Field	Definition	Type	Default
CID	Coordinate system identification number.	Integer > 0	Required
RID	Identification number of a coordinate system that is defined independently from this coordinate system.	Integer ≥ 0	0
Ai, Bi, Ci	Coordinates of three points in coordinate system defined in field 3.	Real	Required

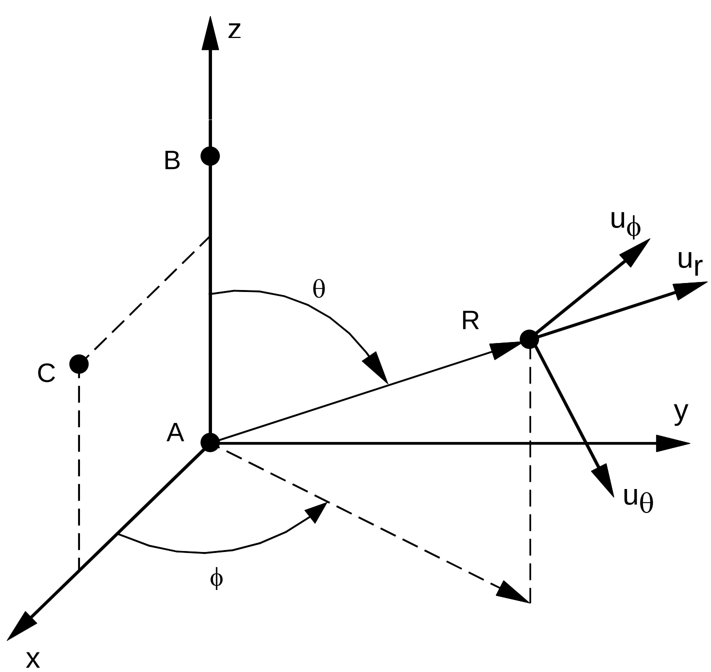


Figure 1. CORD2S Definition.

(Continued)

Remarks:

1. Continuation entry must be present.
2. The three points (A1, A2, A3), (B1, B2, B3), (C1, C2, C3) must be unique and non-colinear. The model translator checks for non-colinearity.
3. Coordinate system identification numbers on all CORD1C, CORD1R, CORD1S, CORD2C, CORD2R, and CORD2S entries must all be unique.
4. The reference coordinate system must be independently defined.
5. A RID of zero (or blank) references the basic coordinate system.
6. The location of a grid point (P in the sketch) in this coordinate system is given by (R, θ, ϕ) where θ and ϕ are measured in degrees.
7. The displacement coordinate directions at P are dependent on the location of P as shown above by (u_r, u_θ, u_ϕ).
8. Points on the z-axis will not have their displacement directions defined in this coordinate system since unique values of θ and ϕ can not be defined on the z-axis. For the case of points on the z-axis, the basic rectangular system will be used for displacement output.

CPENTA — Five-Sided Solid Element Connection

Description: The CPENTA entry defines the connections of a five-sided solid element with six grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CPENTA	EID	PID	G1	G2	G3	G4	G5	G6	

Example:

CPENTA	112	2	3	15	14	4	103	115	

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Property identification number of a PSOLID entry.	Integer > 0	Required
Gi	Grid point identification numbers of connection points.	Integer ≥ 0 or blank	Required

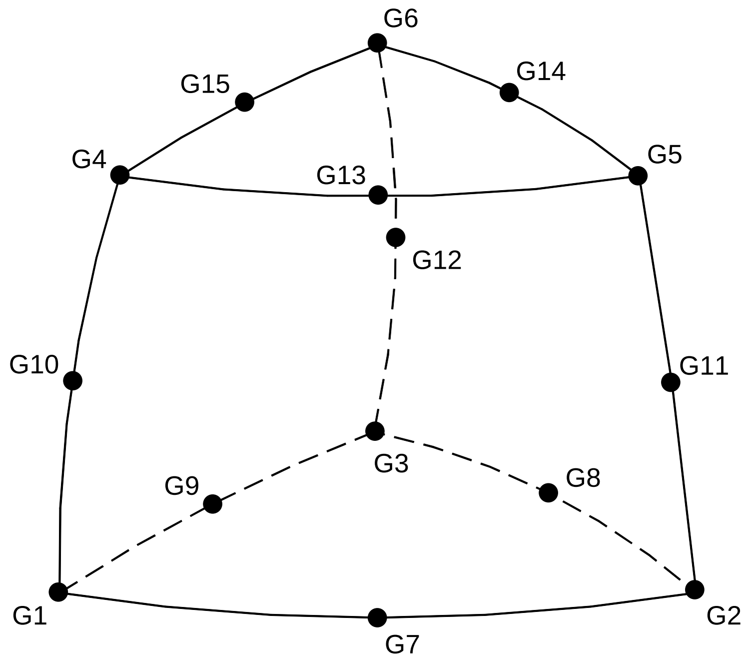


Figure 1. CPENTA Element Connection.

(Continued)

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. The topology of the diagram must be preserved; i.e., G1, G2, G3 define a triangular face.
3. Components of stress are output in the global coordinate system unless a material coordinate system is defined on the PSOLID entry.
4. Figure 1 shows the numbering for both a six-node and a fifteen-node triangular prismatic element. Autodesk Explicit supports only a six-node triangular prismatic element.

CQUAD4 — Quadrilateral Plate Element Connection

Description: The CQUAD4 entry defines a quadrilateral element that can be used as a membrane or shell. (A shell element has bending stiffness while the membrane element does not.) For both the membrane and shell formulations, the state of the stress in the element is plane stress. The elements do have strains through the thickness of the element, which will lead to thinning of the element.

Format:

1	2	3	4	5	6	7	8	9	10
CQUAD4	EID	PID	G1	G2	G3	G4	THETA/MCID		

Example:

CQUAD4	61	11	101	111	201	202	0.0		

Field	Definition	Type	Default
EID	Element identification number.	Integer $>$ 0	Required
PID	Identification number of a PSHELL or PCOMP property entry.	Integer $>$ 0	Required
Gi	Grid point identification numbers of connection points.	Integer $>$ 0, all unique	Required
THETA	Material property orientation angle in degrees.	Real or blank	See Remark 5
MCID	Material coordinate system identification number.	Integer $\geq$ 0	See Remark 5

Remarks:

1. Element identification numbers must 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 degrees.
4. Stresses are output in the global coordinate system unless THETA or MCID is used to define a local coordinate system.
5. If THETA/MCID field is blank, field 5 of the PSHELL continuation entry will be used. If this field is also blank, then THETA = 0.0 is assumed.
6. This element is identical to the CQUADR element.

(Continued)

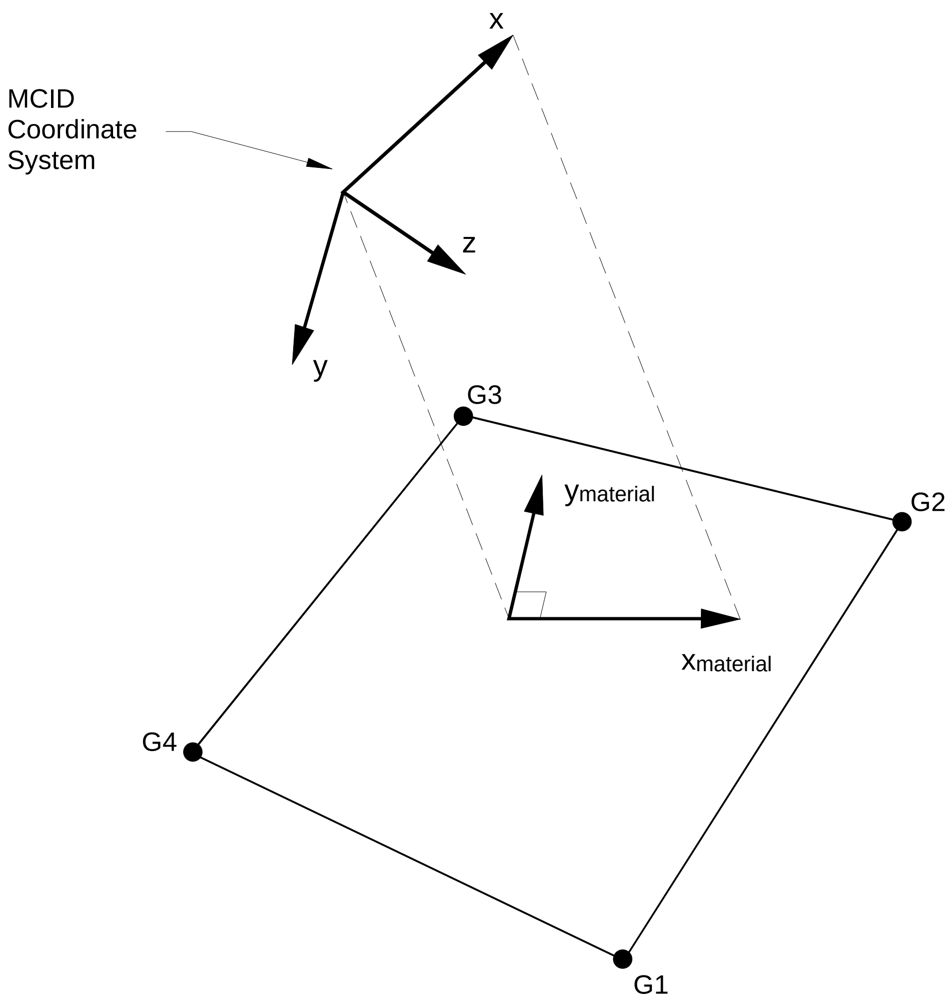


Figure 1. MCID Coordinate System Definition.

(Continued)

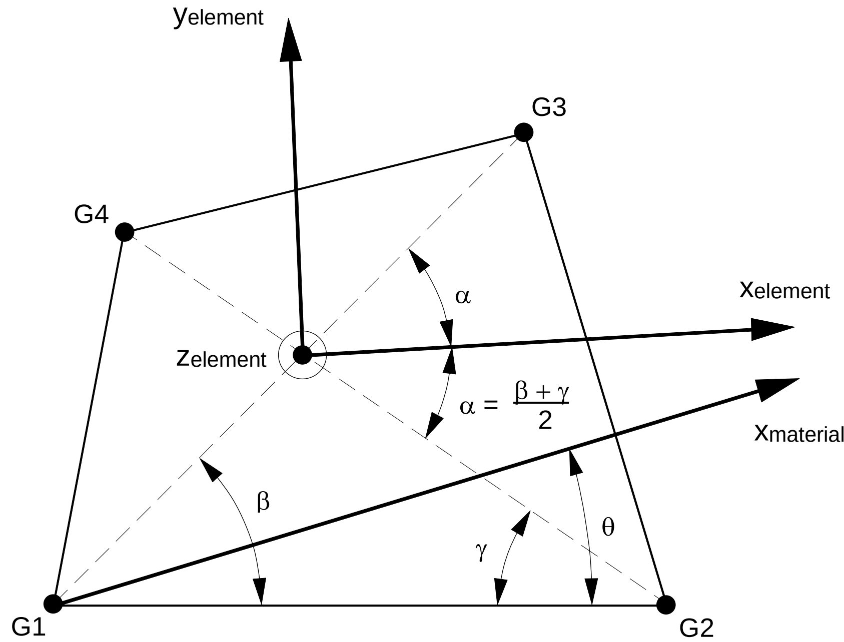
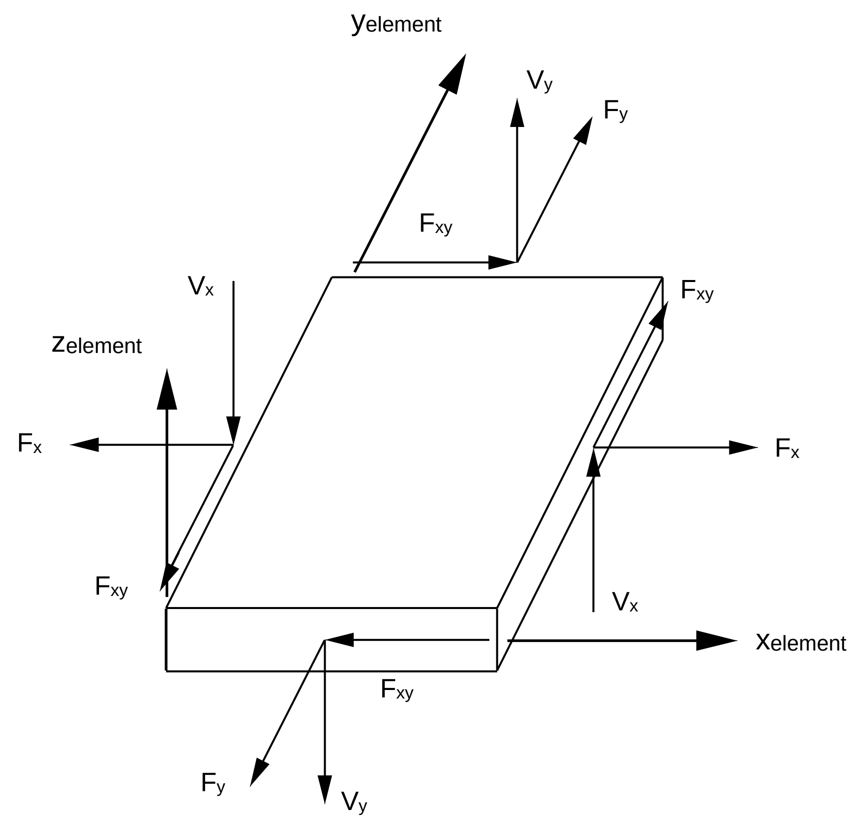


Figure 2. CQUAD4 Element Geometry and Coordinate System.

(Continued)



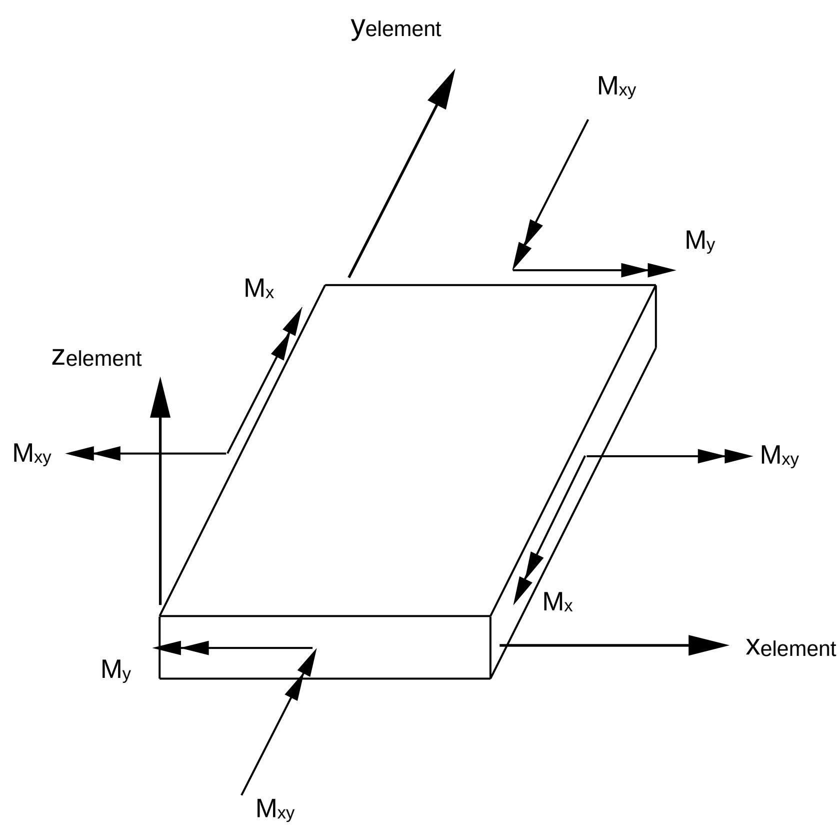


Figure 3. Forces and Moments in CQUAD4 Elements.

(Continued)

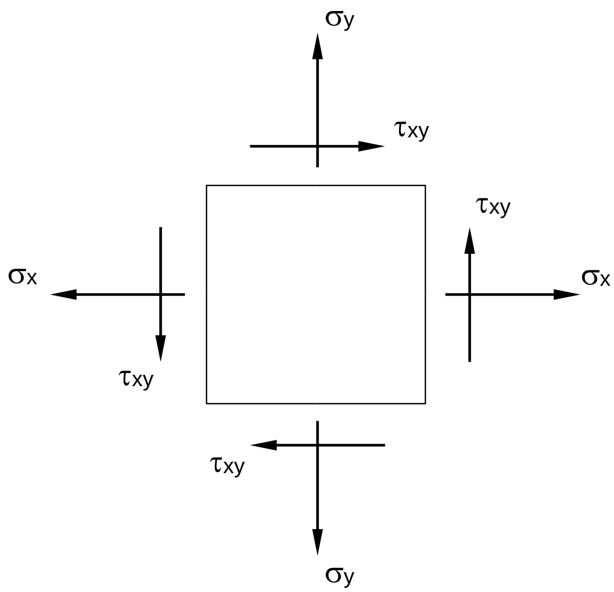


Figure 4. Stresses in CQUAD4 Elements.

CQUADR — Quadrilateral Plate Element Connection

Description: The CQUADR entry defines a quadrilateral element that can be used as a membrane or shell. (A shell element has bending stiffness while the membrane element does not.) For both the membrane and shell formulations, the state of the stress in the element is plane stress. The elements do have strains through the thickness of the element, which will lead to thinning of the element.

Format:

1	2	3	4	5	6	7	8	9	10
CQUADR	EID	PID	G1	G2	G3	G4	THETA/MCID		

Example:

CQUADR	61	11	101	111	201	202			

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Identification number of a PSHELL or PCOMP property entry.	Integer > 0	Required
Gi	Grid point identification numbers of connection points.	Integer > 0 all unique	Required
THETA	Material property orientation angle in degrees.	Real	
MCID	Material coordinate system identification number.	Integer ≥ 0	See Remark 7

Remarks:

1. Element identification numbers must 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^{o}.
4. Components of stress are output in the surface coordinate system unless THETA or MCID are used to define a local coordinate system.
5. The rotational degrees of freedom at the connection points and normal to the element are active in the element formulation and must not be constrained unless at a boundary.
6. This element is identical to the CQUAD4.
7. If THETA/MCID field is blank, field 5 of the PSHELL continuation entry will be used. If this field is also blank, then THETA = 0.0 is assumed.

(Continued)

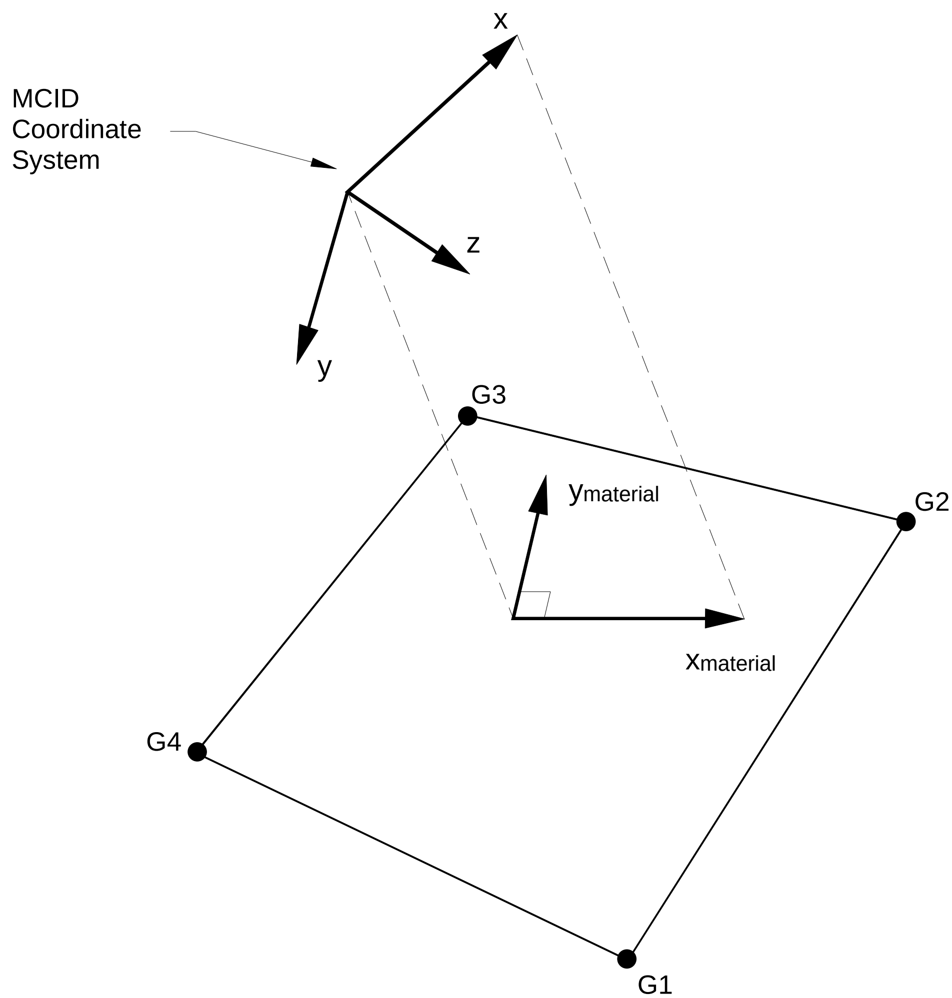


Figure 1. MCID Coordinate System Definition.

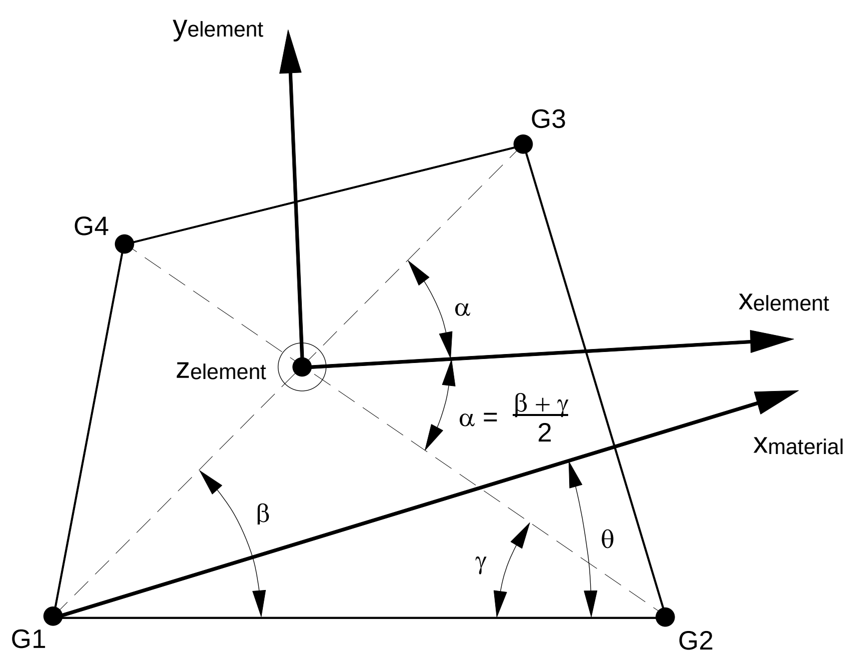


Figure 2. CQUADR Element Geometry and Coordinate System.
(Continued)

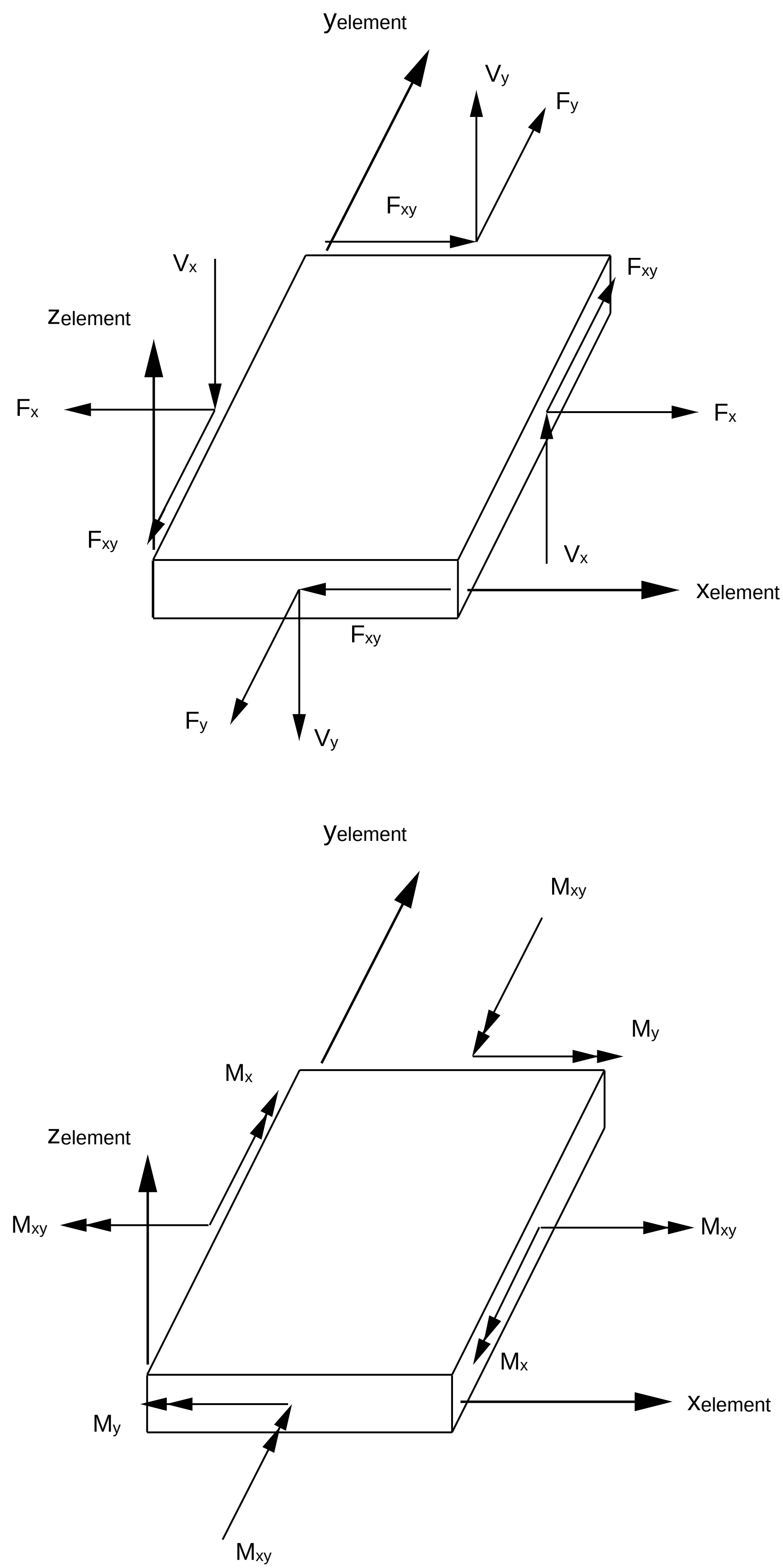


Figure 3. Forces and Moments in CQUADR Elements.

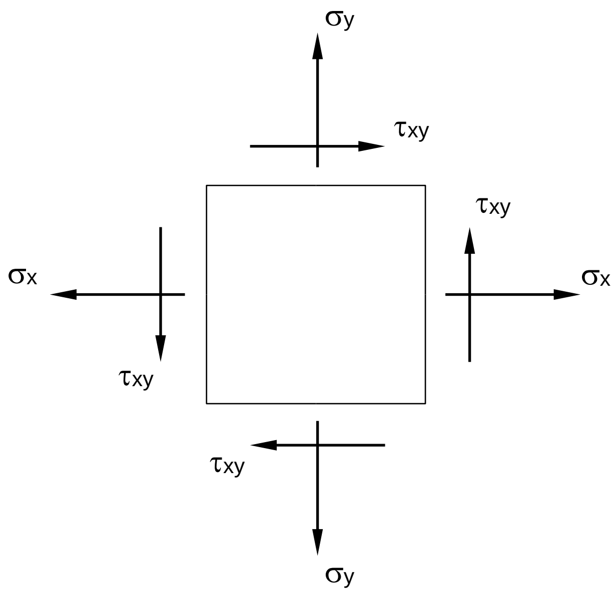


Figure 4. Stresses in CQUADR Elements.

CROD — Rod Element Connection

Description: The CROD entry defines a truss element. The element carries only tension or compression. There is no bending or torsion in the element.

Format:

1	2	3	4	5	6	7	8	9	10
CROD	EID	PID	G1	G2					

Example:

CROD	61	11	101	111					

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Identification number of a PROD property entry.	Integer > 0	Required
G1, G2	Grid point identification numbers of connection points.	Integer > 0, G1 ≠ G2	Required

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.

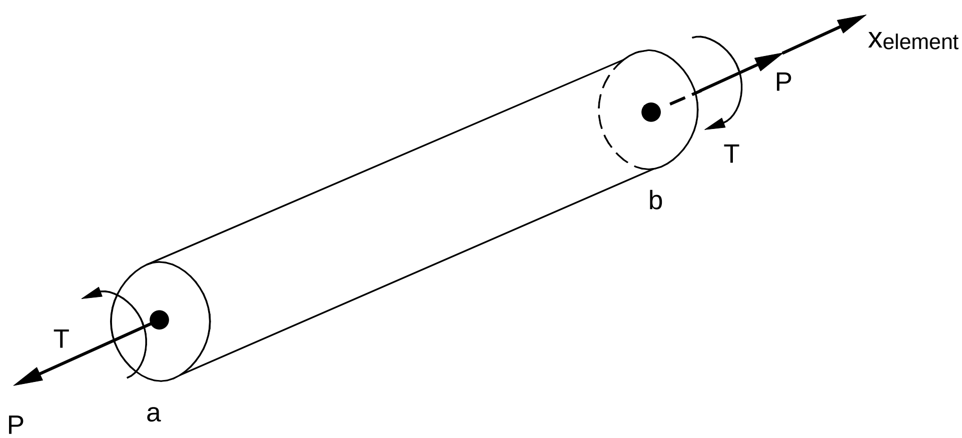


Figure 1. CROD Element Internal Forces and Moments.

CTETRA Four-Sided Solid Element Connection

Description: The CTETRA entry defines the connections of a four-sided solid element with four grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CTETRA	EID	PID	G1	G2	G3	G4			

Example:

CTETRA	112	2	3	15	14	4			

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Property identification number of a PSOLID entry.	Integer > 0	Required
Gi	Grid point identification numbers of connection points.	Integer ≥ 0 or blank, all unique	Required

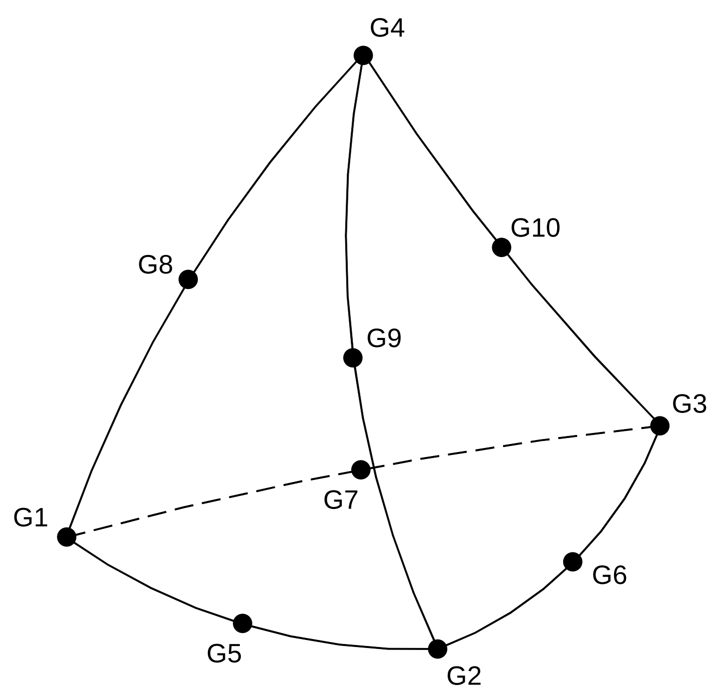


Figure 1. CTETRA Element Connection.

(Continued)

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. The topology of the diagram must be preserved; i.e., G1, G2, G3 define a circular face.
3. Components of stress are output in the global coordinate system unless a local coordinate system is defined on the PSOLID option.
4. Figure 1 shows the numbering for both a four-node and a ten-node tetrahedral element. Autodesk Explicit supports only a four-node tetrahedral element.

CTRIA3 Triangular Element Connection

Description: The CTRIA3 entry defines a triangular element that can be used as a membrane or shell. (A shell element has bending stiffness while the membrane element does not.) For both the membrane and shell formulations, the state of the stress in the element is plane stress. The elements do have strains through the thickness of the element, which will lead to thinning of the element.

Format:

1	2	3	4	5	6	7	8	9	10
CTRIA3	EID	PID	G1	G2	G3	THETA/MCID			

Example:

CTRIA3	61	11	101	111	202				

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Identification number of a PSHELL property entry.	Integer > 0	Required
Gi	Grid point identification numbers of connection points.	Integer > 0, all unique	Required
THETA	Material property orientation angle in degrees.	Real	See Remark 3
MCID	Material coordinate system identification number.	Integer ≥ 0	See Remark 3

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. Stresses are output in the surface coordinate system or in the coordinate system defined using THEATA/MCID or the coordinate system defined on the PSHELL continuation entry.
3. If THETA/MCID is blank, field 5 of the PSHELL continuation entry will be used. If this field is also blank, then THETA = 0.0 is assumed.
4. This element is identical to the CTRIAR element.

(Continued)

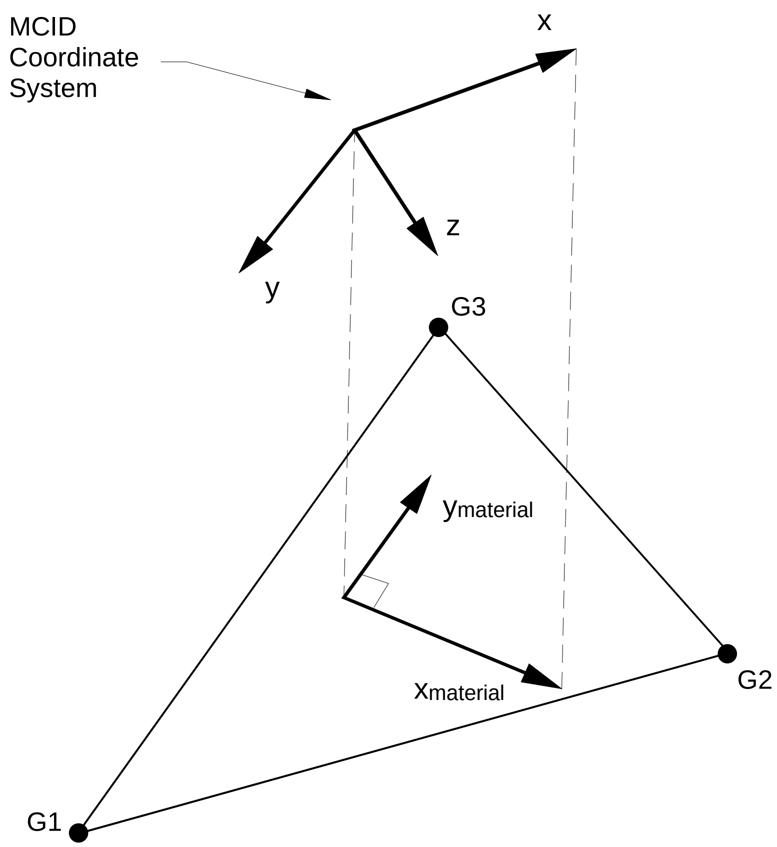


Figure 1. MCID Coordinate System Definition.

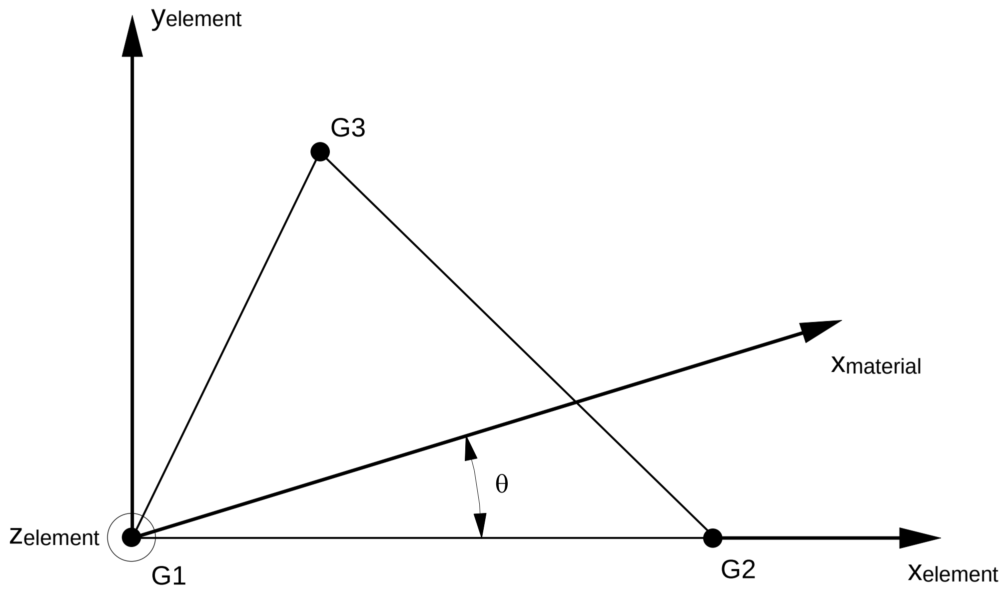


Figure 2. CTRIA3 Element Geometry and Coordinate Systems.

(Continued)

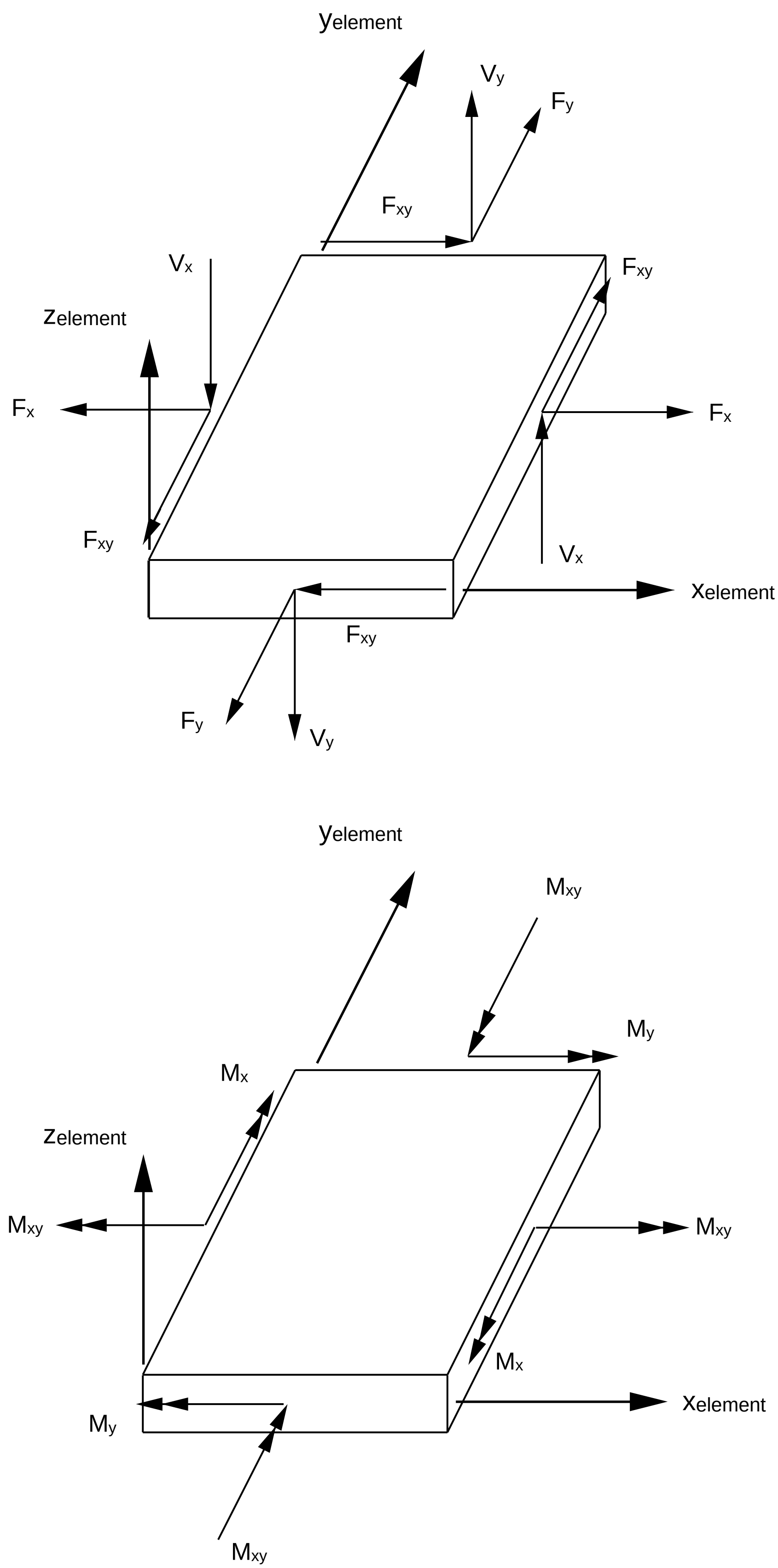


Figure 3. Forces and Moments in CTRIA3 Elements.

(Continued)

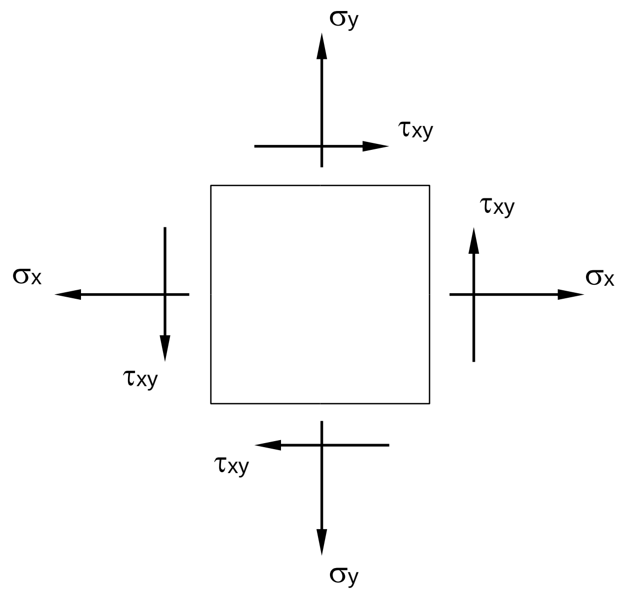


Figure 4. Stresses in CTRIA3 Elements.

CTRIAR Triangular Element Connection

Description: The CTRIAR entry defines a triangular element that can be used as a membrane or shell. (A shell element has bending stiffness while the membrane element does not.) For both the membrane and shell formulations, the state of the stress in the element is plane stress. The elements do have strains through the thickness of the element, which will lead to thinning of the element.

Format:

1	2	3	4	5	6	7	8	9	10
CTRIAR	EID	PID	G1	G2	G3	THETA/MCID			

Example:

CTRIAR	61	11	101	111	202				

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Identification number of a PSHELL property entry.	Integer > 0	Required
Gi	Grid point identification numbers of connection points.	Integer > 0, all unique	Required
THETA	Material property orientation angle in degrees.	Real	See Remark 4
MCID	Material coordinate system identification number.	Integer ≥ 0	See Remark 4

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. Stresses are output in the surface coordinate system or in the coordinate system defined using THETA/MCID or the coordinate system defined on the PSHELL continuation entry.
3. The rotational degrees of freedom at the connection points and normal to the element are active in the element formulation and must not be constrained unless at a boundary. If they are constrained then inaccurate results will be obtained.
4. If THETA/MCID is blank, field 5 of the PSHELL continuation entry will be used. If this field is also blank, then THETA = 0.0 is assumed.
5. This element is identical to the CTRIA3 element.

(Continued)

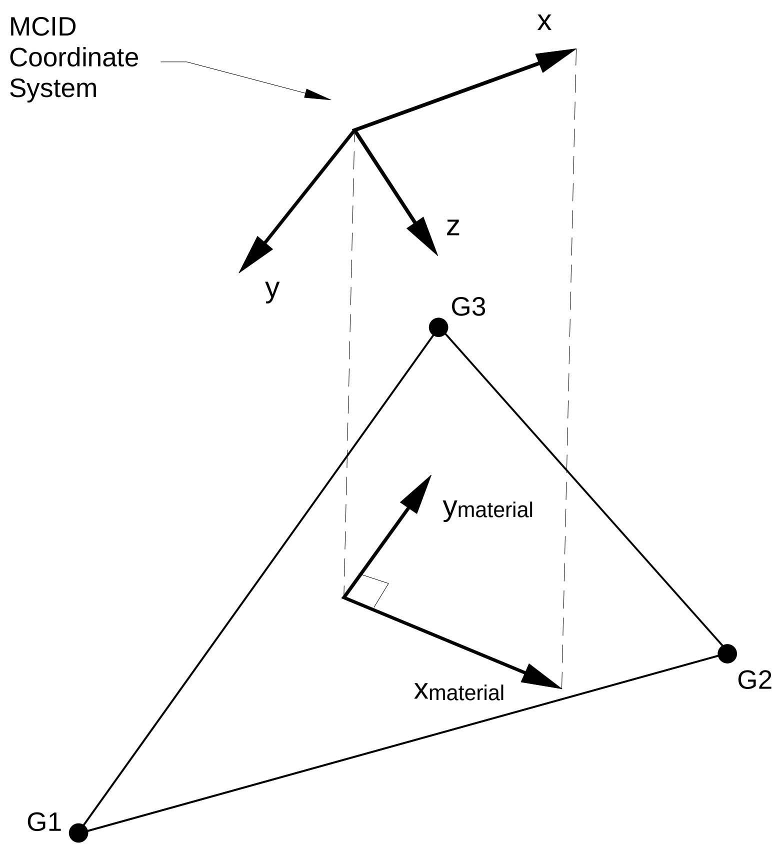


Figure 1. MCID Coordinate System Definition.

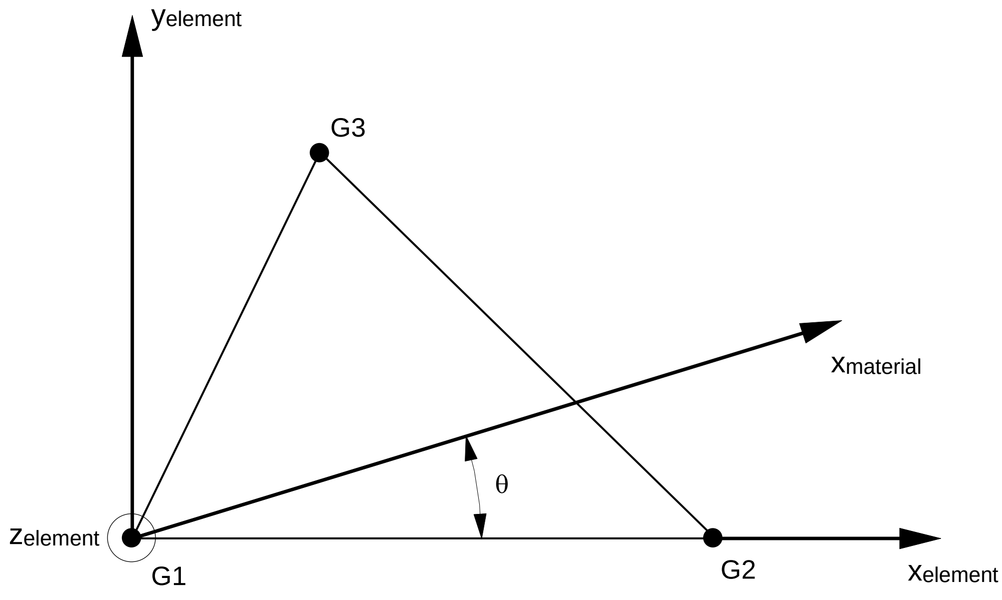


Figure 2. CTRIAR Element Geometry and Coordinate Systems.

(Continued)

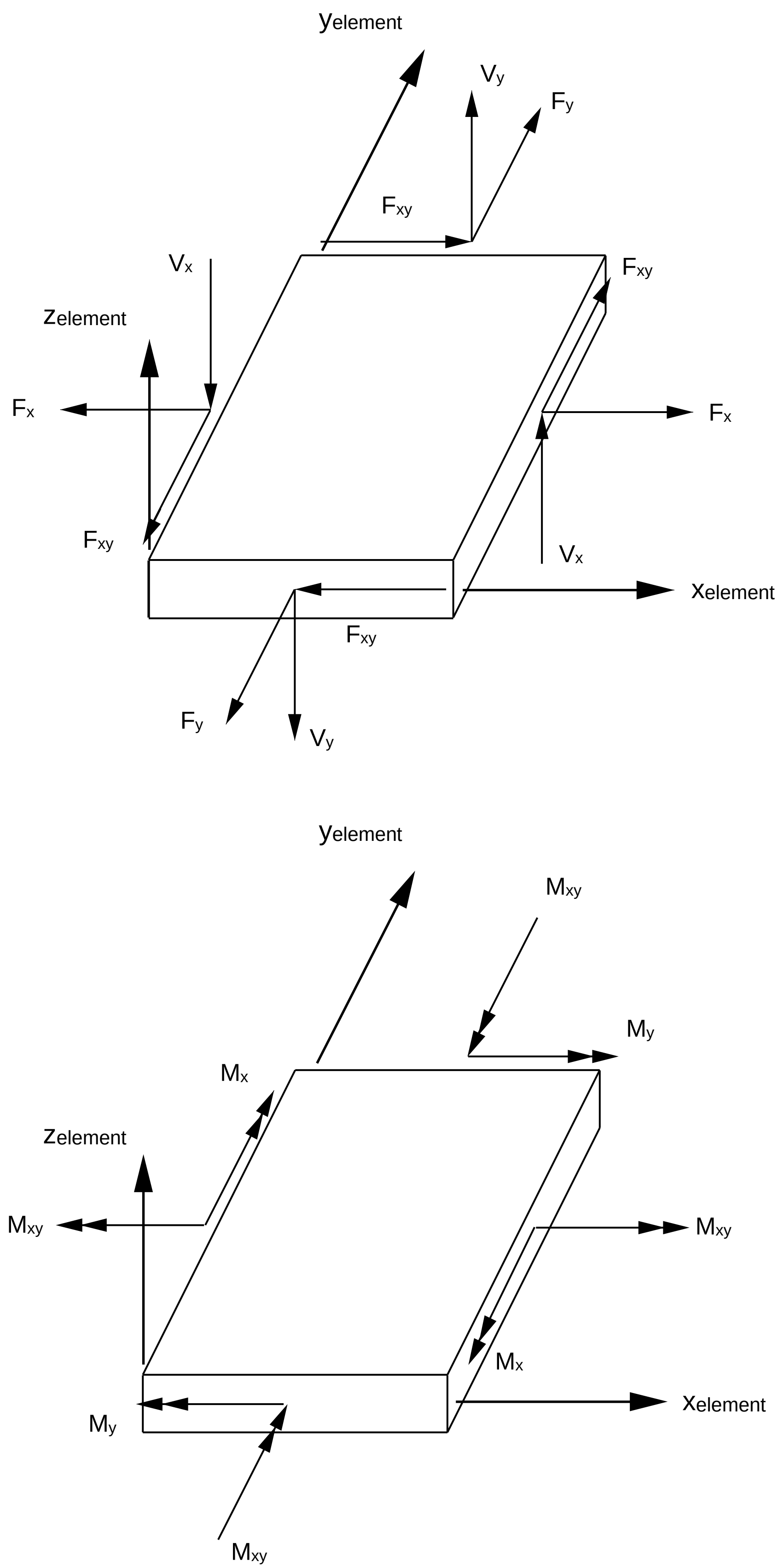


Figure 3. Forces and Moments in CTRIAR Elements.

(Continued)

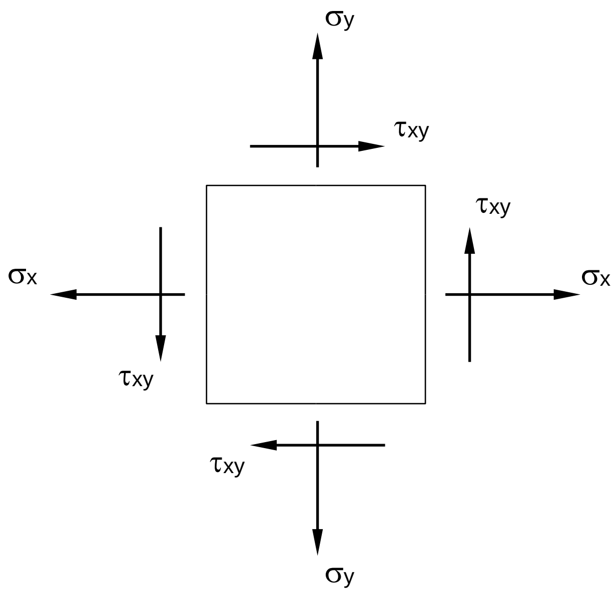


Figure 4. Stresses in CTRIAR Elements.

DLOAD — Dynamic Load Combination (Superposition)

Description: The DLOAD entry defines time-varying boundary conditions for transient response problems. The boundary conditions can be either force boundary conditions or kinematic boundary conditions. The DLOAD entry defines a linear combination of time-varying boundary conditions defined via TLOAD1 entries.

Format:

1	2	3	4	5	6	7	8	9	10
DLOAD	SID	S	S1	L1	S2	L2	S3	L3	
	S4	L4	- etc. -						

Example:

DLOAD	20	1.5	2.2	6	-3.6	7	6.0	10	
	-4.5	12							

Field	Definition	Type	Default
SID	Load set identification number.	Integer > 0	Required
S	Scale factor. This scale factor applies to all sets, Si, named on the DLOAD entry.	Real	Required
Si	Scale factors. The scale factor Si is applied only to the set Li.	Real	Required
Li	Set identification numbers for force boundary conditions or kinematic boundary conditions.	Integer > 0; SID ≠ Li	Required

Remarks:

1. The scale for set Li listed on the DLOAD entry is S*Si. The scale factor S is a “global” scale factor for the DLOAD entry. The scale factor S is applied to all sets Li.
2. Each Li must be unique from any other Li on the same entry.
3. Dynamic load sets must be selected in the Case Control Section with DLOAD = SID.
4. A DLOAD entry may not reference a set identification number defied by another DLOAD entry.
5. TLOAD1 sets may be combined only through the use of the DLOAD entry.
6. SID must be unique for all TLOAD1 entries.

ENDDATA

Bulk Data Delimiter

Description: The ENDDATA entry designates the end of the Bulk Data Section.

Format:

ENDDATA

Remarks:

1. ENDDATA is required.

FORCE Static Load

Description: The FORCE entry defines a force at a grid point by specifying a vector.

Format:

1	2	3	4	5	6	7	8	9	10
FORCE	SID	G	CID	F	N1	N2	N3		

Example:

FORCE	3	441	4	10.0	1.0	-1.0	0.0		

Field	Definition	Type	Default
SID	Load set identification number.	Integer > 0	Required
G	Grid point identification number.	Integer ≥ 0 or blank	0
CID	Coordinate system identification number.	Integer > 0	Required
F	Load vector scale factor.	Real	Required
N1, N2, N3	Load vector components of vector measured in the coordinate system defined by CID.	Real	Required; must have at least one nonzero component

Remarks:

1. The static load applied to grid point G is given by:

$$\vec{f} = F\vec{N}$$

   In the above equation, $\vec{N}$ is the vector defined in fields 6, 7 and 8.
2. Load sets can be selected in the Case Control Section (LOAD = SID). Load sets selected by a LOAD case control entry represent constant forces.
3. Load sets may be referenced by a TLOAD1 entry. Load sets selected by a TLOAD1 entry can be time-varying forces.
4. A CID of zero references the basic coordinate system.

FORCE1 — Static Load, Alternate Form 1

Description: The FORCE1 entry defines a static load at a grid point by specification of a value and two grid points that determine the direction.

Format:

1	2	3	4	5	6	7	8	9	10
FORCE1	SID	G	F	G1	G2				

Example:

FORCE1	3	141	-4.5	10	11				

Field	Definition	Type	Default
SID	Load set identification number.	Integer > 0	Required
G	Grid point identification number.	Integer > 0	Required
F	Load magnitude.	Real	Required
G1, G2	Grid point identification numbers.	Integer > 0; $G1 \neq G2$	Required

Remarks:

1. The static load applied to grid point G is given by:

$$\vec{f} = F\vec{n}$$

   In the above equation, $\vec{n}$ is a unit vector parallel to a vector for G1 to G2.
2. Load sets must be selected in the Case Control Section (LOAD = SID).

GRAV Gravity Vector

Description: The GRAV entry is used to define a gravity load for the structural model.

Format:

1	2	3	4	5	6	7	8	9	10
GRAV	SID	CID	G	N1	N2	N3			

Example:

GRAV	3	1	4.5	0.0	0.5	-1.0			

Field	Definition	Type	Default
SID	Load set identification number.	Integer > 0	Required
CID	Coordinate system identification number.	Integer ≥ -1 or blank	0
G	Gravity vector scale factor.	Real	Required
N1, N2, N3	Gravity vector components measured in coordinate system defined by CID.	Real	Required; must have at least one nonzero component

Remarks:

1. The static load applied to grid point G is given by:

$$\vec{g} = G\,\vec{n}\,f(x, y, z)$$

where $\vec{n}$ is the unit vector defined in fields 5, 6, and 7 and $f(x, y, z)$ is defined as the product of scale factors returned by tables defined in fields 2, 3, and 4 on the continuation entry.

2. A CID of zero references the basic coordinate system.
3. If CID = -1, the gravity vector components are in the local displacement coordinate system of the grid points.
4. Gravity loads may be combined with "simple loads" (e.g., FORCE, MOMENT). The SID on a GRAV entry may be the same as that on a simple load entry.
5. Load sets must be selected in the Case Control Section (LOAD = SID).

GRID Grid Point

Description: The GRID entry defines the location of a geometric grid point, the directions of its displacement, and its permanent single-point constraints.

Format:

1	2	3	4	5	6	7	8	9	10
GRID	ID	CP	X1	X2	X3	CD	PS		

Example:

GRID	3	1	4.5	1.0	7.5	2			

Field	Definition	Type	Default
ID	Grid point identification number.	Integer > 0	Required
CP	Identification number of coordinate system in which the location of the grid point is defined.	Integer ≥ 0 or blank	0
X1, X2, X3	Location of the grid point in coordinate system CP.	Real	Required
CD	Identification number of coordinate system in which the displacements, degrees of freedom, constraints, and solution vectors are all defined at the grid point.	Integer ≥ 0 or blank	0
PS	Permanent single-point constraints associated with grid point (any of the digits 1-6 with no imbedded blanks).	Integer ≥ 0 or blank	

Remarks:

1. All grid point identification numbers must be unique with respect to all other grid, scalar, and extra points.
2. The meaning of X1, X2 and X3 depend on the type of coordinate system, CP, as follows (see CORDi entry descriptions):

Type	X1	X2	X3
RECTANGULAR	X	Y	Z
CYLINDRICAL	R	θ (degrees)	Z
SPHERICAL	R	θ (degrees)	ϕ (degrees)

3. The collection of all CD coordinate systems defined on all GRID entries is called the global coordinate system. All degrees of freedom, constraints, and solution vectors are expressed in the global coordinate system.
4. A zero (or blank if the GRDSET entry is not specified) in the CP or CD fields refers to the basic coordinate system.

INITDIS — Initial Displacement Condition

Description: The INITDIS entry defines values for the initial translational displacement.

Format:

1	2	3	4	5	6	7	8	9	10
INITDIS	TID	SID	Dx	Dy	Dz	TYPE			

Example:

INITDIS	10	8	0	12.5	-5.0	GRID			
INITDIS	88	9	-100.	0	0	PART			

Field	Definition	Type	Default
TID	Identification number.	Integer > 0	Required
SID	Set identification number.	Integer > 0	Required
Dx	Component of initial displacement in the global x-direction.	Real	0.0
Dy	Component of initial displacement in the global y-direction.	Real	0.0
Dz	Component of initial displacement in the global z-direction.	Real	0.0
TYPE	Type of Set. Choices are: ALLGRID — All grid points in the mesh are given the initial displacement. For this case the SID is ignored. GRID — All grid points in the set defined by SID are given the initial displacement. ELEM — All grid points in the set of elements defined by SID are given the initial displacement. PART — The SID is interpreted as a property ID and all grid points in the elements that use this property ID are given the initial displacement.	Character	Required

Remarks:

1. Initial condition sets must be selected with the Case Control command IC = TID.
2. If no TIC set id selected in the Case Control Section, all initial conditions are assumed zero.
3. Initial conditions for coordinates not specified on TIC entries will be assumed zero.
4. If the SID references a grid point that is the reference node for a rigid body defined with an RBE2 entry and the PART specification is used for TYPE, the initial condition is applied to the reference node for the rigid body.

(Continued)

5. If the SID references the material identification number on a MATR1 entry (the MATR1 entry defines a rigid material) and the PART specification is used for TYPE, the initial condition is applied to the center of mass of the rigid material defined by the MATR1 entry.

INITVEL Initial Velocity Condition

Description: The INITVEL entry defines values for the initial translational velocity.

Format:

1	2	3	4	5	6	7	8	9	10
INITVEL	TID	SID	Vx	Vy	Vz	TYPE			

Example:

INITVEL	10	8	0	12.5	-5.0	GRID			
INITVEL	88	9	-100.	0	0	PART			

Field	Definition	Type	Default
TID	Identification number.	Integer > 0	Required
SID	Set identification number.	Integer > 0	Required
Vx	Component of initial velocity in the global x-direction.	Real	0.0
Vy	Component of initial velocity in the global y-direction.	Real	0.0
Vz	Component of initial velocity in the global z-direction.	Real	0.0
TYPE	Type of Set. Choices are:	Character	Required
	ALLGRID — All grid points in the mesh are given the initial velocity. For this case the SID is ignored.		
	GRID — All grid points in the set defined by SID are given the initial velocity.		
	ELEM — All grid points in the set of elements defined by SID are given the initial velocity.		
	PART — The SID is interpreted as a property ID and all grid points in the elements that use this property ID are given the initial velocity.		

Remarks:

1. Initial condition sets must be selected with the Case Control command IC = TID.
2. If no TIC set id selected in the Case Control Section, all initial conditions are assumed zero.
3. Initial conditions for coordinates not specified on TIC entries will be assumed zero.
4. If the SID references a grid point that is the reference node for a rigid body defined with an RBE2 entry and the PART specification is used for TYPE, the initial condition is applied to the reference node for the rigid body.

(Continued)

5. If the SID references the material identification number on a MATR1 entry (the MATR1 entry defines a rigid material) and the PART specification is used for TYPE, the initial condition is applied to the center of mass of the rigid material defined by the MATR1 entry.

LSEQ

Static Load Set Definition

Description: The LSEQ entry defines a sequence of static load sets used in transient response analysis.

Format:

1	2	3	4	5	6	7	8	9	10
LSEQ	SID	DAREA	LID						

Example:

LSEQ	109	100	1000						

Field	Definition	Type	Default
SID	Identification number of the LSEQ set.	Integer > 0	Required
DAREA	The DAREA set identification number assigned to this static load vector.	Integer > 0	Required
LID	Load set identification number of a set of static load entries (any entry that may be referenced by the LOAD Case Control command).	Integer > 0 or blank	See Remark 3

Remarks:

1. LSEQ will not be used unless selected in the Case Control Section with the LOADSET command.
2. A load vector will be created for each DAREA identification number referenced by a LSEQ entry.
3. LID and TID cannot both be blank.

MAT1 — Isotropic Material Property Definition

Description: The MAT1 entry defines the material properties for isotropic, elastic materials. In addition to isotropic, elastic constants, density and a coefficient of thermal expansion are also defined on this entry.

Format:

1	2	3	4	5	6	7	8	9	10
MAT1	MID	E	G	NU	RHO	A			

Example:

MAT1	13	1.+7		0.33	0.101				

Field	Definition	Type	Default
MID	Material identification number.	Integer $>$ 0	Required
E	Young's modulus.	Real $\geq$ 0.0 or blank	See Remarks 4, 5, and 6
G	Shear modulus.	Real $\geq$ 0.0 or blank	
NU	Poisson's ratio.	-1.0 $<$ Real $\leq$ 0.5 or blank	
RHO	Mass density.	Real > 0.0	See Remark 7
A	Thermal expansion coefficient.	Real or blank	0.0

Remarks:

1. The material identification number must be unique for all MATi entries.
2. Either E or G must be specified (i.e. nonblank).
3. If any one of E, G, or NU is blank, it will be computed to satisfy the identity E = 2(1 + NU)G; otherwise, values supplied by the user will be used.
4. If E and NU or G and NU are both blank, they will both be given the value 0.0.
5. Implausible data on one or more MAT1 entries will result in a warning message. Implausible data is defined as any of E $<$ 0.0, or G $<$ 0.0, or NU $>$ 0.5, or NU $<$ 0.0, or $|1 - E / [2(1+NU)G]| > 0.01$.
6. It is strongly recommended that only two of the three values E, G, and NU be input. The three values may be input independently on the MAT2 entry.
7. In Autodesk Explicit, RHO must be a nonzero value greater than zero. The mass density is required to calculate a wave speed for the material.

MAT8 Orthotropic Material Properties for Membrane and Shell

Description: The MAT8 entry defines the material properties for an orthotropic material for membrane and shell elements and for solid elements. In addition to the material properties, the MAT8 entry also lets the user specify various material failure options. The options on the MAT8 entry are quite extensive, and are outlined in more detail in the Remarks section for the MAT8 entry

Format:

1	2	3	4	5	6	7	8	9	10
MAT8	MID	E1	E2	NU12	G12			RHO	
	A1	A2	TREF	Xt	Xc	Yt	Yc	S	
		F12	STRN	CS	EC	GC	ALPHA0	SB	
	EF1	NUF12	MSMF	PNPT	PNPC				
	E3	NU23	NU31	E1RSF	E2RSF	G12RSF	G1ZRSF	G2ZRSF	

Example:

MAT8	101	90.+6	1.+7	0.3	3.+5			0.066	
	29.-6	1.1-6	175.0	1.+3	1.1+4	4.+2	2.+2	5.+3	
			1.0						

Field	Definition	Type	Default
MID	Material identification number. Referenced on a PSHELL or PCOMP entry only.	Integer > 0	Required
E1	Modulus of elasticity in longitudinal direction, also defined as the fiber direction or 1-direction.	Real ≠ 0.0	Required
E2	Modulus of elasticity in lateral direction, also defined as the matrix direction or 2-direction.	Real ≠ 0.0	Required
NU12	Poisson's ratio ($\varepsilon_2/\varepsilon_1$ for uniaxial loading in 1-direction). Note that $\upsilon_{21} = \varepsilon_2/\varepsilon_1$ for uniaxial loading in 2-direction is related to υ_{12}, E_1, and E_2 by the relation $\upsilon_{12} E_2 = \upsilon_{21} E_1$.	Real	Required
G12	In-plane shear modulus. See Remark 2.	Real ≥ 0.0 or blank	0.0
RHO	Mass density.	Real > 0.0	0.0
Ai	Thermal expansion coefficient in i-direction.	Real or blank	0.0
TREF	Reference temperature for the calculation of thermal loads. See Remarks 4 and 6.	Real or blank	0.0

(Continued)

Field	Definition	Type	Default
Xt, Xc	Allowable stresses or strains in tension and compression, respectively, in the longitudinal direction. Required if composite element failure index is desired. See Remark 3.	Real ≥ 0.0 or blank	Default value for Xc is Xt
Yt, Yc	Allowable stresses or strains in tension and compression, respectively, in the lateral direction. These parameters are required if composite element failure index is desired. See Remark 3.	Real ≥ 0.0 or blank	Default value for Yc is Yt
S	Allowable stress or strain for in-plane shear. See Remark 3.	Real ≥ 0.0 or blank	0.0
F12	Interaction term in the tensor polynomial theory of Tsai-Wu. This parameter is required if composite element failure index by Tsai-Wu theory is desired and if value of F12 is different from 0.0. See Remark 7.	Real	0.0
STRN	For the maximum strain theory only (see STRN in PCOMP entry). Indicates whether Xt, Xc, Yt, Yc, and S are stress or strain allowables.	Real = 1.0 for strain allowable	Blank for stress allowable
CS	Honeycomb sandwich core cell size. The CS parameter is required if the material defines the core of a honeycomb sandwich and a dimpling stability index is desired (LAM = HCS on the PCOMP entry).	Real ≥ 0.0 or blank	0.0
EC	Honeycomb sandwich core Young's modulus. The EC parameter is used for a stability index analysis.	Real ≥ 0.0 or blank	See Remark 12
GC	Honeycomb sandwich core shear modulus. The GC parameter is used for a stability index analysis.	Real ≥ 0.0 or blank	See Remark 12
ALPHA0	Fracture angle for uniaxial transverse compression in degrees. The ALPHA0 parameter is used in the NASA LaRC02 failure theory only (see LARC02 in PCOMP entry).	0.0 < Real < 90.0	53.0
SB	Allowable shear stress of the composite laminate bonding material (allowable inter-laminar shear stress). See Remark 9.	Real > 0.0	PCOMP entry value
EF1	Modulus of elasticity of fiber. The EF1 parameter is used in the Puck PCP failure theory only (see PUCK in PCOMP entry).	Real > 0.0 or blank	E1/0.6
NUF12	Poisson's ratio of fiber. The NUF12 parameter is used in the Puck PCP failure theory only (see PUCK in PCOMP entry).	Real ≥ 0.0 or blank	0.3
MSMF	Mean stress magnification factor. The MSMF parameter is used in the Puck PCP failure theory only (see PUCK in PCOMP entry). See Remark 10.	Real ≥ 0.0 or blank	1.1
PNPT	Failure-envelope slope parameter for transverse tension. The PNPT parameter is used in the Puck PCP failure theory only (see PUCK in PCOMP entry). See Remark 10.	Real ≥ 0.0 or blank	0.35

(Continued)

Field	Definition	Type	Default
PNPC	Failure-envelope slope parameter for transverse compression. The PNPC parameter is used in the Puck PCP failure theory only (see PUCK in PCOMP entry). See Remark 10.	Real $\geq$ 0.0 or blank	0.3
E3	Modulus of elasticity in thickness direction for shells and membranes. Modulus of elasticity in 3-direction for solids.	Real $\neq$ 0.0	
NU23	($\varepsilon_3/\varepsilon_2$ for uniaxial loading in 2-direction). Note that $\upsilon_{32} = \varepsilon_3/\varepsilon_2$ for uniaxial loading in 3-direction is related to υ_{23}, E_2, and E_3 by the relation $\upsilon_{23} E_3 = \upsilon_{32} E_2$.	Real $\neq$ 0.0	
NU31	($\varepsilon_1/\varepsilon_3$ for uniaxial loading in 3-direction). Note that $\upsilon_{13} = \varepsilon_1/\varepsilon_3$ for uniaxial loading in 1-direction is related to υ_{31}, E_1, and E_3 by the relation $\upsilon_{31} E_1 = \upsilon_{13} E_3$.	Real $\neq$ 0.0	
E1RSF	Longitudinal modulus of elasticity reduction scale factor for nonlinear composite Progressive Ply Failure Analysis (PPFA). See Remark 11.	0.0 $\leq$ Real $\leq$ 1.0	0.04
E2RSF	Lateral modulus of elasticity reduction scale factor for nonlinear composite Progressive Ply Failure Analysis (PPFA). See Remark 11.	0.0 $\leq$ Real $\leq$ 1.0	0.04
G12RSF	In-plane shear modulus reduction scale factor for nonlinear composite Progressive Ply Failure Analysis (PPFA). See Remark 11.	0.0 $\leq$ Real $\leq$ 1.0	0.2
G1ZRSF	Transverse shear modulus reduction factor in 1-Z plane for non-linear composite Progressive Ply Failure Analysis (PPFA).		
G2ZRSF	Transverse shear modulus reduction factor in 2-Z plane for non-linear composite Progressive Ply Failure Analysis (PPFA).		

Remarks:

1. The material identification number must be unique for all MATi entries.
2. The transverse shear moduli G1Z and G2Z are computed automatically from the in-plane shear modulus G12.
3. Xt, Yt, and S are required for composite element failure calculations when requested in the FT field of the PCOMP entry. Xc and Yc are also used but not required.
4. MAT8 materials may be made temperature dependent by use of the MATT8 entry.
5. The mass density, RHO, will be used to automatically compute mass for all structural elements.
6. TREF is used only as the reference temperature for the calculation of thermal.
7. The interaction term F12 is experimentally determined from test specimens under biaxial loading. The fact that F12 must be determined experimentally, plus a stability constraint on F12, makes the Tsai-Wu theory difficult to use. It is recommended that F12 be set to zero. The stability criterion is: $\left(\frac{1}{x_t x_c}\right)\left(\frac{1}{y_t y_c}\right) - F_{12}^2 > 0$

(Continued)

8. The default value for ALPHA0 has been found experimentally and is typical for fiber reinforced polymer laminates. See the Nastran User's Manual, Reference 5 for additional information.
9. The allowable shear stress value SB corresponds to the top surface of the ply. The default value for SB is defined in the SB field of the PCOMP entry and will be used when this field is blank.
10. The default values for MSMF, PNPT, and PNPC are for carbon fibers. See the Nastran User's Manual, Reference 13 and the table below for additional materials.

Variable	**Carbon Fiber**	**Glass Fiber**
MSMF	1.10	1.30
PNPT	0.35	0.30
PNPC	0.30	0.25

11. Recommended values for E1RSF, E2RSF, and G12RSF are shown in the table below.

Variable	**Recommended Value**
E1RSF	0.04
E2RSF	0.04
G12RSF	0.20

MAT12 Solid Element Orthotropic Material Property Definition

Description: The MAT12 entry defines the material properties for an orthotropic material for solid elements.

Format:

1	2	3	4	5	6	7	8	9	10
MAT12	MID	E1	E2	E3	NU12	NU23	NU31	RHO	
	G12	G23	G31	A1	A2	A3	TREF	GE	

Example:

MAT12	105	2.+7	2.+7	1.+4	0.1	0.0	0.0	0.066	
	4.5+5	2.5+5	2.5+5	1.1-6	1.1-6	0.0	70.0		

Field	Definition	Type	Default
MID	Material identification number. Referenced on a PSHELL or PCOMP entry only.	Integer > 0	Required
E1	Modulus of elasticity in longitudinal direction, also defined as the fiber direction or 1-direction.	Real > 0.0	Required
E2	Modulus of elasticity in lateral direction, also defined as the matrix direction or 2-direction.	Real > 0.0	Required
E3	Modulus of elasticity in thickness direction, also defined as the matrix direction or 3-direction.	Real > 0.0	Required
NU12	Poisson's ratio ($-\varepsilon_2/\varepsilon_1$ for uniaxial loading in 1-direction). Note that $\upsilon_{21} = -\varepsilon_1/\varepsilon_2$ for uniaxial loading in 2-direction is related to υ_{12}, E_1, and E_2 by the relation $\upsilon_{12} E_2 = \upsilon_{21} E_1$. See Remark 3.	Real	Required
NU23	Poisson's ratio ($-\varepsilon_3/\varepsilon_2$ for uniaxial loading in 2-direction). Note that $\upsilon_{32} = -\varepsilon_2/\varepsilon_3$ for uniaxial loading in 3-direction is related to υ_{23}, E_2, and E_3 by the relation $\upsilon_{23} E_3 = \upsilon_{32} E_2$. See Remark 3.	Real	Required
NU31	Poisson's ratio ($-\varepsilon_1/\varepsilon_3$ for uniaxial loading in 3-direction). Note that $\upsilon_{13} = \varepsilon_3/\varepsilon_1$ for uniaxial loading in 1-direction is related to υ_{31}, E_1, and E_3 by the relation $\upsilon_{31} E_1 = \upsilon_{13} E_3$. See Remark 3.	Real	Required
RHO	Mass density.	Real or blank	0.0
G12	Shear modulus in plane 1-2.	Real > 0.0	Required
G23	Shear modulus in plane 2-3.	Real > 0.0	Required
G31	Shear modulus in plane 3-1.	Real > 0.0	Required

(Continued)

Field	Definition	Type	Default
Ai	Thermal expansion coefficient in i-direction.	Real or blank	0.0
TREF	Reference temperature for the calculation of thermal loads.	Real or blank	0.0
GE	Structural element damping coefficient. See Remarks 9, 10, and 12.	Real or blank	0.0

Remarks:

1. The material identification number must be unique for all MATi entries.
2. An approximate value for G23 and G31 is the in-plane shear modulus G12. If test data is not available to accurately determine G23 and G31, the value to G12 may be supplied for G23 and G31.
3. Material stability requires that

$$E_i > \upsilon_{ij}^2 E_j$$

$$1 - \upsilon_{12}\upsilon_{21} - \upsilon_{23}\upsilon_{32} - \upsilon_{31}\upsilon_{13} - 2\upsilon_{21}\upsilon_{32}\upsilon_{13} > 0$$

   If either condition is not met a warning message will be issued.

4. It may be difficult to find all nine orthotropic constants. In some practical problems, the material properties may be reduced to normal anisotropy in which the material is isotropic in a plane (i.e., plane 1-2) and has different properties in the direction normal to this plane. In the plane of isotropy, the properties are reduced to

$$E_1 = E_2 = E_p$$

$$\upsilon_{31} = \upsilon_{32} = \upsilon_{np}$$

$$\upsilon_{13} = \upsilon_{23} = \upsilon_{pn}$$

$$G_{13} = G_{23} = G_n$$

   with $\upsilon_{np}/E_n = \upsilon_{pn}/E_p$ and $G_p = \dfrac{E_p}{2(1+\upsilon_p)}$

   There are five independent material constants for normal anisotropy (i.e., E_p, E_n, υ_p, υ_{np}, and G_n). In case the material has a planar anisotropy, in which the material is orthotropic only in a plane, the elastic constants are reduced to seven (i.e., E_1, E_2, E_3, υ_{12}, G_{12}, G_{23}, and G_{31}).

5. The mass density, RHO, will be used to automatically compute mass for all structural elements.
6. Weight density may be used in field 9 if the value 1/g is entered on the PARAM, WTMASS entry, where g is the acceleration of gravity.
7. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0, by 2.0.
8. TREF and GE are ignored if the MAT12 entry is referenced by a PCOMP entry.
9. TREF is used only as the reference temperature for the calculation of thermal loads in linear solutions. If TEMPERATURE(INITIAL) is specified, TREF will be ignored.

MATB Brittle Failure Material Model

Description: The MATB entry specifies a brittle failure material. Tensile cracks form in the orthogonal principal strain directions based upon the tensile strength provided. In compression, the model can behave as an elastic material, an elastic/perfectly plastic material, or as an elastic/plastic strain softening material.

This model can also be used to specify a “no tension” material by setting the tensile strength to zero.

Format:

1	2	3	4	5	6	7	8	9	10
MATB	MID	E	G	NU	RHO	A			
	CMODEL	TS	CS	TMULT	CMULT				

Example:

MATB	35	5.E6		.18	1.e-4				
	1	100.	2000.	3	9				

Field	Definition	Type	Default
MID	Material identification number.	Integer > 0	Required
E	Young’s modulus.	Real ≥ 0.0 or blank	See Remark 3
G	Shear modulus.	Real ≥ 0.0 or blank	See Remark 3
NU	Poisson’s ratio.	-1.0 < Real ≤ 0.5 or blank	See Remark 3
RHO	Mass density.	Real ≥ 0.0	Required
A	Thermal expansion coefficient.	Real or blank	0.0
CMODEL	Compression behavior: 1 = Elastic Compression 2 = Elastic/Perfectly Plastic Compression 3 = Elastic/Plastic Strain Softening Compression	Integer > 0	1, See Remark 5
TS	Tensile Strength	Real ≥ 0.0	Required, See Remark 4
CS	Compressive Strength	Real ≥ 0.0 or blank	1.E30, See Remark 6

(Continued)

Field	Definition	Type	Default
TMULT	Tensile deletion strain multiplier.	Integer ≥ 1	1000, See Remarks 7 and 9
CMULT	Compression deletion strain multiplier.	Integer ≥ 1	1000, See Remarks 8 and 9

Remarks:

1. The material identification number must be unique for all MATi entries.
2. Either E or G must be specified (i.e. nonblank).
3. If any one of E, G, or NU is blank, it will be computed to satisfy the identity E = 2(1 + NU)G; otherwise, values supplied by the user will be used.
4. The tensile behavior of the material is purely elastic up to the tensile strength value. Individual cracks can form in orthogonal principal strain directions. The stress strain curve in tension strain softens after cracking occurs as shown in Figure 1.

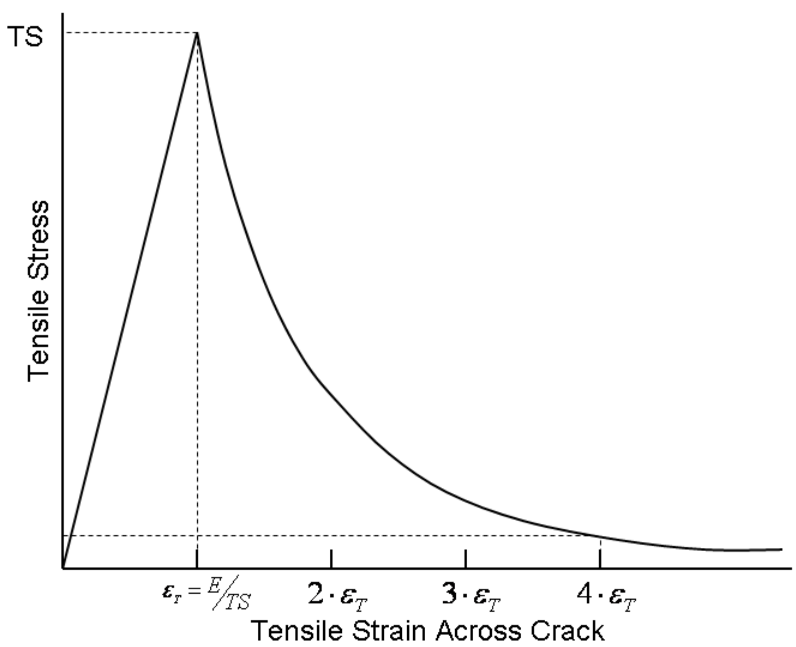


Figure 1. Tensile Stress/Strain curve.

(Continued)

5. The compressive behavior of the material may either be Elastic (Hooke's law), Elastic/Perfectly-Plastic (Figure 2) or Elastic/Plastic Strain Softening (Figure 3).

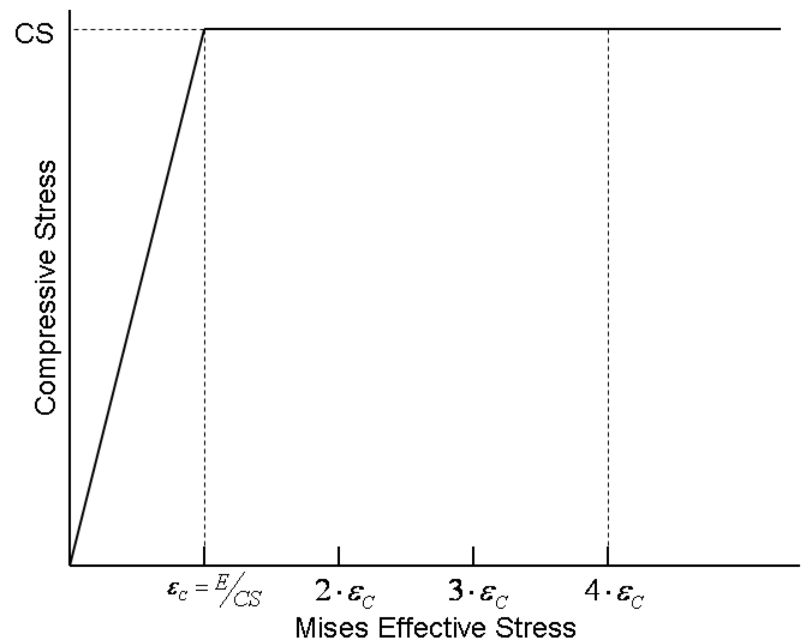


Figure 2. Elastic/Perfectly Plastic Compression Model (CMODEL = 2).

(Continued)

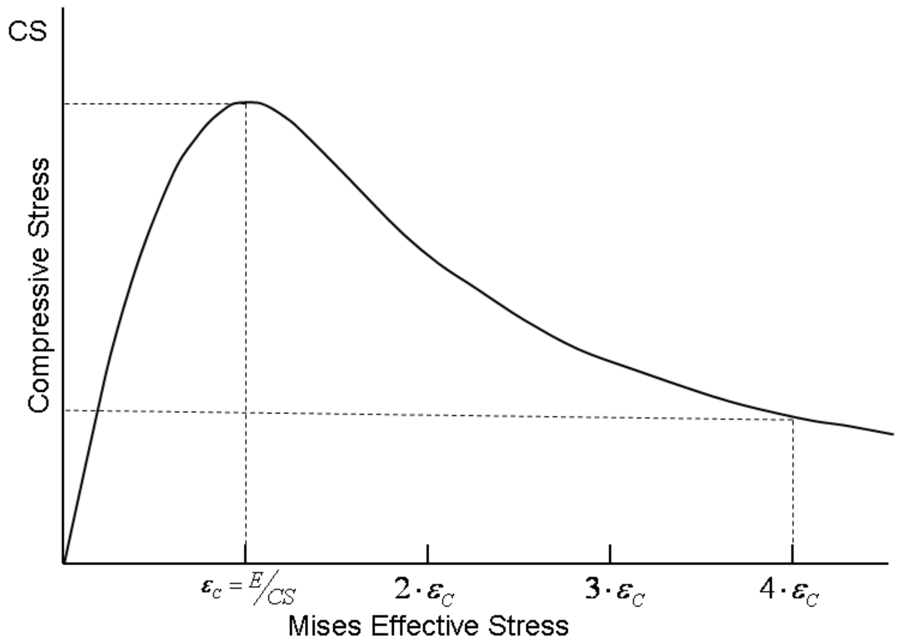


Figure 3. Elastic/Plastic Strain Softening Model (CMODEL = 3).

6. A compressive strength value is only required if CMODEL = 2 or 3. The default value of 1.E30 implies infinite strength.
7. The tensile strain at which the material point is deleted from the mesh (if the BRITTLE deletion criteria was specified for this part PID) is determined from this multiple of the failure strain. In Figure 1, the deletion strain is shown for TMULT = 4. The default value of 1000 implies no element deletion in the material.
8. The compressive strain at which the material point is deleted from the mesh (if the BRITTLE deletion criteria was specified for this part PID) is determined from this multiple of the failure strain. In Figures 2 and 3, the deletion strain is shown for CMULT = 4. The default value of 1000 implies no deletion in the material.
9. The deletion strains defined by TMULT and CMULT have no effect unless the BRITTLE element DELETION option is specified for this part PID.

MATC — Concrete Material Model

Description: The MATC entry sets up a concrete material model. This entry is used if a MAT1 entry is specified with the same MID as that on the MATC entry.

Format:

1	2	3	4	5	6	7	8	9	10
MATC	MID	FPC	CYM	ARATIO	SI				

Example:

MATC	25	5000.							

MATC	25	28.E6	OFF	6	SI				

Field	Definition	Type	Default
MID	Identification number of a MAT1 entry.	Integer > 0	Required
FPC	Compressive strength of concrete (f_c'). See Remark 1.	Real	Required
CYM	Flag indicating whether to compute Young's Modulus from the correlation models for the compressive strength or use the value supplied on the MAT1 entry. If Young's modulus is computed, then Poisson's Ratio is set to a value of 0.18.	Character	TRUE
ARATIO	Fraction energy ratio. This value controls the strain softening curve after tensile failure. The higher the value the more the ductility in the tensile regime. The default value is highly recommended.	Real > 0	8.
SI	Flag that indicates that the units for the problem are SI units. Set this value "SI" to have the code convert the units correctly. See Remark 2.	Character	ENGLISH

Remarks:

1. The only required parameter for the concrete model is f_c' . Note that the units must be consistent with the SI flag given in field 6.
2. If SI units are used, care must be taken that all model dimensions are in SI units.

MATCF Crushable Foam Material Model

Description: The MATCF entry specifies a crushable foam material where the volumetric and deviatoric response of the material can be specified with independent plasticity models. This model may only be used with elements that use the PSOLID property type.

The pressure versus volumetric strain behavior can be defined using a piecewise linear curve where volumetric plasticity and volumetric bulking can be specified in the compression regime. The volumetric tensile behavior is elastic and defines a volumetric fracture pressure that defines a "no tension" material when the volumetric strain exceeds the P_{fr} value determined from the input parameters.

The deviatoric plasticity is pressure dependent and may be defined with a linear dependence on pressure or a quadratic dependence on pressure. When a linear dependence is used this model is similar to a classical Drucker-Prager material model.

Format:

1	2	3	4	5	6	7	8	9	10
MATCF	MID	E	G	NU	RHO	A			
	MODEL	A0	SLOPE	PS	YS				
	EV0	P0	EV1	P1	EV2	P2			
	ENDT								

Example: Low density foam (1 pcf) Linear dependence of yield on pressure.

MATCF	35	5.E5		0.1	1.498E-5				
	1	100.	10.						
	0.0	0.0	0.001	200.	0.003	200.	0.005	400.	
	0.007	800	0.009	1800.	0.010	3800.	ENDT		

Field	Definition	Type	Default
MID	Material identification number.	Integer > 0	Required, See Remarks 1 and 2
E	Young's modulus.	Real ≥ 0.0 or blank	See Remarks 3 and 4
G	Shear modulus.	Real ≥ 0.0 or blank	See Remarks 3 and 4
NU	Poisson's ratio.	-1.0 < Real ≤ 0.5 or blank	See Remark 4
RHO	Mass density.	Real ≥ 0.0	Required
A	Thermal expansion coefficient.	Real or blank	0.0

(Continued)

Field	Definition	Type	Default
MODEL	Yield Stress Dependence on Pressure: 1 = Linear 2 = Quadratic 3 = Elastic/Plastic Strain Softening Compression	Integer > 0	1, See Remark 5
A0	Yield Stress at Zero Pressure	Real ≥ 0.0	Required, See Remarks 5 and 6
SLOPE	Slope of Yield Stress vs Pressure Curve	Real ≥ 0.0 or blank	0.0, See Remarks 5 and 6
PS	Pressure at which SLOPE is specified (ignored if MODEL=1)	Real ≥ 0.0 or blank	0.0, See Remark 6
EVn	Volumetric Strain	Real ≥ 0.	See Remark 7
Pn	Pressure Value	Real ≥ 0.	See Remark 7

Remarks:

1. The material identification number must be unique for all MATi entries.
2. This material may only be used with Solid element types.
3. Either E or G must be specified (i.e. nonblank).
4. If any one of E, G, or NU is blank, it will be computed to satisfy the identity E = 2(1 + NU)G; otherwise, values supplied by the user will be used.
5. The deviatoric yield stress is pressure dependent and the dependency can be either linear or quadratic. For the linear case (MODEL=1), shown in Figure 1, the user specifies the yield at zero pressure, A0, and a constant slope for the yield stress as a function of pressure, SLOPE. This defines a conical shape of the yield stress versus pressure surface as shown in Figure1. For the quadratic case (MODEL=2), shown in Figure 2, the user specifies A0, SLOPE, the pressure as the specified slope, PS, and the yield at the specified slope, YS. The coefficients A1 and A2 of the quadratic dependency are automatically determined from the user supplied parameters. The value of the tensile volumetric fracture pressure is determined such that the quadratic curve passes through the user supplied value of A0 and matches the SLOPE provided at the pressure (PS,YS).

(Continued)

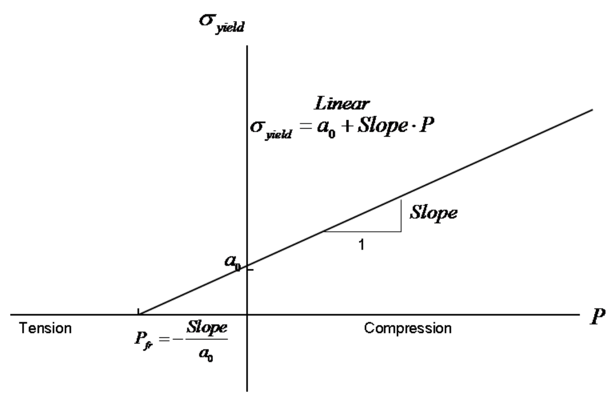


Figure 1. Linear Dependence of Yield Stress on Pressure (MODEL=1).

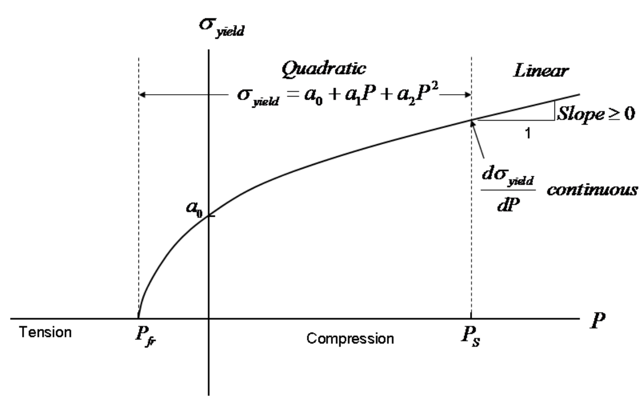


Figure 2. Quadratic Dependence of Yield Stress on Pressure (MODEL=2).

6. A value of zero is allowable for the SLOPE for the linear case (CMODEL=1). This specifies a constant yield stress, independent of pressure as shown in Figure 3. In this case, the tensile volumetric fracture pressure is assumed to be infinite (i.e. no tensile failure occurs).

(Continued)

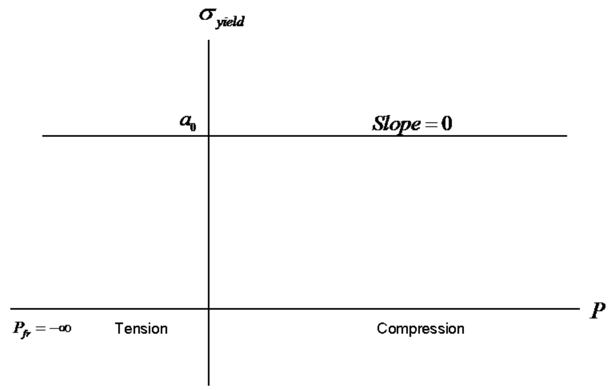


Figure 3. No Pressure Dependence of Yield Stress (MODEL = 1).

7. The value of the pressure at which the slope is defined (PS) may be zero. In this case, only the tensile range of pressures uses the quadratic form for the yield stress dependence as shown in Figure 4.

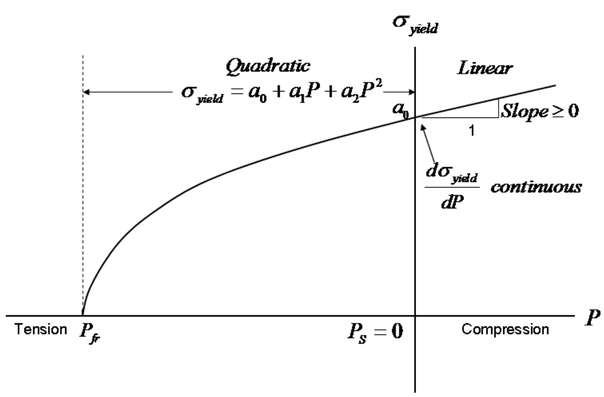


Figure 4. Quadratic Case With PS Value of Zero (MODEL = 2).

8. The EVn, Pn points define the piecewise linear behavior of the pressure versus volumetric strain in compression as shown in Figure 5. At least 2 points must be provided, in which case a linear response in compression is defined. The first point (EV0, PV0) must be defined at zero strain and pressure. The tensile volumetric response is purely elastic up to the point of tensile volumetric fracture defined by P_{fr}, which is determined from the deviatoric coefficients as described in Remark 1.

(Continued)

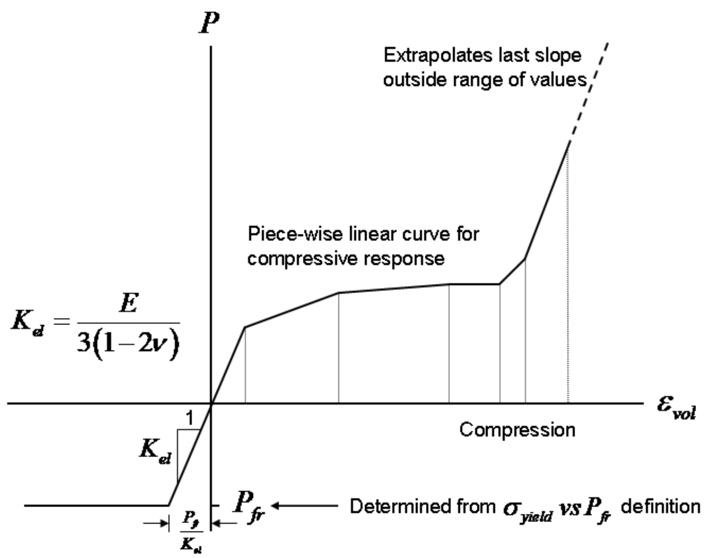


Figure 5. Pressure Versus Volumetric Response Curve.

MATHP — Hyperelastic Material Properties

Description: Defines material properties for use in fully nonlinear (i.e., large strain and large rotation) hyperelastic analysis of rubber-like materials (elastomers) for isoparametric solid elements.

Format:

1	2	3	4	5	6	7	8	9	10
MATHP	MID	A10	A01	D1	RHO	AV	TREF	GE	
		NA	ND						
	A20	A11	A02	D2					
	A30	A21	A12	A03	D3				
	A40	A31	A22	A13	A04	D4			
	A50	A41	A32	A23	A14	A05	D5		
	TAB1	TAB2	TAB3					TABD	

Example:

MATHP	100	153.8	38.5	2.+5					

Field	Contents	Type	Default
MID	Material identification number.	Integer > 0	Required
Aij	Material constants related to distortional deformation.	Real	0.0
Di	Material constants related to volumetric deformation.	Real ≥ 0	10^3∗(A10 + A01) for D1. 0.0 for D2 through D5
RHO	Mass density in original configuration.	Real	0.0
AV	Volumetric coefficient of thermal expansion.	Real	0.0
TREF	Reference temperature for the calculation of thermal loads.	Real	0.0
GE	Structural element damping coefficient. See Remarks 7 and 9.	Real	0.0
NA	Order of the distortional strain energy polynomial function.	0 < Integer ≤ 5	1
ND	Order of the volumetric strain energy polynomial function.	0 < Integer ≤ 5	1

(Continued)

Field	Contents	Type	Default
TAB1	Table identification number of TABLES1 entry that contains simple tension/compression data to be used in the estimation of the material constants Aij. xi values in the TABLES1 entry must be stretch ratios ℓ/ℓ_0 and yi values must be values of the engineering stress F/A_0. Stresses are negative for compression and positive for tension. If this convention is not followed the solution may fail to converge.	Integer > 0 or blank	
TAB2	Table identification number of TABLES1 entry that contains equibiaxial tension data to be used in the estimation of the material constants Aij. xi values in the TABLES1 entry must be stretch ratios ℓ/ℓ_0. yi values must be values of the engineering stress F/A_0. ℓ is the current length, F is the current force, ℓ_0 is the initial length and A_0 is the cross-sectional area. In the case of pressure of a spherical membrane, the engineering stress is given by $Pr_0\lambda^2/2t_0$ where P is the current value of the pressure and r_0, t_0 is the initial radius and thickness.	Integer > 0 or blank	
TAB3	Table identification number of TABLES1 entry that contains simple shear data to be used in the estimation of the material constants Aij. xi values in the TABLES1 entry must be values of the shear tangent γ and yi values must be values of the engineering stress F/A_0.	Integer > 0 or blank	
TABD	Table identification number of TABLES1 entry that contains pure volumetric compression data to be used in the estimation of the material constants Di. xi values in the TABLES1 entry must be values of the volume ratio $J = \lambda^3$ where $\lambda = \ell/\ell_0$ is the stretch ratio in all three directions; yi values must be values of the pressure, assumed positive in compression.	Integer > 0 or blank	

(Continued)

Remarks:

1. The generalized Mooney-Rivlin strain energy may be expressed as follows:

$$U(J,\bar{I}_1,\bar{I}_2)=\sum_{i+j=1}^{NA}Aij(\bar{I}_1-3)^i(\bar{I}_2-3)^j+\sum_{i=1}^{ND}Di(J-1-AV(T-T_0))^{2i}$$

   where $\bar{I}_1$ and $\bar{I}_2$ are the first and second distortional strain invariants, respectively; $J = det\,F$ is the determinate of the deformation gradient; and 2D1 = *K* and 2(A10 + A01) = *G* at small strains, in which *K* is the bulk modulus. The model reduces to a Mooney-Rivlin material if NA = 1 and to a Neo-Hookean material if NA = 1 and A01 = 0.0 (See Remark 2). For Neo-Hookean or Mooney-Rivlin materials no continuation entry is required. T is the current temperature and T_0 is the initial temperature.

2. Hyperelastic materials show a fully incompressible or nearly incompressible behavior. Full incompressibility is not presently available, while nearly incompressible behavior can be simulated using a large value of D1.

3. Aij and Di are obtained from least squares fitting of experimental data. One or more of four experiments (TAB1 to TAB4) may be used to obtain Aij. Di may be obtained from pure volumetric compression data (TABD). If all TAB1 through TAB4 are blank, Aij must be specified by the user. Parameter estimation, specified through any of the TABLES1 entries, supersedes the manual input of the parameters.

4. If ND = 1 and a nonzero value of D1 is provided or is obtained from experimental data in TABD, then the parameter estimation of the material constants Aij takes compressibility into account in the cases of simple tension/compression, equibiaxial tension, and general biaxial deformation. Otherwise, full incompressibility is assumed in estimation the material constants.

5. The mass density, RHO, will be used to automatically compute mass for all structural elements.

6. Weight density may be used in field 9 if the value 1/g is entered on the PARAM, WTMASS entry, where g is the acceleration of gravity.

7. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0, by 2.0.

8. TREF is used only as the reference temperature for the calculation of thermal loads in linear solutions. If TEMPERATURE(INITIAL) is specified, TREF will be ignored.

9. If PARAM, W4 is not specified, GE is ignored in transient response analysis. (See Section 5, *Parameters*, for more information on W4).

MATHP1 Hyperelastic Material Properties, Alternate Form

Description: Defines material properties for use in fully nonlinear (i.e., large strain and large rotation) hyperelastic analysis of rubber-like materials (elastomers) for isoparametric solid elements.

Format:

1	2	3	4	5	6	7	8	9	10
MATHP1	MID	MU1	ALPHA1	D1	RHO	AV	TREF	GE	
		NA	ND						
	MU2	ALPHA2	D2	MU3	ALPHA3	D3			
	D4								

Example:

MATHP1	100	0.3245	2.0	1.45+4					
		2	1						
	-0.2345	-2.0							

Field	Contents	Type	Default
MID	Material identification number.	Integer > 0	Required
MUi	Shear moduli related to distortional deformation.	Real	0.0
ALPHAi	Exponents related to distortional deformation.	Real	0.0
Di	Material constants related to volumetric deformation.	Real ≥ 0	See Remark 2
RHO	Mass density in original configuration.	Real	0.0
AV	Volumetric coefficient of thermal expansion.	Real	0.0
TREF	Reference temperature for the calculation of thermal loads.	Real	0.0
GE	Structural element damping coefficient. See Remarks 6 and 8.	Real	0.0
NA	Order of the distortional strain energy polynomial function.	0 < Integer ≤ 3	1
ND	Order of the volumetric strain energy polynomial function.	0 < Integer ≤ 4	1

(Continued)

Remarks:

1. The generalized Ogden strain energy may be expressed as follows:

$$U(\lambda_1,\lambda_2,\lambda_3,J)=\sum_{i=1}^{NA}\frac{\mu_i}{\alpha_i}\left[(\lambda_1)^{\alpha_i}+(\lambda_2)^{\alpha_i}+(\lambda_3)^{\alpha_i}-3\right]+\sum_{i=1}^{ND}Di\left(J-1-AV(T-T_0)\right)^{2i}$$

   where λ_1, λ_2 and λ_3 are principal stretches; $J = det\,F$ is the determinate of the deformation gradient; and 2D1 = *K* at small strains, where *K* is the bulk modulus. T is the current temperature and T_0 is the initial temperature.

2. The default for D1 is $\frac{1}{4}\left(\sum_{i=1}^{NA}\mu_i\alpha_i\right)*10^3$. The default for D2 through D4 is zero.
3. Hyperelastic materials show a fully incompressible or nearly incompressible behavior. Full incompressibility is not presently available, while nearly incompressible behavior can be simulated using a large value of D1.
4. The mass density, RHO, will be used to automatically compute mass for all structural elements.
5. Weight density may be used in field 9 if the value 1/g is entered on the PARAM, WTMASS entry, where g is the acceleration of gravity.
6. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0, by 2.0.
7. TREF is used only as the reference temperature for the calculation of thermal loads in linear solutions. If TEMPERATURE(INITIAL) is specified, TREF will be ignored.
8. If PARAM, W4 is not specified, GE is ignored in transient response analysis. (See Section 5, *Parameters*, for more information on W4).

(Continued)

MATHPF — Hyperfoam Material Properties

Description: Defines material properties for use in elastometric foams.

Format:

1	2	3	4	5	6	7	8	9	10
MATHPF	MID	MU1	ALPHA1	NU1	RHO	AV	TREF	GE	
	NA								
	MU2	ALPHA2	NU2	MU3	ALPHA3	MU3			
	MU4	ALPHA4	NU4	MU5	ALPHA5	MU5			
	MU6	ALPHA6	NU6						
	TAB1	TAB2	TAB3					TABD	

Example:

MATHPF	100	160.	2.	.48		.01			
	2								
	40.	-.2	0.						

Field	Contents	Type	Default
MID	Material identification number.	Integer > 0	Required
MUi	Shear moduli related to distortional deformation.	Real	0.0 See Remark 2
ALPHAi	Exponents related to distortional deformation.	Real	0.0 See Remark 2
NUi	Material constants related to volumetric deformation.	Real ≥ 0	0.0 See Remark 2
RHO	Mass density in original configuration.	Real	0.0
AV	Volumetric coefficient of thermal expansion.	Real	0.0
TREF	Reference temperature for the calculation of thermal loads.	Real	0.0
GE	Structural element damping coefficient. See Remarks 6 and 8.	Real	0.0
NA	Order of the strain energy polynomial function.	0 < Integer ≤ 6	1 See Remark 3
TAB1	Table identification number of TABLES1 entry that contains simple tension/compression data to be used in the estimation of the material constants Aij. xi values in the TABLES1 entry must be stretch ratios ℓ/ℓ_0 and yi values must be values of the engineering stress F/A_0.	Integer > 0 or blank	

(Continued)

	Stresses are negative for compression and positive for tension. If this convention is not followed the solution may fail to converge.	
TAB2	Table identification number of TABLES1 entry that contains equibiaxial tension data to be used in the estimation of the material constants Aij. xi values in the TABLES1 entry must be stretch ratios ℓ/ℓ_0. yi values must be values of the engineering stress F/A_0. ℓ is the current length, F is the current force, ℓ_0 is the initial length and A_0 is the cross-sectional area. In the case of pressure of a spherical membrane, the engineering stress is given by $P r_0 \lambda^2 / 2 t_0$ where P is the current value of the pressure and r_0, t_0 is the initial radius and thickness.	Integer > 0 or blank
TAB3	Table identification number of TABLES1 entry that contains simple shear data to be used in the estimation of the material constants Aij. xi values in the TABLES1 entry must be values of the shear tangent γ and yi values must be values of the engineering stress F/A_0 .	Integer > 0 or blank
TABD	Table identification number of TABLES1 entry that contains pure volumetric compression data to be used in the estimation of the material constants Di. xi values in the TABLES1 entry must be values of the volume ratio $J = \lambda^3$ where $\lambda = \ell/\ell_0$ is the stretch ratio in all three directions; yi values must be values of the pressure, assumed positive in compression.	Integer > 0 or blank

Remarks:

1. The hyperfoam generalized strain energy may be expressed as follows:

$$U(\lambda_1, \lambda_2, \lambda_3, J) = \sum_{i=1}^{NA} \frac{\mu_i}{\alpha_i^2}\left[(\lambda_1)^{\alpha_i} + (\lambda_2)^{\alpha_i} + (\lambda_3)^{\alpha_i} - 3 + \frac{1}{\beta_i}\left((J)^{-\alpha_i\beta_i} - 1\right)\right]$$

   where λ_1, λ_2 and λ_3 are principal stretches; $J = det\,F$ is the determinate of the deformation gradient;.

2. Up to 6 coefficients may be entered for the μ_i, α_i and β_i terms. Values for the $i = 1$ coefficients are required. All other default to values.

3. Values for μ_i, α_i and β_i for $i = 2, NA$ are required. Blank lines are not required for $i > NA$.

4. The β_i coefficients are related to the Poisson's ratio values, ν_i , by

$$\beta_i = \frac{\nu_i}{1 - 2\nu_i} \qquad \nu_i = \frac{\beta_i}{1 + 2\beta_i}$$

5. At small strains, the initial shear modulus, μ_o is given by

$$\mu_o = \sum_{i=1}^{N} \mu_i$$

(Continued)

6. At small strains, the initial bulk modulus, K_0, is given by

$$K_0 = \sum_{i=1}^{N} 2\mu_i \left(\frac{1}{3} + \beta_i \right)$$

7. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0, by 2.0.
8. TREF is used only as the reference temperature for the calculation of thermal loads.

(Continued)

MATL8 — MCT Definition for Membrane and Shell Elements

Description: The MATL8 entry defines the material properties for composite membrane and shell elements using the Multi-Continuum Theory method. The MCT method is a multiscale approach to analysis of composites. Failure of the composite lamina is calculated by evaluating the stress state in either the fiber or in the matrix rather than the homogenized composite lamina. The MCT method does allow for the interaction of failure in the fiber and matrix. The MCT method is applicable to uni-directional and woven composites. High fidelity micromechanics models enable the generation and optimization of composite properties. The MCT ply failure analysis is enabled by specifying MCT in the FT field of the PCOMP entry.

Format:

1	2	3	4	5	6	7	8	9	10
MATL8	MID	MIDM	MIDF	MIDC	FVF	TYPE		MCTMAT	
	LC	L	D	T	W		FBVF	WBVF	
	MIDX	MIDL	MIDW	MIDP					

Example:

MATL8	101	200	300	400	.7	1		0.066	
	1.E-2	1.E-2	1.E-3						

Field	Definition	Type	Default
MID	Material identification number. Referenced on a PSHELL or PCOMP entry only.	Integer > 0	Required
MIDM	Material identification number for the matrix material. See Remark 4.	Integer	Required
MIDF	Material identification number for the reinforcement (fiber) material. See Remark 4.	Integer	Required
MIDC	Material identification number for the composite material. See Remark 4.	Integer	Required
FVF	Volume fraction of fiber.	$0.3 \le$ Real ≤ 0.9	Required
TYPE	Reinforcement type, selected by one of the following values: 1 = Aligned continuous fibers 6 = Plain weave fabrics (MCT only) See Remarks 4 and 5.	Integer	1

(Continued)

Field	Definition	Type	Default
MCTMAT	MCT material input, selected by one of the following values: 1 = Perform MCT optimization of input materials 2 = Use input materials without modification 3 = Use default Carbon/Epoxy fiber matrix 4 = Use default Glass/Epoxy fiber matrix 5 = use default Kevlar/Epoxy fiber matrix See Remarks 6, 7, and 9.	Integer	1
LC	Short fiber critical length.	Real ≥ 0.0	Required if TYPE = 3
L	Fiber Length (not used).	Real ≥ 0.0	Required if TYPE = 3, 4 , or 5
D	Fiber Diameter (not used).	Real ≥ 0.0	Required if TYPE = 3 or 5
T	Fiber plate thickness (not used).	Real ≥ 0.0	
W	Fiber plate width.	Real ≥ 0.0	Required if TYPE = 4
FBVF	Fill bundle volume fraction. See Remark 8.	0.2 ≤ Real ≤ 0.37	Required if TYPE = 6
WBVF	Warp bundle volume fraction. See Remark 8.	0.2 ≤ Real ≤ 0.37	FBVF
MIDX	(not used)	Integer > 0	
MIDL	(not used)	Integer > 0	
MIDW	(not used)	Integer > 0	
MIDP	(not used)	Integer > 0	

Remarks:

1. The material identification number must be unique for all MATi entries.
2. The transverse shear moduli G1Z and G2Z are computed automatically from the in-plane shear modulus G12.
3. The MCT (Multicontinuum Theory) method is a multiscale approach to composites analysis. Failure in the composite lamina is calculated by evaluating the stress state in either the fiber or matrix, rather than the homogenized composite lamina, allowing one to capture interactions between the two. The method is applicable to unidirectional and woven composites. High fidelity micromechanics models enable the generation/optimization of composite properties from properties of the matrix and fiber. MCT ply failure analysis is enabled by specifying MCT in the FT field of the PCOMP entry.
4. MIDM and MIDF may reference either a MAT1 or MAT8 entry. For MAT1 entries the E, G, NU, and RHO fields must be non-zero. The A, ST, SC, and SS fields are optional. For MAT8 entries the E1, E2, NU12, GI12, and RHO fields must be non-zero. The A1, Xt, Xc, Yt, Tc, and S fields are optional. MIDC is required for the MCT method. MIDC must reference a MAT8 entry only. MIDC specifies properties for the generated MAT8 material that are not calculated. The material allowables (Xt, Xc, Yt, etc.) must be specified on the MAT8 referenced by MIDC if failure index/strength are desired.
5. The Type field defines the fiber type. Fiber types are detailed in the following table:

(Continued)

TYPE	Description	Example
1	Aligned continuous fiber composite lamina. Individual continuous fibers oriented in a defined direction.	Unidirectional graphite fibers in an epoxy resin.
6	Plain weave composite lamina. Woven fabric, where fill and warp threads interlace alternately resulting in equal properties in each direction.	Graphite cloth in an epoxy resin.

6. The MCTMAT field affects how material properties specified on MIDM, MIDF, and MIDC are processed. When MCTMAT is set to 1 (default) MIDM and MIDF properties are optimized using a very high fidelity micromechanics model resulting in generated MIDC values. When MCTMAT is set to 2, the MIDM, MIDF, and MIDC values are assumed already optimized and no adjustment in values is made. MCTMAT set to 3, 4, or 5 provide optimized default values for common materials.
7. MCT default material properties (MCTMAT = 3, 4, or 5) require that PARAM, UNITS be specified for the correct selection of default material units corresponding to the model input material property units (see Section 5, Parameters, for more information on UNITS).
8. Material stability requires that if FBVF ≠ WBVF, then FBVF + WBVF ≤ .68. If this condition is not met a fatal error will be issued.
9. MCT default fiber and matrix material properties (MCTMAT = 3, 4, or 5) are listed in the following table in metric units:

Variables	Carbon Fiber	Glass Fiber	Kevlar Fiber	Epoxy (Carbon)	Epoxy (Glass)	Epoxy (Kevlar)
E1	2.3E+11 Pa	8.0E+10 Pa	8.0E+10 Pa	3.5E+9 Pa	3.35E+9 Pa	3.5E+9 Pa
E2	1.5E+9 Pa	8.0E+10 Pa	6.9E+10 Pa	3.5E+9 Pa	3.35E+9 Pa	3.5E+9 Pa
E3	1.5E+9 Pa	8.0E+10 Pa	6.9E+10 Pa	3.5E+9 Pa	3.35E+9 Pa	3.5E+9 Pa
G12	1.5E+9 Pa	3.3E+10 Pa	2.8E+10 Pa	1.3E+9 Pa	1.24E+9 Pa	1.3E+9 Pa
G13	1.5E+9 Pa	3.3E+10 Pa	2.8E+10 Pa	1.3E+9 Pa	1.24E+9 Pa	1.3E+9 Pa
G23	1.5E+9 Pa	3.3E+10 Pa	2.8E+10 Pa	1.3E+9 Pa	1.24E+9 Pa	1.3E+9 Pa
NU12	0.20	0.20	0.36	0.35	0.35	0.35
NU23	0.20	0.20	0.36	0.35	0.35	0.35
NU31	0.20	0.20	0.23	0.35	0.35	0.35
A1	-5.5E-7 /°C	4.9E-6 /°C	-5.0E-6 /°C	5.3E-5 /°C	5.8E-5 /°C	5.3E-5 /°C
A2	1.0E-5 /°C	4.9E-6 /°C	4.1E-5 /°C	5.3E-5 /°C	5.8E-5 /°C	5.3E-5 /°C
A3	1.0E-5 /°C	4.9E-6 /°C	4.1E-5 /°C	5.3E-5 /°C	5.8E-5 /°C	5.3E-5 /°C

(Continued)

MATR1 Rigid Material

Description: The MATR1 allows you to define a rigid body through the use of material identification number.

The MATR1 entry allows you to declare that a material is rigid. For a rigid material, all properties for that material except the density are ignored.

A part (set of elements) that references the material identification number on a MATR1 entry will constitute a rigid body. All of the nodes associated with the elements in the part will be used to define the rigid body. Autodesk Explicit automatically determines properties (i.e. location of the center of mass, total mass, and rotary inertia tensor about the center of mass) of the rigid body. An internal reference node for the rigid body is created with six degrees of freedom (three translations and three rotations).

Boundary conditions can be defined for the rigid body using the SPCR and SPCRD entries. Any boundary condition applied to nodes of the elements will be ignored. You must specify the boundary conditions for the rigid body using the reference node through the SPCR and SPCRD entries.

Format:

1	2	3	4	5	6	7	8	9	10
MATR1	MID		CID	M					
	I11	I21	I22	I31	I32	I33			

Example:

MATR1	25		4	5.35					

Field	Definition	Type	Default
MID	Identification number of a MAT1 or MAT8 entry.	Integer > 0	Required
CID	Coordinate system identification number.	Integer ≥ 0	0
M	Mass value to be added to computed mass for rigid body.	Real or blank	0.0
Iij	Mass moments of inertia added to the calculated inertia tensor for the rigid body. The inertia terms are defined in terms of the coordinate system referenced by the coordinate system identification number CID. See Remark 3.	I11, I22, and I33, Real ≥ 0.0; I21, I31, and I32, Real ≥ 0.0.	0.0

Remarks:

1. The continuation may be omitted.
2. The coordinate system identification number, CID, is used for the definition of the inertia terms.
3. The form of the inertia matrix about its center of mass is taken as:

(Continued)

$$\begin{bmatrix} M & & & & & \\ & M & & & & \\ & & M & & & \\ & & & I_{11} & & \\ & & & -I_{21} & I_{22} & \\ & & & -I_{31} & -I_{32} & I_{33} \end{bmatrix}$$

In the above equations:

$$M = \int \rho dV$$

$$I_{11} = \int \rho\left(x_2^2 + x_3^2\right)dV$$

$$I_{22} = \int \rho\left(x_1^2 + x_3^2\right)dV$$

$$I_{33} = \int \rho\left(x_1^2 + x_2^2\right)dV$$

$$I_{21} = \int \rho x_1 x_2 dV$$

$$I_{31} = \int \rho x_1 x_3 dV$$

$$I_{32} = \int \rho x_2 x_3 dV$$

The mass M is added to the mass that is initially computed for the rigid body, M_C. The total mass for the rigid body, **M**, is $M+M_C$.

The x_1, x_2, x_3 are components of distance from the center of gravity in the coordinate system defined in field 4. Only the magnitude of I_{ij} should be supplied; the negative signs for the off-diagonal terms are supplied automatically. The inertia terms are transformed to the principal coordinate system for the inertia tensor for the rigid body.

MATS1 — Material Stress Dependence

Description: Specifies stress-dependent material properties. This entry is used if a MAT1 entry is specified with the same MID.

Format:

1	2	3	4	5	6	7	8	9	10
MATS1	MID	TID	TYPE	H	YF	HR	LIMIT1	LIMIT2	
	DILATION	YTYPE							

Example:

MATS1	25	100	1	1.E5	1	1	2.E4		

Field	Definition	Type	Default
MID	Identification number of a MAT1, MAT2, MAT8, MAT9, or MAT12 entry.	Integer > 0	Required
TID	Identification number of a TABLES1, TABLESR or TABLEST entry. If H is given, then this field must be blank. See Remark 2.	Integer ≥ 0 or blank	
YF	Yield function criterion, selected by one of the following values 1 = von Mises 4 = Drucker-Prager	Integer	von Mises
H	Work hardening slope (slope of stress vs. plastic strain) in units of stress. For more than a single slope in the plastic range, the stress-strain data must be supplied on a TABLES1 entry referenced by TID, and this field must be blank. See Remark 1.	Real	
YF	Yield function criterion, selected by one of the following values: 1 = von Mises	Integer	von Mises
HR	Hardening rule, selected by one of the following values: 1 = Isotropic	Integer	Isotropic
LIMIT1	Initial yield point. Y_1. Ignored if TID ≠ 0	Real	0.0
LIMIT2	Internal friction angle (measured in degrees) for the Drucker-Prager yield criteria.	Real	0.0
DILATION	Dilation angle (measured in degrees) for the Drucker-Prager yield criteria.	Real < 71.5	0.0
YTYPE	Yield curve type for the Drucker-Prager yield criteria. YTYPE = 1 Compression YTYPE = 2 Tension YTYPE = 3 Shear	Integer > 0	1

(Continued)

Remarks:

1. If TYPE = PLASTIC, either the table identification TID or the work hardening slope H may be specified, but not both. If the TID is omitted, the work hardening slope H must be specified unless the material is perfectly plastic. The plasticity modulus (H) is related to the tangential modulus (E_T) by

$$H = \frac{E_T}{1 - \frac{E_T}{E}}$$

where E is the elastic modulus and $E_T = \frac{dY}{d\varepsilon}$ is the slope of the uniaxial stress-strain curve in the plastic region. See Figure 1.

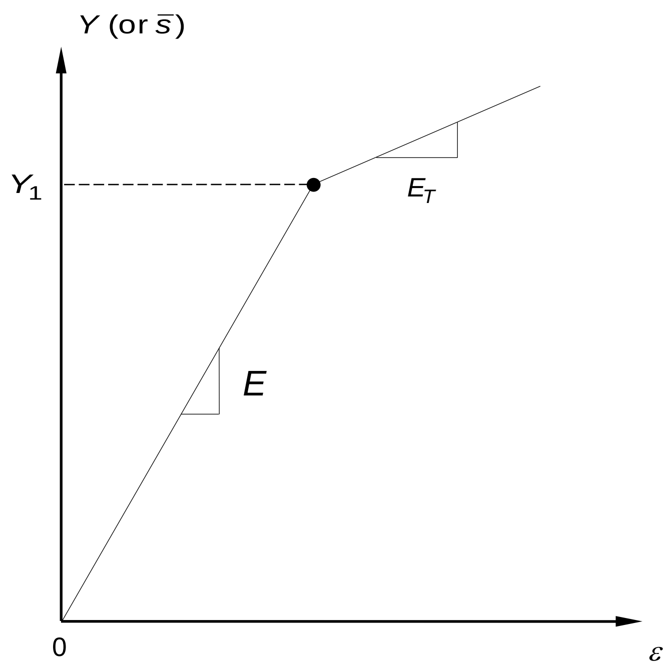


Figure 1. Stress-Strain Curve Definition When H is Specified in Field 5.

2. If TID is given, TABLES1 entries (Xi, Yi) of stress-strain data (ε_k, Y_k) must conform to the following rule: the curve must be defined in the first quadrant. The first point must be at origin (X1 = 0, Y1 = 0) and the second point (X2, Y2) must be at the initial yield point (Y_1 or $2c$) specified on the MATS1 entry. The slope of the line joining the origin to the yield stress must be equal to the value of E.

(Continued)

MOMENT — Static Moment

Description: The MOMENT entry defines a moment at a grid point by specifying a vector.

Format:

1	2	3	4	5	6	7	8	9	10
MOMENT	SID	G	CID	M	N1	N2	N3		

Example:

MOMENT	3	441	4	10.0	1.0	-1.0	0.0		

Field	Definition	Type	Default
SID	Load set identification number.	Integer > 0	Required
G	Grid point identification number.	Integer > 0	Required
CID	Coordinate system identification number.	Integer ≥ 0 or blank	0
M	Moment vector scale factor.	Real	Required
N1, N2, N3	Moment vector components measured in coordinate system defined by CID.	Real	Required; must have at least one nonzero component

Remarks:

1. The moment applied to grid point G is given by:

$$\vec{m} = M\vec{N}$$

where $\vec{N}$ is the vector defined in fields 6, 7 and 8.

2. Load sets must be selected in the Case Control Section (LOAD = SID).
3. A CID of zero references the basic coordinate system.

(Continued)

MOMENT1 Static Moment, Alternate Form 1

Description: The MOMENT1 entry defines a moment at a grid point by specification of a value and two grid points that determine the direction.

Format:

1	2	3	4	5	6	7	8	9	10
MOMENT1	SID	G	M	G1	G2				

Example:

MOMENT1	3	141	-4.5	10	11				

Field	Definition	Type	Default
SID	Load set identification number.	Integer $>$ 0	Required
G	Grid point identification number.	Integer $>$ 0	Required
M	Moment magnitude.	Real	Required
G1, G2	Grid point identification numbers.	Integer $>$ 0; G1 = G2	Required

Remarks:

1. The static load applied to grid point G is given by:

$$\vec{m} = M\vec{n}$$

where $\vec{n}$ is a unit vector parallel to a vector for G1 to G2.

2. Load sets must be selected in the Case Control Section (LOAD = SID).

MULLINS — Mullins effect

Description: Specifies the Mullins effect for hyperelastic materials. This option must reference a material ID on a MATHP or MATHP1 option..

Format:

1	2	3	4	5	6	7	8	9	10
MULLINS	MID	R	M	BETA					

Example:

MULLINS	3	5.	220.	.1					

Field	Definition	Type	Default
MID	Material identification number.	Integer > 0	Required
R	Mullins coefficient	Real > 0	0.0
M	Mullins coefficient	Real > 0	0.0
BETA	Mullins coefficient	Real > 0	0.0

(Continued)

MVISCO — Viscoelastic Material Properties

Description: Defines material properties for viscoelastic materials. . This entry is used if a MAT1, MATHP, MATHP1 or MATHPF entry is specified with the same MID.

Format:

1	2	3	4	5	6	7	8	9	10
MVISCO	MID	TYPE	C1	C2	TREF				
	T1	V1	T2	V2	T3	V3	T4	V4	
	T5	V5							

Example:

MVISCO	5	1							
	.25	.5	.5	.25	.75	.1			

Field	Contents	Type	Default
MID	Material identification number.	Integer > 0	Required See Remark 1
TYPE	Type of viscoelastic table to be used to define the behavior: Type = 1 Uniaxial Prony Series Type = 2 Shear Prony Series Type = 3 Uniaxial Test Data Type = 4 Shear Test Data	Real	0.0 See Remarks 2 and 3
C1	Coefficient for theromorheologically simple behavior shift function.	Real	0.0 See Remark 4
C1	Coefficient for theromorheologically simple behavior shift function.	Real	0.0 See Remark 4
TREF	Reference Temperature for theromorheologically simple behavior model.	Real	0.0 See Remark 4
Ti	Time values in the table.	Real ≥ 0	0.0 See Remarks 2 and 3
Di	Dependent variable values in the table.	Real ≥ 0	0.0 See Remarks 2 and 3

(Continued)

Remarks:

1. This option looks for a elastic MAT1, MATHP, MATHP1, or MATHPF material with the same MID and augments the elastic properties to include viscoelastic effects.
2. The TYPE parameter indicates how the table defined by the Ti, Vi values are to be interpreted.
   - TYPE = 1 The table contains the raw Uniaxial Prony Series coefficients (Ti = TAUi, Di = Ei).
   - TYPE = 2 The table contains the raw Shear Prony Series coefficients (Ti = TAUi, Di = Gi).
   - TYPE = 3 The table contains the raw Uniaxial Test data (Ti = Timei, Di = Ei).
   - TYPE = 4 The table contains the raw Shear Test data (Ti = Timei, Di = Ei).

   When test data are supplied (TYPE=3 or TYPE=4), the appropriate Prony Series coefficients are fit to the data.
3. Enter as many lines as necessary to define the table with up to 4 pairs of Ti, Di values per line. Reading is terminated when a blank field is found, an ENDT value is found, or a new keyword line is read.
4. The theromorheologically simple behavior shift function is defined by the Williams-Landell-Ferry approximation

$$\xi(t) = \int_0^t \varphi\left(T\left(\tau\right)\right)d\tau = \int_0^t \frac{-C_1\left(T - T_{ref}\right)}{C_2 + \left(T - T_{ref}\right)}d\tau$$

(Continued)

NEIXPROP — Numerical Stabilization Defaults

Description: The NEIXPROP entry allows you to set the default coefficients for various numerical stabilization algorithms for specific parts of the model.

Format:

1	2	3	4	5	6	7	8	9	10
NEIXPROP	PID	HGVAL	BV1	BV2	TSVAL	DRILL			

Example:

NEIXPROP	3	.1							

Field	Definition	Type	Default
PID	Part identification number of the part in the model to which these properties are to be applied.	Integer > 0	Required
HGVAL	Hourglass stiffness scale factor.	Real > 0	0.05
BV1	Linear bulk viscosity coefficient.	Real > 0	0.06
BV2	Quadratic bulk viscosity coefficient.	Real > 0	1.2
TSVAL	Transverse shear stiffness scale factor.	Real > 0	0.83333
DRILL	Drilling moment stiffness factor.	Real > 0	5.E-4

(Continued)

NREFLECT Non-Reflecting Boundary

Description: The NREFLECT entry specifies faces that define a non-reflecting boundary on solid elements. An incident wave encountering a non-reflecting boundary "passes" through the boundary without being reflected. The boundary must be defined by exterior faces on solid elements.

The definition of the segments defining a non-reflecting surface follows the same rules as the definition of a BSSEG surface. Either 4-node quadrilateral faces or 3-node triangular faces may be defined A triangular segment is defined by specifying a zero or blank grid ID for the fourth node.

The segments defining the surface should all have their outward normal defined such that they point out of the solid body (towards the semi-infinite medium). The outward normal for the segments is defined by the usual counter-clockwise orientation of the grid point number on each segment.

Format:

1	2	3	4	5	6	7	8	9	10
NREFLECT	SID	G1A	G2A	G3A	G4B	G1B	G2B	G3B	
	G4B	G1C	G2C	G3C	G4C	- etc.-			

Example:

NREFLECT	100	204	201	202	203	203	202	199	
	198	198	199	196	197	197	196	195	
	194								

Field	Definition	Type	Default
SID	Set identification number for the nonreflecting boundary.	Integer > 0	Required
Gi	Grid points defining the face.	Integer ≥ 0	See Remarks 1 and 2

Remarks:

1. If the face is a quadrilateral face (four nodes), each of the Gi are unique grid point identification numbers. If the face is a triangular face (three nodes), then G3 is equal to G4. A triangular face can be specified by skipping (leave blank) the 4^{th} node GID or by repeating the 4^{th} node GID as the same as the 3^{rd} node GID.
2. A sequential, counter-clockwise numbering of the grid point identification on the face defines the positive normal for the facing pointing in the exterior direction.

(Continued)

PARAM Parameter

Description: The PARAM entry specifies values for parameters used in solution sequences.

Format:

1	2	3	4	5	6	7	8	9	10
PARAM	N	V							

Example:

PARAM	ALPHA	1.0E-5							

Field	Definition	Type
N	Parameter name.	Character
V	Parameter value.	Integer, real, or character

Remarks:

1. Only parameters for which assigned values are allowed may be given values via the PARAM entry.
2. See Section 5, *Parameters*, for a list of parameter definitions.

(Continued)

PBAR — Bar Element Property

Description: The PBAR entry defines the properties of bar elements (CBAR entry).

Format:

1	2	3	4	5	6	7	8	9	10
PBAR	PID	MID	A	I1	I2	J			

Example:

PBAR	40	5	4.5	2.9	5.45				

Field	Definition	Type	Default
PID	Property identification number.	Integer > 0	Required
MID	Material identification number. See Remark 2.	Integer > 0	Required
A	Area of the beam cross-section.	Real > 0.0	Required
I1	Area moments of inertia for bending in plane 1 about the neutral axis.	Real > 0.0	Required
I2	Area moments of inertia for bending in plane 2 about the neutral axis.	Real > 0.0	Required
J	Torsional constant.	Real or blank	0.0

Remarks:

1. PBAR entries must all have unique property identification numbers.
2. PBAR entries may only reference MAT1 material entries.

PBEAM Beam Element Property

Description: The PBEAM entry defines the properties of beam elements (CBEAM entry).

Format:

1	2	3	4	5	6	7	8	9	10
PBEAM	PID	MID	A	I1	I2	I12	J		

Example:

PBEAM	40	5	4.5	2.9	5.45				

Field	Definition	Type	Default
PID	Property identification number.	Integer > 0	Required
MID	Material identification number. See Remark 2.	Integer > 0	Required
A	Area of the beam cross-section.	Real > 0.0	Required
I1	Area moments of inertia for bending in plane 1 about the neutral axis.	Real > 0.0	Required
I2	Area moments of inertia for bending in plane 2 about the neutral axis.	Real > 0.0	Required
I12	Area product of inertia ($I1*I2 > I12^2$).	Real	0.0
J	Torsional constant.	Real or blank	0.0

Remarks:

1. PBEAM entries must all have unique property identification numbers.
2. PBEAM entries may only reference MAT1 material entries.

PCOMP — Layered Composite Element Property

Description: The PCOMP entry defines the properties of an n-ply composite material laminate.

Format:

1	2	3	4	5	6	7	8	9	10
PCOMP	PID	Z0		SB	FT	TREF		LAM	
	MID1	T1	THETA1		MID2	T2	THETA2		
	MID3	T3	THETA3		- etc.-				

Example:

PCOMP	190	-0.256		2500.0	TSAI				
	200	0.065	0.0		210	0.04	45.0		
	220	0.03	60.0						

Field	Definition	Type	Default
PID	Property identification number.	Integer > 0	Required
Z0	Distance from the reference plane to the bottom surface.	Real	-1/2 element thickness
SB	Allowable shear stress of the bonding material (allowable interlaminar shear stress). The SB parameter is required if failure index is desired.	Real > 0.0	
FT	Failure theory. The following theories are allowed. (If blank then no failure calculation is preformed.) HILL for the Hill theory HOFF for the Hoffman theory TSAI for the Tsai-Wu theory STRESS for the maximum stress theory STRAIN for the maximum strain theory MCT for Multicontinuum theory	Character or blank	
TREF	Reference temperature. See Remark 3.	Real	0.0

(Continued)

Field	Definition	Type	Default
LAM	Laminate option. If LAM = SYM, only plies on one side of the element centerline are specified. The plies are numbered starting with 1 for the bottom layer. If an odd number of plies is desired with LAM = SYM then the center ply thickness (Ti) should be half the actual thickness.	Character or blank	If blank, all plies must be specified
MIDi	Material ID of the various plies. The plies are identified by serially numbering them from 1 at the bottom layer. The MID's must refer to MAT1 or MAT8 Bulk Data entries. See Remark 6.	Integer > 0	MID1 required, see Remark 1
Ti	Thicknesses of various plies. See Remark 1.	Real or blank	T1 required
THETAi	Orientation angle of the longitudinal direction of each ply with the material axis of the element. (If the material angle on the element connection entry is 0.0, the material axis and side 1-2 of the element coincide.) The plies are numbered serially starting with 1 at the bottom layer. The bottom layer is defined as the surface with the largest minus Z value in the element coordinate system.	Real or blank	0.0

Remarks:

1. The default for MID2, …, MIDn is the last defined MIDi. In the example above, MID1 is the default for MID2, MID3, and MID4. The same logic applies to Ti.
2. At least one of the four values (MIDi, Ti, THETAi) must be present for a ply to exist. The minimum number of plies is one.
3. TREF given on the PCOMP entry will be used for all plies of the element; it will override values supplied on material entries for individual plies. If the PCOMP references temperature dependent material properties, then TREF given on the PCOMP will be used as the temperature to determine material properties. TEMPERATURE Case Control commands are ignored for deriving the equivalent PSHELL and MAT1 entries used to describe the composite element.
4. The failure index for the boundary material is calculated as Failure Index =max(τ_{1z}, τ_{1z}). The Failure Index for the ply is calculated as shown in the following table, on the next page.

(Continued)

Theory	Failure Index	Remarks
Hill	$\frac{\sigma_1^2}{x^2} - \frac{\sigma_1\sigma_2}{x^2} + \frac{\sigma_2^2}{y^2} + \frac{\tau_{12}^2}{s^2} = \text{F.I.}$	Orthotropic materials with equal strengths in tension and compression.
Hoffman	$\left(\frac{1}{x_t} - \frac{1}{x_c}\right)\sigma_1 + \left(\frac{1}{y_t} - \frac{1}{y_c}\right)\sigma_2 + \frac{\sigma_1^2}{x_t x_c} + \frac{\sigma_2^2}{y_t y_c} + \frac{\tau_{12}^2}{s^2} - \frac{\sigma_1\sigma_2}{x_t x_c} = \text{F.I.}$	Orthotropic materials under a general state of plane stress with unequal tensile and compressive strengths.
Tsai-Wu	$\left(\frac{1}{x_t} - \frac{1}{x_c}\right)\sigma_1 + \left(\frac{1}{y_t} - \frac{1}{y_c}\right)\sigma_2 + \frac{\sigma_1^2}{x_t x_c} + \frac{\sigma_2^2}{y_t y_c} + \frac{\tau_{12}^2}{s^2} + 2F_{12}\sigma_1\sigma_2 = \text{F.I.}$	Orthotropic materials under a general state of plane stress with unequal tensile and compressive strengths.
Max Stress	$\text{Max}\left[\left(\frac{\sigma_1}{X_t}\right), \left(\frac{\sigma_2}{Y_t}\right), \left(\frac{\lvert\tau_{12}\rvert}{S}\right)\right]$	None
Max Strain	$\text{Max}\left[\left(\frac{\varepsilon_1}{X_t}\right), \left(\frac{\varepsilon_2}{Y_t}\right), \left(\frac{\lvert\gamma_{12}\rvert}{S}\right)\right]$	None

5. The STRENGTHRATIO model parameter is used to request the output of the Tsai Strength Ratio (R) instead of Failure Index. (See Section 5, *Parameters*, for more information on STRENGTHRATIO.)
6. This entry may be used to define a layered shell element. The MIDi fields may only reference MAT1 or MAT8 entries.

PDAMP Damper Element Property

Description: The PDAMP entry specifies the damping value of a damper element (CDAMP1 or CDAMP2 entry).

Format:

1	2	3	4	5	6	7	8	9	10
PDAMP	PID1	B1	PID2	B2	PID3	B3	PID4	B4	

Example:

PDAMP	14	3.2	16	4.0					

Field	Definition	Type	Default
PIDi	Property identification number.	Integer > 0	Required
Bi	Force per unit velocity	Real	Required

Remarks:

1. PDAMP entries must all have unique property identification numbers.
2. Up to four damping properties may be defined on a single entry.

PELAS — Elastic Element Property

Description: The PELAS entry specifies the stiffness and stress coefficient of a spring element (CELAS1 or CELAS2 entry).

Format:

1	2	3	4	5	6	7	8	9	10
PELAS	PID	K	GE		PID	K	GE		

Example:

PELAS	24	1.+3							

Field	Definition	Type	Default
PID	Property identification number.	Integer > 0	Required
K	Elastic property value.	Real	Required
GE	Structural element damping coefficient. See Remark 4.	Real or blank	0.0

Remarks:

1. PELAS entries must all have unique property identification numbers.
2. The use of negative spring values may result in fatal errors.
3. One or two elastic spring properties may be defined on a single entry.
4. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0, by 2.0.
5. If PARAM, W4 is not specified, GE is ignored in transient response analysis. (See Section 5, *Parameters*, for more information on W4.)

PELAST — Force Dependent Elastic Property

Description: The PELAST entry defines the force dependent properties for a PELAS Bulk Data entry. Force versus displacement values for nonlinear spring behavior are defined by this option.

Format:

1	2	3	4	5	6	7	8	9	10
PELAST	PID			TKNID					

Example:

PELAST	24			40					

Field	Definition	Type	Default
PID	Identification number of a PELAS entry.	Integer > 0	Required
TKNID	Identification number of a TABLEDi entry that defines the nonlinear force versus displacement relationship.	Integer > 0	

Remarks:

1. PELAST may only be referenced by CELAS1 or CELAS2 elements.

PLOAD Static Pressure Load

Description: The PLOAD entry defines a uniform pressure load on a triangular or quadrilateral surface comprised of surface elements and/or the faces of solid elements.

Format:

1	2	3	4	5	6	7	8	9	10
PLOAD	SID	P	G1	G2	G3	G4			

Example:

PLOAD	5	-3.5	15	12	19				

Field	Definition	Type	Default
SID	Load set identification number.	Integer > 0	Required
P	Pressure value.	Real	Required
Gi	Grid point identification numbers.	Integer > 0; G4 may be blank	Required

Remarks:

1. Load sets must be selected in the Case Control Section (LOAD = SID).
2. The grid points define either a triangular or a quadrilateral surface to which a pressure is applied. If G4 is blank, the surface is triangular.
3. In the case of a triangular surface, the assumed direction of the pressure is computed according to the right-hand rule using the sequence of grid points G1, G2, G3 illustrated in Figure 1. The total load on the surface is divided into three equal parts and applied to the grid points as concentrated loads. A minus sign in field 3 reverses the direction of the load.
4. In the case of a quadrilateral surface, the grid points G1, G2, G3, and G4 should form a consecutive sequence around the perimeter. The right-hand rule is applied to find the assumed direction of the pressure. Four concentrated loads are applied to the grid points in approximately the same manner as for a triangular surface.

(Continued)

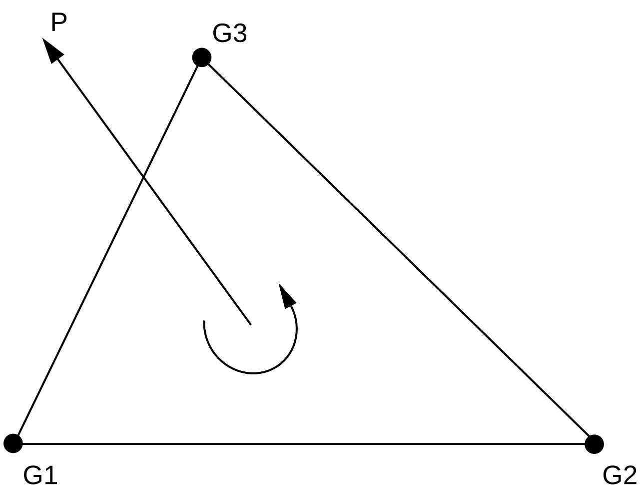


Figure 1. Pressure Convention for Triangular Surface.

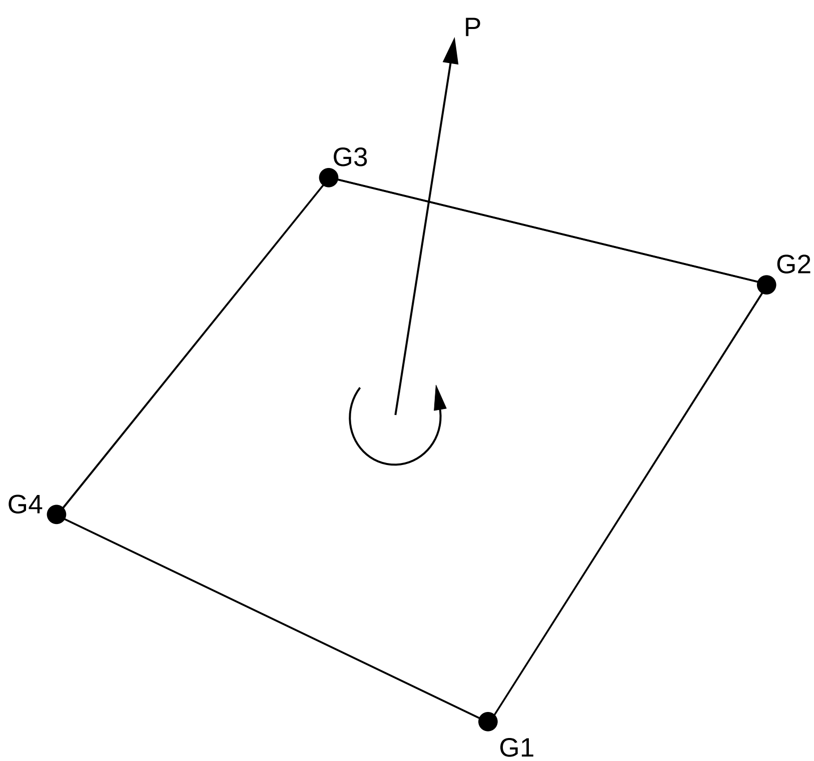


Figure 2. Pressure Convention for Quadrilateral Surface.

PLOAD2 Pressure Load on Shell Elements

Description: The PLOAD2 entry defines a uniform pressure load applied to shell elements. Only CQUAD4, CQUADR, CTRIA3, or CTRIAR elements may have a pressure load applied to them via this entry.

Format:

1	2	3	4	5	6	7	8	9	10
PLOAD2	SID	P	EID1	EID2	EID3	EID4	EID5	EID6	

Example:

PLOAD2	30	-1.3	106	222	21				

Alternate Format and Example:

PLOAD2	SID	P	EID1	THRU	EID2				

PLOAD2	40	12.0	16	THRU	122				

Field	Definition	Type	Default
SID	Load set identification number.	Integer > 0	
P	Pressure value.	Real	
EIDi	Element identification number(s).	Integer > 0; EID1 < EID2	

Remarks:

1. Load sets must be selected in the Case Control Section (LOAD = SID).
2. At least one EID must be present on each PLOAD2 entry.
3. If the alternate form is used, all elements EID1 through EID2 that are not compatible or do not exist will be skipped.
4. Elements must not be specified more than once.
5. The direction of the pressure is computed according to the right-hand rule using the grid point sequence specified on the element entry.
6. All elements directly referenced must exist.
7. Continuations are not allowed.

PLOAD4 — Pressure Loads on Face of Shell and Solid Elements

Description: The PLOAD4 entry defines a load on a face of a shell or solid element. Only CQUAD4, CQUADR, CTRIA3, CTRIAR, CHEXA, CPENTA, and CTETRA elements may have a pressure load applied to them via this entry.

Format:

1	2	3	4	5	6	7	8	9	10
PLOAD4	SID	EID	P1	P2	P3	P4	G1	G3 or G4	
	CID	N1	N2	N3					

Example:

PLOAD4	2	1405	1.0	1.5	1.5	1.0			

Alternate Format and Example:

PLOAD4	SID	EID1	P1	P2	P3	P4	THRU	EID2	
	CID	N1	N2	N3					

PLOAD4	2	1106	10.0	8.0	5.0		THRU	1143	
	6	0.0	1.0	0.0					

Field	Definition	Type	Default
SID	Load set identification number.	Integer > 0	Required
EID	Element identification number.	Integer > 0	Required
P1, P2, P3, P4	Load per unit surface area (pressure) at the corners of the face of the element.	Real or blank	P1 is the default for P2, P3, and P4
G1	Identification number of a grid point connected to a corner of the face.	Integer > 0 or blank	Required for solid elements
G3	Identification number of a grid point connected to a corner diagonally opposite to G1 on the same face of a CHEXA or CPENTA element. Required for the quadrilateral faces of CHEXA and CPENTA elements. Must be omitted for a triangular face on a CPENTA element.	Integer > 0 or blank	Required for CHEXA and CPENTA elements
G4	Identification number of the CTETRA grid point located at the corner; this grid point may not reside on the face being loaded.	Integer > 0 or blank	Required for CTETRA elements

(Continued)

Field	Definition	Type	Default
CID	Coordinate system identification number.	Integer ≥ 0 or blank	See Remark 2
N1, N2, N3	Components of vector measured in coordinate system defined by CID. Used to define the direction (but not the magnitude) of the load intensity.	Real	Required if CID is not blank and must have at least one non-zero component

Remarks:

1. The load set ID (SID) is selected by the Case Control command LOAD.
2. The continuation entry is optional. If fields 2, 3, 4, and 5 of the continuation entry are blank, the load is assumed to be pressure acting normal to the face. If these fields are not blank, the load acts in the direction defined in these fields. Note that if CID is a curvilinear coordinate system, the direction of loading may vary over the surface of the element. The load intensity is the load per unit of surface area, not the load per unit of area normal to the direction of loading.
3. For the faces of solid elements, the direction of positive pressure (defaulted continuation) is inward. For triangular (and quadrilateral faces) the load intensity P1 acts at grid point G1 and load intensities P2, P3 (and P4) act at the other corners in a sequence determined by applying the right-hand rule to the outward normal.
4. For shell elements, the direction of positive pressure (default continuation) is in the direction of positive normal, determined by applying the right-hand rule to the sequence of connected grid points. The load intensities P1, P2, P3 (and P4) act respectively at corner points G1, G2, G3 (and G4) for triangular (and quadrilateral) elements.
5. If P2, P3, and P4 are blank fields, the load intensity is uniform and equal to P1. P4 has no meaning for a triangular face and may be left blank in this case.
6. G1 and G3 are ignored for CTRIA3, CTRIAR, CQUAD4, and CQUADR elements.
7. The alternate format is available only for CTRIA3, CTRIAR, CQUAD4, and CQUADR elements. The continuation entry may be used in the alternate format.
8. For triangular faces of CPENTA elements, G1 is an identification number of a corner grid point that is on the face being loaded and the G3 or G4 field is left blank. For faces of CTETRA elements, G1 is the identification number of a corner grid point that is on the face being loaded and G4 is an identification number of the corner grid point that is not on the face being loaded. Since a CTETRA element has only four corner points, G4 will be unique and different for each of the four faces of a CTETRA element.

PLSOLID — Nonlinear Large Strain Solid Element Property

Description: Defines a nonlinear large strain solid element property (CHEXA, CPENTA, and CTETRA elements only).

Format:

1	2	3	4	5	6	7	8	9	10
PLSOLID	PID	MID	MCID						

Example:

PLSOLID	2	100	6						

Field	Definition	Type	Default
PID	Property identification number.	Integer $>$ 0	Required
MID	Identification number of a MAT1, MAT9, MAT12, MATHP, or MATHP1 entry.	Integer $>$ 0	Required
MCID	Identification number of the material coordinate system. See Remarks 3 and 4.	Integer $\geq$ -1 or blank	See Remark 3

Remarks:

1. PSOLID entries must have unique identification numbers.
2. Isotropic (MAT1), orthotropic (MAT8), hyperleastic (MATHP, MATHP1), or hyperfoam (MATHPF) material properties may be referenced.
3. See the CHEXA, CPENTA, or CTETRA entry for the definition of the element coordinate system. The material coordinate system (MCID) may be the basic system (0), any defined system (Integer $>$ 0), or the global coordinate system (-1 or blank). The default for MCID is the global coordinate system.

PROD

Rod Element Property

Description: The PROD entry defines the properties of rod elements (CROD entry).

Format:

1	2	3	4	5	6	7	8	9	10
PROD	PID	MID	A	J					

Example:

PROD	44	100	0.1	0.2E-3					

Field	Definition	Type	Default
PID	Property identification number.	Integer > 0	Required
MID	Material identification number.	Integer > 0	Required
A	Area of rod cross-section.	Real	Required
J	Torsional constant.	Real or blank	0.0

Remarks:

1. PROD entries must all have unique property identification numbers.
2. For structural problems, PROD entries may only reference MAT1 material entries.

PSHELL

Shell Element Property

Description: The PSHELL entry defines the membrane, bending, and transverse shear properties of shell elements (CTRIA3, CTRIAR, CQUAD4, and CQUADR entries).

Format:

1	2	3	4	5	6	7	8	9	10
PSHELL	PID	MID1	T	MID2					
				THETA/MCID					

Example:

PSHELL	44	100	0.1	100					

Field	Definition	Type	Default
PID	Property identification number.	Integer > 0	Required
MID1	Material identification number for the membrane.	Integer > 0 or blank	Required
T	Membrane thickness for Ti on the connection entry.	Real or blank	Required
MID2	Material identification number for bending.	Integer ≥ -1 or blank	See Remarks 4, 5, and 6
THETA	Material property orientation angle in degrees.	Real or blank	See Remark 7
MCID	Material coordinate system identification number.	Integer ≥ 0	See Remark 7

Remarks:

1. All PSHELL property entries must have unique identification numbers.
2. PSHELL entries may reference MAT1 material property entries.
3. Isotropic (MAT1) material properties may be referenced.
4. If only MID1 is specified, the elements referencing the PSHELL entry will have only membrane stiffness. If MID2 is specified in addition to MID1, the elements referencing the PSHELL entry will have bending and transverse shear stiffness in addition to membrane stiffness.
5. If MID2 specified, it must be the same as MID1.
6. The default for the MID2 field is MID1 when MID1 is a nonlinear material.
7. THETA/MCID is used only if field 8 of the CQUAD4 or CQUADR, or field 7 of the CTRIA3 or CTRIAR entry is blank. If field 5 of the PSHELL continuation is also blank, then THETA = 0.0 is assumed.

PSOLID — Solid Element Property

Description: The PSOLID entry defines the properties of solid elements (CHEXA, CPENTA, and CTETRA entries).

Format:

1	2	3	4	5	6	7	8	9	10
PSOLID	PID	MID	MCID						

Example:

PSOLID	2	100	6						

Field	Definition	Type	Default
PID	Property identification number.	Integer > 0	Required
MID	Identification number of a MAT1 entry.	Integer > 0	Required
MCID	Identification number of the material coordinate system. See Remark 3.	Integer ≥ -1 or blank	See Remark 3

Remarks:

1. PSOLID entries must have unique identification numbers.
2. Isotropic (MAT1), orthotropic (MAT8), hyperleastic (MATHP, MATHP1), or hyperfoam (MATHPF) material properties may be referenced.
3. See the CHEXA, CPENTA, or CTETRA entry for the definition of the element coordinate system. The material coordinate system (MCID) may be the basic system (0), any defined system (Integer > 0), or the global coordinate system (-1 or blank). The default for MCID is the global coordinate system.

(Continued)

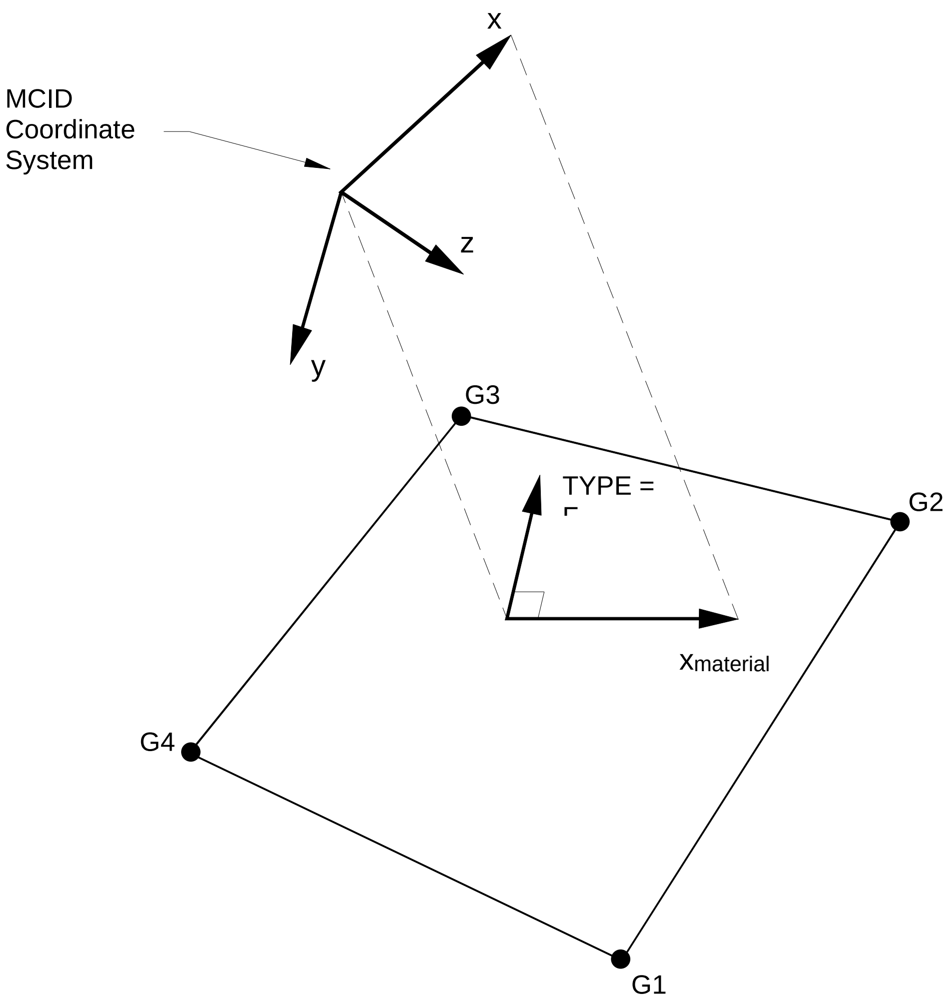


Figure 1. Layered Solid Element MCID Coordinate System Definition.

PSTRAIN — Initial Strain Definition

Description: The PSTRAIN entry defines values for initial stress in a part of the model.

This option sets the strain values in a part to the specified values. An amplitude function may be provided on the optional second line. This amplitude is typically used to ramp-up the strains from zero to the desired values over some time to mitigate dynamic effects that would occur if the strains were to be applied instantaneously at the beginning of time. If no amplitude function TID is provided, the initial strains are applied instantaneously. Note that in this case no stress will be computed and the effect of this option will not have any impact on the analysis results except to set the total strain value to an initial value that does not correspond to any stress.

A release time is typically specified on the optional second line when an amplitude is applied. The release time defines the time at which the amplitude is no longer applied to set the strains in the material. The default value of release time is zero which implies the amplitude function is used throughout the duration of the analysis.

Format:

1	2	3	4	5	6	7	8	9	10
PSTRAIN	SID	PID	EPSX	EPSY	EPSZ	GAMXY	GAMYZ	GAMZX	
	TID	RT							

Example: Initial Stress in a 3D Solid part

PSTRAIN	10	99	-.004	-.0005	.0005	0.	0.	0.	
	19	1.E-4							

Example: Initial Stress in a Beam part

PSTRAIN	10	18	-.003						

Example: Initial Stress in a Shell part

PSTRAIN	10	33	-.003	-.002		-.0012			

Field	Definition	Type	Default
SID	Identification number.	Integer > 0	Required See Remark 1
PID	Part identification number.	Integer > 0	Required See Remark 2
EPSX	XX component of strain	Real	0.0 See Remark 3
EPSY	YY component of strain	Real	0.0 See Remark 3
EPSZ	ZZ component of strain	Real	0.0

			See Remark 3
GAMXY	XY component of strain	Real	0.0 See Remark 3
GAMYZ	YZ component of strain	Real	0.0 See Remark 3
TAUZX	ZX component of strain	Real	0.0 See Remark 3
TID	Identification number	Integer ≥ 0	0 See Remark 4
RT	Release time	Real ≥ 0.0	0.0 See Remark 5

Remarks:

1. The SID must reference a DLOAD ID in the Case control.
2. The PID must reference a valid PSOLID, PSHELL, PBEAM, or PROD property ID. All elements connected to the PID specified will have the initial strain applied to them.
3. The number of strain components specified in this option depends upon the property type of the referenced property ID.
4. The amplitude function is applied to the strain values defined upon the first line as:

   $$\varepsilon(t) = A(t)\varepsilon_0$$

   If no amplitude TID is supplied, the specified initial strains are applied instantaneously at the beginning of time.
5. If no release time is supplied, the strains are applied according to the amplitude specification and not released.

(Continued)

PSTRESS — Initial Stress Definition

Description: The PSTRESS entry defines values for initial stress in a part of the model.

This option sets the stress values in a part to the specified values. An amplitude function may be provided on the optional second line. This amplitude is typically used to ramp-up the stresses from zero to the desired values over some time to mitigate dynamic effects that would occur if the stresses were to be applied instantaneously at the beginning of time. If no amplitude function TID is provided, the initial stresses are applied instantaneously. If an amplitude function is provided, the elements to which this pre-stress is applied will not compute stress increments from the strain increments using the material model while the amplitude is active.

A release time is typically specified on the optional second line when an amplitude is applied. The release time defines the time at which the amplitude is no longer applied and the material model is released to begin computing stress increments from the strain increments using the material model. The default value of release time is zero which implies the amplitude function is used throughout the analysis.

Format:

1	2	3	4	5	6	7	8	9	10
PSTRESS	SID	PID	SIGX	SIGY	SIGZ	TAUXY	TAUYZ	TAUZX	
	TID	RT							

Example: Initial Stress in a 3D Solid part

PSTRESS	10	99	1882.	-662.	512.	118.5	331.	-18.	
	19	.001							

Example: Initial Stress in a Beam part

PSTRESS	10	18	562.						

Example: Initial Stress in a Shell part

PSTRESS	10	33	118.	77.5		-27.6			

Field	Definition	Type	Default
SID	Identification number.	Integer > 0	Required See Remark 1
PID	Part identification number.	Integer > 0	Required See Remark 2
SIGX	XX component of stress	Real	0.0 See Remark 3
SIGY	YY component of stress	Real	0.0 See Remark 3

SIGZ	ZZ component of stress	Real	0.0 See Remark 3
TAUXY	XY component of stress	Real	0.0 See Remark 3
TAUYZ	YZ component of stress	Real	0.0 See Remark 3
TAUZX	ZX component of stress	Real	0.0 See Remark 3
TID	Identification number	Integer ≥ 0	0 See Remark 4
RT	Release time	Real ≥ 0.0	0.0 See Remark 5

Remarks:

1. The SID must reference a DLOAD ID in the Case control.
2. The PID must reference a valid PSOLID, PSHELL, PBEAM, or PROD property ID. All elements connected to the PID specified will have the initial stress applied to them.
3. The number of stress components specified in this option depends upon the property type of the referenced property ID.
4. The amplitude function is applied to the stress values defined upon the first line as:

   $$\sigma(t) = A(t)\sigma_0$$

   If no amplitude TID is supplied, the specified initial stresses are applied instantaneously at the beginning of time.
5. If no release time is supplied, the stresses are applied according to the amplitude specification and not released.

(Continued)

RBE2 Rigid Body Element, Form 2

Description: The RBE2 entry defines a rigid body whose degrees of freedom are specified at a single grid point and whose dependent degrees of freedom are specified at an arbitrary number of grid points.

This option allows you to create a rigid body by giving a list of grid points.

Note that all degrees of freedom are made rigid in Autodesk Explicit.

Format:

1	2	3	4	5	6	7	8	9	10
RBE2	EID	GN		GM1	GM2	GM3	GM4	GM5	
	GM6	GM7	GM8	GM9	- etc.-				

Example:

RBE2	12	2		15	18	22	25	27	
	34								

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
GN	Identification number of grid point to which all six independent degrees of freedom for the element are assigned.	Integer > 0	Required
GMi	Grid point identification numbers at which dependent degrees of freedom are assigned.	Integer > 0	

Remarks:

1. Rigid elements, unlike MPCs, are not selected through the Case Control Section.
2. The grid point designated as GN is referred to as the reference node. It may or may not be at the center of mass of the rigid body.
3. The grid points GMi make up what is called the set of member nodes for the rigid body (the member list).

ROTVEL — Initial Rotational Velocity

Description: The ROTVEL entry defines values for an initial rotational velocity.

Format:

1	2	3	4	5	6	7	8	9	10
ROTVEL	TID	SID	Ax	Ay	Az	Bx	By	Bz	
	OMEGA	TYPE	Vx	Vy	Vz				

Example:

ROTVEL	10	8	0.	0.	0.	1.	1.	1.	
	100.	GRID	-50.						

Field	Definition	Type	Default
TID	Identification number.	Integer > 0	Required
SID	Set identification number.	Integer > 0	Required
Ai	Coordinates of 1st point on axis of rotation. The axis of rotation is defined as the vector from Point A to Point B.	Real	0.0
Bi	Coordinates of 2nd point on axis of rotation. The axis of rotation is defined as the vector from Point A to Point B.	Real	0.0
OMEGA	Rotational velocity about the vector from Point A to Point B in radians/sec.	Real	0.0
TYPE	Type of Set. Choices are: ALLGRID — All grid points in the mesh are given the initial velocity. For this case the SID is ignored. GRID — All grid points in the set defined by SID are given the initial velocity. ELEM — All grid points in the set of elements defined by SID are given the initial velocity. PART — The SID is interpreted as a property ID and all grid points in the elements that use this property ID are given the initial velocity.	Character	Required
Vx	Translational velocity in the X direction	Real	0.0
Vy	Translational velocity in the Y direction	Real	0.0
Vz	Translational velocity in the Z direction	Real	0.0

SPC — Zero Value Single Point Constraint

Description: The SPC entry defines sets of single-point constraints and enforced zero displacements. Time dependent enforced displacement must be enforced using the SPCD option.

Note that the SPC option does allow you to prescribe an initial non-zero displacement. In general, instantaneous non-zero boundary conditions applied in an explicit dynamics analysis introduce very strong shocks into the model and, if the magnitude of the displacement is large enough, can turn elements inside-out in the first time increment. If you absolutely must introduce a displacement at time zero into the mesh, use the SPCD option with an appropriate amplitude function (TABLED1, TABLED2 or TABLEH options).

Format:

1	2	3	4	5	6	7	8	9	10
SPC	SID	G1	C1		G2	C2			

Example:

SPC	2	32	436						

Field	Definition	Type	Default
SID	Identification number of single point constraint set.	Integer > 0	Required
Gi	Grid point identification number.	Integer > 0	Required
Ci	Component number of global coordinate. (Up to six unique digits may be placed in the field with no embedded blanks.)	1 ≤ Integers ≤ 6	Required

Remarks:

1. Single-point constraint sets must be selected in the Case Control Section (SPC = SID).
2. From one to twelve degrees of freedom may be defined on a single entry.
3. Continuations are not allowed

SPC1 — Zero Value Single Point Constraint, Alternate Form

Description: The SPC1 entry defines sets of single-point constraints with zero displacements. Time dependent enforced displacement must be enforced using the SPCD option.

Format:

1	2	3	4	5	6	7	8	9	10
SPC1	SID	C	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	- etc.-					

Example:

SPC1	2	123	436	432	455	460	470		

Alternate Format and Example:

SPC1	SID	C	G1	THRU	G2				

SPC1	2	246	2	THRU	122				

Field	Definition	Type	Default
SID	Identification number of single-point constraint set.	Integer > 0	Required
Gi	Grid point identification number(s).	Integer > 0; G1 < G2	Required
C	Component number of global coordinate. (Up to six unique digits may be placed in the field with no embedded blanks.)	1 ≤ Integers ≤ 6	Required

Remarks:

1. Note that enforced displacements are not available via this entry.
2. Single-point constraint sets must be selected in the Case Control Section (SPC = SID) to be used.
3. If the alternate form is used, points in the sequence G1 through G2 are not required to exist. Points that do not exist will be skipped.

SPCADD — Single Point Constraint Set Combination

Description: The SPCADD entry defines a single-point constraint set as a union of single-point constraint sets defined via SPC or SPC1 entries.

Format:

1	2	3	4	5	6	7	8	9	10
SPCADD	SID	S1	S2	S3	S4	S5	S6	S7	
	S8	S9	- etc.-						

Example:

SPCADD	2	4	5	6	8				

Field	Definition	Type	Default
SID	Identification number of single point constraint set.	Integer > 0	Required
Si	Identification numbers of single-point constraint sets defined via SPC or by SPC1 entries.	Integer > 0; SID $\neq$ Si	Required

Remarks:

1. The Si values must be unique.
2. Single-point constraint sets must be selected in the Case Control Section (SPC = SID) to be used.
3. No Si may be the identification number of a single-point constraint set defined by another SPCADD entry.

SPCD — Enforced Displacement Value

Description: The SPCD entry defines an enforced displacement value for the transient analysis.

The SPCD entry can be referenced with a LOAD entry. If an SPCD entry is referenced with a LOAD entry, the specified displacement or rotation degrees of freedom will not vary with time. In general, an SPCD entry will be referenced with a TLOAD1 entry, which will use the information on the SPCD entry to create a time-varying kinematic boundary condition.

In general, it is highly recommended that you not introduce an initial non-zero displacement at time zero in the analysis. Instantaneous non-zero boundary conditions applied in an explicit dynamics analysis introduce very strong shocks into the model and, if the magnitude of the displacement is large enough, can turn elements inside-out in the first time increment. Care should be taken to specify an appropriate amplitude via the TABLED1, TABLED2 or TABLEH options.

Format:

1	2	3	4	5	6	7	8	9	10
SPCD	SID	G1	C1	D1	G2	C2	D2		

Example:

SPCD	2	523	246	1.6					

Field	Definition	Type	Default
SID	Identification number of single load set.	Integer > 0	Required
Gi	Grid point identification number.	Integer > 0	Required
Ci	Component number of global coordinate. (Up to six unique digits may be placed in the field with no embedded blanks.)	1 ≤ Integers ≤ 6	Required
D	Enforced displacement for all coordinates designated by G and C.	Real or blank	0.0

Remarks:

1. Values of D will override the values specified on a conflicting SPC Bulk Data entry, if the SID is selected on the LOAD Case Control command.
2. The Bulk Data LOAD combination entry will not combine an SPCD load.
3. This is the only method for applying enforced displacements, because the SPC and SPC1 Bulk Data entries only enforce zero displacement boundary conditions.

SPCR — Zero Kinematic Conditions for Rigid Bodies

Description: The SPCR entry defines enforced zero displacements or zero rotations on a rigid body. Time dependent enforced displacements or rotations on a rigid body must be enforced using the SPCRD entry.

Note that the SPCR option does allow you to prescribe an initial non-zero displacement. This is not recommended in general. Instantaneous, non-zero boundary conditions applied in an explicit dynamics analysis introduce very strong shocks into the model. If the magnitude of the initial displacement is large enough, the initial displacement can turn elements inside-out in the first time increment.

If grid points that belong to a rigid body have conflicting single-point constraints applied to them by an SPC entry and an SPC1 entry, this SPCR option will over-ride those values.

Format:

1	2	3	4	5	6	7	8	9	10
SPCR	SID	PID	C						

Example:

SPCR	2	32	346						

Field	Definition	Type	Default
SID	Identification number of single point constraint set.	Integer > 0	Required
PID	Property identification number of the property entry that uses the MATR1 entry.	Integer > 0	Required
Ci	Component number of global coordinate. (Up to six unique digits may be placed in the field with no embedded blanks.)	1 ≤ Integers ≤ 6	Required

Remarks:

1. Single-point constraint sets must be selected in the Case Control Section (SPC = SID).
2. From one to six degrees of freedom may be defined on a single entry.
3. Continuations are not allowed.
4. The SID on the SPCR entry references either the material identification number, MID, on a MATR1 entry or the element identification number, EID, on an RBE2 entry.
5. The constraints are applied to the reference node for the rigid body.

SPCRD — Time-Varying Kinematic Conditions for Rigid Bodies

Description: The SPCRD entry enforces time varying displacements and rotations on a rigid body. Constant displacements or rotations on a rigid body can be enforced using the SPCR entry.

In general, it is not recommended that you introduce an initial non-zero displacement at time zero. Instantaneous non-zero boundary conditions applied in an explicit dynamics analysis introduce very strong shocks into the model. If the magnitude of the displacement is large enough, the initial displacement can turn elements inside-out in the first time increment. Care should be taken to specify an appropriate amplitude via the TABLED1, TABLED2 or TABLEH options used to define the time-history for and SPCRD entry.

If grid points that belong to a rigid body have conflicting single-point constraints applied to them by an SPC entry or an SPC1 entry, this SPCRD option will override those values.

Format:

1	2	3	4	5	6	7	8	9	10
SPCRD	SID	PID	C	D					

Example:

SPCRD	2	4	246	1.E-3					

Field	Definition	Type	Default
SID	Identification number of single load set.	Integer > 0	Required
PID	Property identification number of the property entry that uses the MATR1 entry.	Integer > 0	Required
C	Component number of global coordinate. (Up to six unique digits may be placed in the field with no embedded blanks.)	1 ≤ Integers ≤ 6	Required
D	Enforced displacement for all coordinates of the rigid body reference node designated by PID and C.	Real or blank	0.0

Remarks:

1. The LOAD entry will not combine an SPCRD entry with other loads.
2. The SID on the SPCRD entry references either the material identification number, MID, on a MATR1 entry or the element identification number, EID, on an RBE2 entry.
3. The constraints are applied to the reference node for the rigid body.

SRS Structural Response Spectrum

Description: The SRS entry defines a set of grid point locations for which a structural response spectrum will be ccomputed for the dynamic event.

Format:

1	2	3	4	5	6	7	8	9	10
SRS	SID	NSID	MINFREQ	MAXFREQ	NUMINC	FIITERF	GRAV		
	DF1	DP2	DP3	Etc.					

Example:

SRS	5	99	100.	8000.	1000	1000.	386.4		
	1.	2.	3.	4.	5.				

Field	Definition	Type	Default
SID	Identification number.	Integer > 0	Required See Remark 1
NSID	Set identification number	Integer > 0	Required See Remark 2
MINFREQ	Minimum frequency (Hertz) for structural response spectrum	Real	1.
MaxFREQ	Maximum frequency (Hertz) for structural response spectrum	Real	200.
NUMINC	Number of frequence increments in structural response spectrum	Integer > 0	100
FILTERF	Filter frequency (Hertz)	Real ≥ 0	0. See Remark 3
GRAV	Gravity value used to compute structural response spectrum	Real > 0	1.
DFi	Damping Fractions (percent of critical damping)	Real	See Remark 4

Remarks:

1. SID must reference a DLOAD ID in the Case Definition.
2. NSID must reference a valid set of GRID IDs defined in the Case Definition.
3. If the FILTERF value is zero the acceleration history is not filtered prior to computing the structural response spectrum
4. Enter as many damping factors as desired. The case for zero damping is automatically included so the line containing damping factors can simply be skipped if desired.

TABLED1 Dynamic Load Tabular Function, Form 1

Description: The TABLED1 entry defines a tabular function for use in generating time-dependent loads or kinematic boundary conditions.

Format:

1	2	3	4	5	6	7	8	9	10
TABLED1	TID								
	x1	y1	x2	y2	x3	y3	- etc.-		

Example:

TABLED1	32								
	-2.0	8.0	1.9	6.5	3.1	7. 6	ENDT		

Field	Definition	Type	Default
TID	Table identification number.	Integer > 0	Required
xi, yi	Tabular values.	Real	Required

Remarks:

1. xi must be ascending order.
2. Discontinuities in the function are not allowed.
3. At least one continuation entry must be specified.
4. The end of the table is indicated by the existence of ENDT in either of the two fields following the last entry. Any continuations that follow the entry containing the end-of-table flag ENDT will be ignored.
5. TABLEM1 uses the algorithm:

$$y = y_T(x)$$

where x is input to the table and y is returned. The table look-up is performed using linear interpolation within the bounds of the x values of the table. Outside the bounds the function is assumed to be constant. That is, if x is less than x1, then the function evaluates as y1. If x is greater than xN, then the function evaluates as yN.

(Continued)

TABLED2 Dynamic Load Tabular Function, Form 2

Description: The TABLED2 entry defines a parametric tabular function for use in generating time-dependent dynamic loads

Format:

1	2	3	4	5	6	7	8	9	10
TABLED2	TID	X1							
	x1	y1	x2	y2	x3	- etc.-			

Example:

TABLED2	16	-12.5							
	2.0	-3.5	3.0	-5.2	4.0	5.9	8.0	6.4	
	SKIP	SKIP	10.0	6.7	ENDT				

Field	Definition	Type	Default
TID	Table identification number.	Integer > 0	Required
X1	Table parameter.	Real	0.0
xi, yi	Tabular values.	Real	Required

Remarks:

1. xi must be in ascending order.
2. Discontinuities may not be specified between any two points.
3. At least one continuation entry must be specified.
4. Any xi-yi pair may be ignored by placing SKIP in either of the two fields.
5. The end of the table is indicated by the existence of ENDT in either of the two fields following the last entry. Any continuations that follow the entry containing the end-of-table flag ENDT will be ignored.
6. TABLED2 uses the algorithm:

$$y = y_T(x - x1)$$

where x is input to the table and y is returned. The table look-up is performed using linear interpolation within the bounds of the x values of the table. Outside the bounds the function is assumed to be constant. That is, if x is less than x1, then the function evaluates as y1. If x is greater than xN, then the function evaluates as Yn.

TABLEH — Dynamic Load Harmonic Function Definition

Description: The TABLEH entry defines coefficients of a Fourier Series for use in generating time-dependent dynamic loads.

You may enter as many terms as desired using continuation lines.

The function has the form:

$$f(x) = A_o + \sum_{n=1}^{N} [A_i \cos(x - x_o) + B_i \sin(x - x_0)]$$

Format:

1	2	3	4	5	6	7	8	9	10
TABLEH	TID	A0	X0						
	A1	B1	A2	B2					

Example:

TABLEDH	2	0	0						
	5.2	-7.9	1.1	3.E-2					

Field	Definition	Type	Default
TID	Table identification number.	Integer > 0	Required
A0	Initial value of for function.	Real	0.0
X0	X value offset.	Real	0.0
Ai	Cosine coefficients.	Real	0.0
Bi	Sine coefficients.	Real	0.0

Remarks:

1. Values of D will override the values specified on an SPC Bulk Data entry, if the SID is selected on the LOAD Case Control command.
2. The Bulk Data LOAD combination entry will not combine an SPCR load.

TABLES1 — Material Property Table

Description: The TABLES1 entry defines a tabular function for stress-dependent material properties such as the stress-strain curve.

Format:

1	2	3	4	5	6	7	8	9	10
TABLES1	TID								
	x1	y1	x2	y2	x3	y3	- etc.-		

Example:

TABLES1	45								
	0.0	0.0	0.02	10000.	0.04	14000.	ENDT		

Field	Definition	Type	Default
TID	Table identification number.	Integer > 0	Required
xi, yi	Tabular values.	Real	0.0

Remarks:

5. xi must be in ascending order.
6. Discontinuities may not be specified between any two points.
7. At least one continuation entry must be specified.
8. The end of the table is indicated by the existence of ENDT in either of the two fields following the last entry. Any continuations that follow the entry containing the end-of-table flag ENDT will be ignored.
9. TABLES1 uses the algorithm:

$$y = y_T(x)$$

where x is input to the table and y is returned. The table look-up is performed using linear interpolation within the bounds of the x values of the table. Outside the bounds the function is assumed to be constant. That is, if x is less than x1, then the function evaluates as y1. If x is greater than xN, then the function evaluates as yN. The function evaluates as:

X-AXIS	Y-AXIS	y(x)
LINEAR	LINEAR	$\frac{x_{i+1} - x}{x_{i+1} - x_i} y_i + \frac{x - x_i}{x_{i+1} - x_i} y_{i+1}$

In the above equations, $x_i < x < x_{i+1}$.

TABLEST — Material Property Temperature Dependence Table

Description: Specifies the material property tables for temperature dependent yield stress curves.

Format:

1	2	3	4	5	6	7	8	9	10
TABLEST	TID								
	T1	TID1	T2	TID2	T3	TID3	- etc.-		

Example:

TABLEST	105								
	130.	20	195.	40	216.	60	ENDT		

Field	Definition	Type	Default
TID	Table identification number	Integer > 0	Required
Ti	Temperature values.	Real > 0	Required
TIDi	Table identification numbers of TABLES1 entries	Integer > 0	Required

Remarks:

1. TIDi must be unique with respect to all TABLES1, TABLEST, and TABLESR table identification numbers.
2. Temperature values must be listed in ascending order.
3. The first temperature value will be used as the reference yield surface temperature.
4. The end of the table is indicated by the existence of ENDT in either of the two fields following the last entry. Any continuations that follow the entry containing the end-of-table flag ENDT will be ignored.

TABLESR — Material Property Strain Rate Dependence Table

Description: Specifies the material property tables for strain rate dependent yield stress curves.

Format:

1	2	3	4	5	6	7	8	9	10
TABLESR	TID								
	R1	TID1	R2	TID2	R3	TID3	- etc.-		

Example:

TABLESR	45								
	0.0	10	0.1	11	1.0	12	ENDT		

Field	Definition	Type	Default
TID	Table identification number	Integer > 0	Required
Ri	Strain rate values.	Real > 0	Required
TIDi	Table identification numbers of TABLES1 entries	Integer > 0	Required

Remarks:

1. TIDi must be unique with respect to all TABLES1, TABLEST, and TABLESR table identification numbers.
2. Strain rate values must be listed in ascending order.
3. The first strain rate value will be used as the reference yield surface at zero strain rate.
4. The end of the table is indicated by the existence of ENDT in either of the two fields following the last entry. Any continuations that follow the entry containing the end-of-table flag ENDT will be ignored.

TIC — Transient Initial Condition

Description: The TIC entry defines values for the initial conditions of variables used in transient response analysis. Both displacement and velocity may be specified at independent degrees of freedom. Note that the INITVEL entry and the ROTVEL entry can be used as an alternative to the TIC entry to define initial velocities for parts of the model in a more convenient form.

Format:

1	2	3	4	5	6	7	8	9	10
TIC	SID	G	C	U0	V0				

Example:

TIC	10	25	2	12.5	-5.0				

Field	Definition	Type	Default
SID	Set identification number.	Integer > 0	Required
G	Grid point identification number.	Integer > 0	Required
C	Component number of global coordinate (up to six unique digits may be placed in the field with no embedded blanks.)	0 ≤ Integer ≤ 6	Required
U0	Initial displacement.	Real	0.0
V0	Initial velocity.	Real	0.0

Remarks:

1. Transient initial condition sets must be selected with the Case Control command IC = SID.
2. If no TIC set id selected in the Case Control Section, all initial conditions are assumed zero.
3. Initial conditions for coordinates not specified on TIC entries will be assumed zero.

(Continued)

TLOAD1 — Transient Response Dynamic Load, Form 1

Description: The TLOAD1 entry defines a time-varying load or time varying kinematic boundary conditions of the form:

$$P(t) = A * F(t - \tau)$$

for use in transient response analysis. For example, in the above equation, if *F* represents a force at a point G, then *P(t)* represents the time history of the force at that grid point G.

Format:

1	2	3	4	5	6	7	8	9	10
TLOAD1	SID	EXCITEID		TYPE	TID				

Example:

TLOAD1	10	100			205				

Field	Definition	Type	Default
SID	Set identification number.	Integer > 0	Required
EXCITEID	DAREA, SPCD, or SPCRD set identification number that defines *A*.	Integer > 0	Required
TYPE	Defines the nature of the dynamic excitation. See Remark 2.	0 ≤ Integer ≤ 3 or character	0
TID	TABLEDi set identification number that defines *F*(*t*).	Integer > 0	Required

Remarks:

1. Dynamic load sets must be selected with the Case Control command DLOAD = SID.
2. The nature of the dynamic excitation is defined in the following table:

TYPE	Type of Dynamic Excitation
0, L, or LOAD	Applied load (force or moment) (default)
1, D, or DISP	Enforced displacement using SPCD or SPCR
2, V, or VELO	Enforced velocity using SPCD or SPCR
3, A, or ACCE	Enforced acceleration using SPCD or SPCR

(Continued)

3. The TYPE field determines the manner in which the EXCITEID field is used as described below:

   a) Excitation specified by TYPE is an applied load

      - If there *is no* LOADSET request in the Case Control then EXCITEID may directly reference load set entries.
      - If there *is a* LOADSET request in the Case Control then the model will reference load set entries specified by the LID or TID field in the selected LSEQ entries corresponding to the EXCITEID.

   b) Excitation specified by TYPE is an enforced motion

      - If there *is no* LOADSET request in the Case Control then EXCITEID will reference SPCD and SPCRD entries. If these entries indicate null enforced motion, Autodesk Explicit will then assume that the excitation is enforced motion will reference DAREA and load set entries just as in the case of applied load excitation.
      - If there *is a* LOADSET request in Case Control then the model will reference SPCD and SPCRD entries specified by the LID field in the selected LSEQ entries corresponding to the EXCITEID. If these entries indicate null enforced motion, Autodesk Explicit will then assume that the excitation is enforced and will reference load set entries corresponding to the DAREA entry in the selected LSEQ entries, just as in the case of applied load excitation.

TSTEPNL — Parameters for Nonlinear Transient Analysis

Description: The TSTEPNL entry defines a set of parameters for nonlinear transient analysis.

Format:

1	2	3	4	5	6	7	8	9	10
TSTEPNL	ID	NDT	DT	NO	NUMRST	KEEP	RTIME		

Example:

TSTEPNL	120	200	0.001						

Field	Definition	Type	Default
ID	Identification number. See Remark 1.	Integer > 0	Required
NDT	Used to specify the duration of the analysis step. The duration is computed as NDT*DT. See Remark 2.	Integer > 0	Required
DT	Time increment between output writes. See Remark 3.	Real > 0.0	Required
NO	Specifies time increment between binary output writes during the analysis. The time increment between binary output writes is NO*DT. See Remark 4.	Integer > 0	0
NUMRST	Number of restart checkpoints to write during the duration of the step specified on the TSTEPNL option that references this RESTARTW option. If this value is zero or negative only one restart checkpoint is written at the end of the duration. See Remarks 5, 6 & 7.	Integer > 0	1
KEEP	Number of restart files to keep. See Remarks 7 & 8.	Integer ≥ -1 or blank	-1
RTIME	Restart read time. Specifies checkpoint time within the restart file specified with the RESTARTR option. See Remark 9.	Real ≥ 0 or blank	blank

Remarks:

1. The TSTEPNL Bulk Data entry must be selected by the Case Control command TSTEPNL = ID. Each solution subcase requires a TSTEPNL command and either applied loads via TLOADi data or initial values from a previous subcase. Multiple subcases are assumed to occur sequentially in time with the initial values of time and displacement conditions of each subcase. Initial conditions specified using the IC Case Control command apply only to the first subcase.
2. NDT and DT are used to define the total duration for analysis, which is NDT*DT. Since the explicit time integration method computes the time increment automatically (based upon stability considerations), the actual number of time steps will usually be a much larger number than NDT.
3. DT is not the time increment used in the analysis. It is the time between binary output writes.

(Continued)

4. Results output is generated at times 0.0, 1*DT, 2*DT … NO*DT. A poor choice of DT and NO can result in very large files. Typically you want to choose values so that 20 to 50 results time frames are saved. Note the default value of 1 gives you NDT+1 results times written to the results file.
5. Large values of NUMRST can result in very large files. Typically the value of NUMRST is set to a value on the order of 10 to 20. A results output always occurs at time zero of the step. Therefore you will always get NUMRST+1 output times on the file.
6. If NUMRST is zero or blank, only one restart checkpoint will be written at the end of the step.
7. The NUMRST option can generate extremely large files. Typically this value is set on the order of 10 to 20. If a more frequent number of restart checkpoints is desired, it is highly recommended you use the KEEP option to retain only the last few restarts on the permanent file.
8. If the KEEP option is not used (value negative or zero), all the checkpoints are written into a single file for restart. The restart file name will be constructed from the base file name plus the ".rst.xml" file extension. If the KEEP option is used, the file names will be constructed as base file names plus the time of the restart checkpoint plus the ".rst.xml" file extension. Only KEEP files will be retained on permanent storage and the code will delete restart files as they become obsolete based upon the KEEP parameter.
9. If a restart reading file has been specified using the RESTARTR option, the RTIME value specifies the checkpoint time within the file from which to read. The time specified must be close to one of the times on the file (a tolerance of .01 is used to define "close"). If no value is specified here (the field is blank) but a RESTARTR option was used to specify a valid restart file, the last checkpoint time in the file is used for the restart.

PARAMETERS

Parameter Descriptions

Parameters are used for input of scalar values and for requesting special features. Parameters can be specified in the Case Control and the Bulk Data Sections of the Model Input File.

Transient Response Processor Parameters:

Parameter	Description	Type	Default

(Continued)

ALPHA	Rayleigh damping mass matrix scale factor.	Real	0.0
AUTOMASSSCALE	The AUTOMASSSCALE parameter sets a minimum time step. The use of this parameter results in the mass of elements with very small time steps being scaled in order to increase their time step. To best use this parameter, first run a data-check of your problem to get the average element time step information and minimum time step information for each element block. Set a minimum time with AUTOMASSSCALE so that the minimum times are raised closer to the average. Use this option carefully as it adds mass to the model. This parameter is mutually exclusive with AMSPERCENT.	Real	-1.0
AMSPERCENT	Sets the value of the maximum allowable percent increase in mass of any of the parts in the model. The application will determine the minimum stable time step corresponding to his mass increase and scale all elements in the part up to this limit. Only one part will actually get scaled to the limit and any other parts will be scaled up to the minimum time step of the part that governs. This parameter is mutually exclusive with AUTOMASSCALE.	Real > 0	0.0
AUTOQS	Specify that the solution procedure is to use the automatic quasi-static option. In this case the duration of the procedure defined on the TSTEPNL command by DT*NDT becomes a pseudo time. For each step in the procedure, the application will output results according to the NO parameter on the TSTEPNL command for the procedure.	ON/OFF	OFF
AUTOQSNUMSTEPS	Specify the number of steps to use to solve an automatic quasi-static procedure. If all the loads are applied in a single linear ramp, only one step is needed. If the amplitude functions that define the loading are not linear, enough step should be taken to follow the variation in the loading. Use the minimum number of steps possible to keep the computational run time down.	Integer>0	1
AMSEXTREMELIM	Sets the value of the ratio of average stability limit for a part to the part's minimum stability limit. This parameter allows the application to perform automatic mass scalin to the extremely small elements in an part. Most often used with tet meshes that have tiny slivers in them. The percent mass scaling applied to parts with extreme range is set by the AMSEXTREMEPERCENT parameter.	Real > 1	5.0
AMSEXTREMEPERCENT	Sets the allowable mass percent scaling value for the parts that have extreme ranges in their courant stability limits.	Real > 0	1.0
BETA	Rayleigh damping stiffness matrix scale factor.	Real	0.0
BSCREATE	Create a BSSEG surface from the free surface of the elements identified by a property ID.	INTEGER	none
HEARTBEAT	Set the interval (number of time steps) between writing a line of output.	INTEGER	50
HISTINT	Set the interval (number of times steps) between writing history data into the XYPLOTS.	INTEGER	Max of 1000 points or all time steps.

(Continued)

MAXADMAXIMUM	Set the maximum allowable value for the MAXAD value on BSCONP definitions. Because Autodesk Explicit normally sets this to an unacceptable value, Autodesk Explicit checks the value against its maximum allowable value. If you want to override the limits use this parameter.	REAL > 0	1.0
MAXADMINIMUM	Set the maximum allowable value for the MAXAD value on BSCONP definitions. Because Autodesk Explicit normally sets this to an unacceptable value, Autodesk Explicit checks the value against its maximum allowable value. If you want to override the limits use this parameter.	REAL > 0	0.5
NLCOMPPLYFAIL	Nonlinear composite Progressive Ply Failure Analysis (PPFA) option. When set to ON, composite plies that fail the user specified failure theory (FT field on the PCOMP Bulk Data entry) will be reduced in material stiffness based on reduction scale factors specified on the MAT1 and MAT8 Bulk Data entries.	ON/OFF	OFF
NUMDELETE	If an element is set to be deleted (because of some failure) criterion, the element is deleted over the number of time steps set by this parameter. The internal forces calculated by the element at the time of failure are reduced to zero over NUMDELETE time steps.	INTEGER	20
STRENGTHRATIO	Controls the output of the Tsai Strength Ratio, which is provided in place of Failure Index for composite element ply results output. When set to OFF, the standard NASTRAN Failure Index is output. When set to ON, the Tsai Strength Ratio is calculated. Strength Ratio is considered more useful than Failure Index because it indicates exactly how to change the applied loading to achieve optimal ply performance (strength ratio equal to 1.0).	ON/OFF	OFF

Parameter	Description	Type	Default
TIMEREDUCTION	Time step scale factor. Use to reduce the time step size computed automatically by the explicit dynamics stability considerations.	Real	1.0
UNITS	Set problem definition units to be SI units.	Character	SI
W4	Damping coefficient for use with PELAST option.	Real	0.0

MODEL INPUT FILE COMMAND AND ENTRY SUMMARY

Model Input File Case Control Command Summary:

Subcase Control	Output Control	Model Modification	Model Generation	Miscellaneous
ANALYSIS	ACCELERATION	DELETION*	CONTACTDEL*	
BEGIN BULK	APPLIEDLOADS		CONTACTGEN*	
DEFORM	CONTACTFORCES*		SET	
DELSTOP	DISPLACEMENT		SETGEN	
DLOAD	ELFORCE			
IC	ELSTRAIN			
LOAD	ELSTRESS			
LOADSET	FORCE			
SOLUTION	PLASTICSTRAIN*			
SPC	REACTIONS*			
TSTEPNL	RESTARTREAD*			
	SET			
	SRS*			
	STRAIN			
	STRESS			
	VELOCITY			
	XYPLOT*			

Case Control Commands

* Denotes Autodesk Explicit extension

Model Input File Bulk Data Entry Summary:

Bulk Data Entries

Element	Property	Material	Load	Displacement	Coordinate	Miscellaneous
BSBOUNDS*	PBAR	MAT1	DLOAD	SPC	CORD1C	ENDDATA
BSCONP	PBEAM	MAT8	FORCE	SPC1	CORD1R	GRID
BSCREATE*	PCOMP	MATL8	FORCE1	SPCADD	CORD1S	NEIXPROPS*
BSORIENT*	PDAMP	MAT12	GRAV	SPCD	CORD2C	PARAM
BSNSET*	PELAS	MATC*	INITDIS*	SPCR*	CORD2R	TSTEPNL*
BSSEG	PELAST	MATCF*	INITVEL*	SPCRD*	CORD2S	
BSTHICK*	PLSOLID	MATHP	LOAD			
BSTYPE*	PROD	MATHP1	LSEQ			
CBAR	PSHELL	MATHPF	MOMENT			
CBEAM	PSOLID	MATL8	MOMENT1			
CDAMP1		MATR1*	NREFLECT*			
CDAMP2		MATS1	PLOAD			
CELAS1		MULLINS	PLOAD2			
CELAS2		MVISCO*	PLOAD4			
CHEXA		TABLES1	PSTRAIN*			
CONM2			PSTRESS*			
CPENTA			ROTVEL*			
CQUAD4			TABLED1			
CQUADR			TABLED2			
CROD			TABLEH*			
CTETRA			TABLES1			
CTRIA3			TIC			

CTRIAR	TLOAD1

* Denotes Autodesk Explicit extension