True*Grid*® Output Manual For NASTRAN®

A Guide and a Reference

by

Robert Rainsberger

Version 4.0.0

XYZ Scientific Applications, Inc.

April 25, 2019

Table of Contents

I. NASTRAN Output Guide

This manual teaches the use of **True*Grid*®** when applied to a model to serve as input to the NASTRAN® finite element simulation code. More specifically, this manual discusses the use of commands in **True*Grid*®** to produce material models, element types, boundary conditions, loads, procedures, and cases that are uniquely designed for the NASTRAN® output option in **True*Grid*®**. There are many commands in **True*Grid*®** that are used to define the FEA model no matter which FEA simulation code is being used. For example, the **SF**/**SFI** commands are used to shape the mesh. Such commands are not covered by this manual. They are covered throughly in the **True*Grid*®**'s Users manual.

There are two basic ways to use **True*Grid*®**: interactive and batch. In this manual, we discuss both ways. For any particular command in **True*Grid*®** discussed in this manual, we will show both the dialogue window for interactive use and the syntax for batch use. For a beginner, we recommend using the dialogue windows interactively. Saves the commands that form the model in a session file called tsave according to the syntax discussed here.

There are 3 phases in **True*Grid*®**: **control**, **part**, and **merge**. You can go to the **control** or **merge** phase by typing the command **control** or **merge**. There is a dialogue window for these commands, but it is easier to type the commands. You need to create a part using the **block** or **cylinder** command in order to get into the part phase. Take care when in the part phase. If you type the **control** or **merge** command, you will be ending the development of the part that you started with the **block** or **cylinder** command.

Commands, such as the **NASTMATS** command, which define materials unique to NASTRAN®, are covered completely in this manual. The meaning and purpose of these features within NASTRAN® are not discussed here. This manual is not a substitute for a NASTRAN® User's Manual. Options can be ignored, resulting in defaults being used. You should have some familiarity with the use of NASTRAN® when using the features discussed in this manual. There are many NASTRAN® features supported in **True*Grid*®**, but not all of them. This is a work in progress. Contact us if there is a feature you need to complete your model. You might find the **verbatim** command useful which replicates a string into the **True*Grid*®** output file for NASTRAN®.

True*Grid*® numbers the different element types independently. So there can be a beam element, a shell element, and a brick element, each numbered one. When the bulk data for NASTRAN® is written, each element is assigned a unique identification number. For example, suppose there are 100 beams, 100 bricks, and 100 shells in a model. The beam elements will be numbered from 1 to 100, the brick elements numbered from 101 to 200, and the shell elements numbered from 201 to 300. Most of the data structures in **True*Grid*®** are tables. For example, material models, surfaces, and curves. The integers that identify these objects are the indices into the appropriate table. To keep **True*Grid*®** running efficiently, use reasonably small integers to identify these objects.

Every element has a unique Element Id (EID), every material has a unique Material Id (MID), every property has a unique Property Id (PID), and every coordinate system has a unique Coordinate System Id (CID). Sometimes you choose the identification number, but in most cases these numbers are chosen automatically to insure they are unique. The list of boundary condition, constraint, and load entities in the NASTRAN® bulk data that are linked to a Set Id. (SID) are listed below. A SID identifies a set of nodes or faces linked to one of the entities below. This SID must be unique among the entities. **True*Grid*®** does not insure that the SIDs that you choose are unique among the different entities. You must make sure they are unique. It is likely that if they are not unique, NASTRAN® will let you know.

Entities That Use Unique SIDs

Initial Velocities
Initial Displacements
Beam Displacement
Force
Moments
Pressure
Traction
Fixed Temperature
Constraints (except local system constraints)

Font Conventions

When a command syntax is defined, different fonts are used to indicate the meaning. A literal is highlighted in **bold**. A symbol to be substituted with a literal or a number is *italicized*. A computer example uses the `Courier font`. A button from the Graphical User Interface is in ***italic and bold***.

Supported NASTRAN® Features

There are many features in **True*Grid*®** to create a complete model for NASTRAN®. The table below shows the commands that are used for each feature. Sometimes there are several commands listed. For example, shells can be generated using both the **block** and **cylinder** commands. The **n** and **th** are used to set the properties of these shells.

View some of the properties and features graphically using the **condition** (**co**) and **labels** (**la**) commands in the merge phase. The **tmm** calculates the mass of each part. Merge the nodes using one of the merging commands such as **stp** and, finally, use the **nastran** command to select NASTRAN® as the output option and the **write** command to actually create the input deck for NASTRAN®.

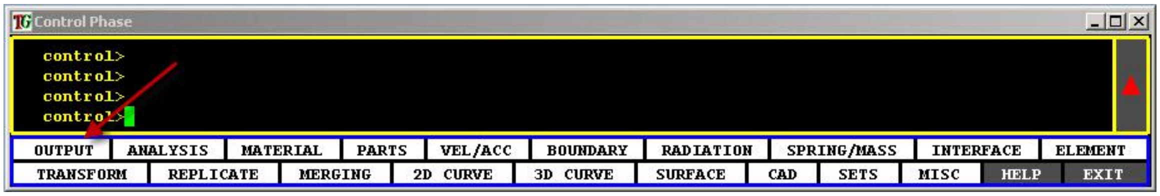
NASTRAN® feature	**True*Grid*®** commands
SOL	NASTOPTS
CEND	NASTOPTS
BEGIN BULK	NASTRAN
ENDDATA	NASTRAN
TITLE	TITLE
TEMP	TE/TEI
TEMPD	GTEMP
SUBCASE	PR/PRI, FC/FC, FCS/FCSI, FCC/FCCI, LCD/FLCD
PBAR	BSD, IBM/IBMI, JBM/JBMI, KBM/KBMI, BM, BEAM
PROD	BSD, IBM/IBMI, JBM/JBMI, KBM/KBMI, BM, BEAM
PTUBE	BSD, IBM/IBMI, JBM/JBMI, KBM/KBMI, BM, BEAM
PBEAM	BSD, IBM/IBMI, JBM/JBMI, KBM/KBMI, BM, BEAM
PBCOMP	BSD, IBM/IBMI, JBM/JBMI, KBM/KBMI, BM, BEAM
PBEND	BSD, IBM/IBMI, JBM/JBMI, KBM/KBMI, BM, BEAM
CBARAO	BSD, IBM/IBMI, JBM/JBMI, KBM/KBMI, BM, BEAM
GRID	BLOCK, CYLINDER, B/BI
CBEAM	IBM/IBMI, JBM/JBMI, KBM/KBMI, BM, BEAM
CBEND	IBM/IBMI, JBM/JBMI, KBM/KBMI, BM, BEAM
CROD	IBM/IBMI, JBM/JBMI, KBM/KBMI, BM, BEAM
CTUBE	IBM/IBMI, JBM/JBMI, KBM/KBMI, BM, BEAM
CHEXA	BLOCK, CYINDER, LINEAR, QUADRATIC
CPENTA	BLOCK, CYINDER, LINEAR, QUADRATIC
CTRETRA	BLOCK, CYINDER, LINEAR, QUADRATIC
CHEXA27	BLOCK, CYINDER, TRIQUADRATIC
CPYRAM	BLOCK, CYLINDER, LINEAR, QUADRATIC
CQUAD4	BLOCK, CYLINDER, LINEAR, QUADRATIC, SHOFF/SHOFFI
CQUAD8	BLOCK, CYLINDER, LINEAR, QUADRATIC, SHOFF/SHOFFI
CQUADR	BLOCK, CYLINDER, LINEAR, QUADRATIC, SHOFF/SHOFFI
CTRIA3	BLOCK, CYLINDER, LINEAR, QUADRATIC, SHOFF/SHOFFI
CTRIAR	BLOCK, CYLINDER, LINEAR, QUADRATIC, SHOFF/SHOFFI
CTRIA6	BLOCK, CYLINDER, LINEAR, QUADRATIC, SHOFF/SHOFFI
CSHEAR	BLOCK, CYLINDER, LINEAR, QUADRATIC
FORCE	FC/FC, FCS/FCSI, FCC/FCCI, NDL, LL
PLOAD4	PR/PRI
MAT1	NASTMATS
MAT2	NASTMATS
MAT3	NASTMATS
MAT4	NASTMATS
MAT5	NASTMATS
MAT8	NASTMATS

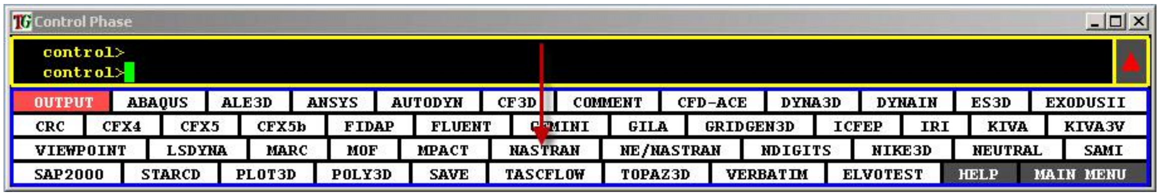
MAT9	NASTMATS
MAT10	NASTMATS
MATS1	NASTMATS
MATS3	NASTMATS
MATS8	NASTMATS
MATT1	NASTMATS
MATT2	NASTMATS
MATT3	NASTMATS
MATT4	NASTMATS
MATT5	NASTMATS
MATT8	NASTMATS
MATT9	NASTMATS
PSOLID	NASTMATS
PSHELL	NASTMATS
CORD2R	NASTMATS
PCOMP	NASTMATS
PSHEAR	NASTMATS
MOMENT	MOM
CMASS2	PM, NPM
MPC	JD, JT, MPC, MPCE
PELAS	SPD
CELAS1	SPDP, SPRING
SET	NSET/NSETI, ESET/ESETI
TIC	VE, DIS
RBE2	RBE
RBE3	RBE
RROD	RBE
RBAR	RBE
RTRPLT	RBE
TABDMP1	LCD/FLCD
TABLED1	LCD/FLCD
TABLED2	LCD/FLCD
TABLED3	LCD/FLCD
TABLED4	LCD/FLCD
TABLEM1	LCD/FLCD
TABLEM2	LCD/FLCD
TABLEM3	LCD/FLCD
TABLEM4	LCD/FLCD
TABLES1	LCD/FLCD
TABLEST	LCD/FLCD
TABRND1	LCD/FLCD
DEFORM	DEFORM
GRAV	NASTOPTS

NLPARM	NASTOPTS
RANDPS	NASTOPTS
RANDT1	NASTOPTS
TSTEP	NASTOPTS
TSTEPNL	NASTOPTS
SPCD	FD/FDI, FDC/FDCI, FDS/FDSI, LCD/FLCD
SPC	FD/FDI, FDC/FDCI, FDS/FDSI, LCD/FLCD
SPC1	B/BI
SPCADD	B/BI
PLOAD4	TRACT

Interactive Dialogue Windows Tailored to NASTRAN®

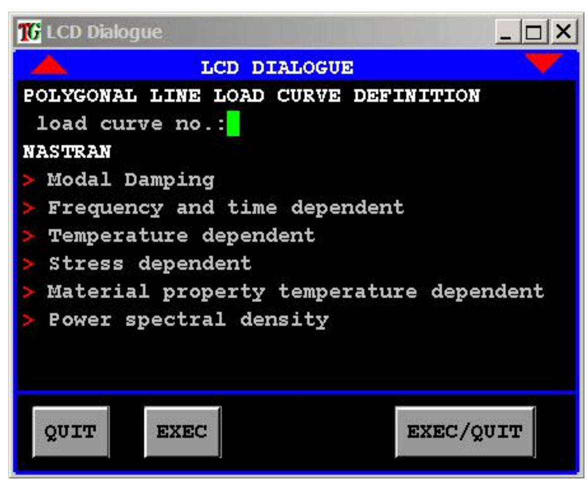
Some of the commands have options designed specifically for different simulation codes. For example, the **LCD** command to create a load curve (table). There are special options available for the three codes listed in the dialogue window. If you click on the word ***NASTRAN***, the options for NASTRAN® will appear.





The output format can be chosen in the ***CONTROL*** phase, when **True*Grid***® first starts up, under the menu item ***OUTPUT***. Click on the ***NASTRAN*** button in the ***OUTPUT*** menu. A dialogue window pops up. There are no parameters to specify - click on ***EXEC/QUIT***. When you finish building the model and specifying all the properties, conditions, and loads, you can go to the ***MERGE*** phase and click on the ***WRITE*** command under the ***OUTPUT*** menu for the model to be written in the NASTRAN® format.

If you take a moment, at the beginning, to choose NASTRAN® as you output format, then you can avoid having to select the ***NASTRAN*** item in many of the dialogue windows. As an example, if we chose NASTRAN® first and then chose the **LCD** command, the interactive dialogue window would be what you see on the right.



This is also true for the SID, SPD, FLCD, SIND, SPRING, BSD, OFFSET, and RBE commands. The SID command (Sliding Interface definition) has only one option for NASTRAN® called a dummy sliding interface to block the merging of nodes in specific areas of the mesh. The BSD, SIND, RBE, SPD and SPRING commands are discussed below. FLCD is like the LCD command. Load curves (tables) can be viewed using the LVC command. The OFFSET command adds an offset number to nodes, elements, node sets, and element sets for the entire mesh.

Transformations

Through out this manual, when “***trans*** ;” is found in the definition of a command, it means the application of a 3D transformation consisting of rotations, translations, and scaling. These transformations are formed using an ordered sequence of simple operators, listed below, followed by a semi-colon to terminate the list. In some cases, these transformations result in a CORD2R card.

The dialogue window to the right is an example of a transformation used to define a local coordinate system.

MX δ_x	Translation along the x-axis
MY δ_y	Translation along the y-axis
MZ δ_z	Translation along the z-axis
V $\delta_x\ \delta_y\ \delta_z$	Translate in a given vector direction
SCV $\delta_x\ \delta_y\ \delta_z$ *dist*	Direction and Distance Translation
DV $x_1\ y_1\ z_1\ x_2\ y_2\ z_2$	Difference Translation
RX θ_x	Rotate about the x-axis
RY θ_y	Rotate about the y-axis
RZ θ_z	Rotate about the z-axis
RAXIS $\theta\ x_0\ y_0\ z_0\ x_n\ y_n\ z_n$	Rotate about a general axis
RXY	Reflect about the xy-plane
RYZ	Reflect about the yz-plane
RZX	Reflect about the zx-plane
TF $x_1\ y_1\ z_1\ x_2\ y_2\ z_2\ x_3\ y_3\ z_3$	Transform To a Frame of Refer.
FTF $x_1\ y_1\ z_1\ x_2\ y_2\ z_2\ x_3\ y_3\ z_3\ x_4\ y_4\ z_4\ x_5\ y_5\ z_5\ x_6\ y_6\ z_6$	Frame of Refer. To Frame of Refer.
INV	Invert the Transformation
CSCA σ	Scale all coordinates
XSCA σ_x	Scale the x-coordinate
YSCA σ_y	Scale the y-coordinate
ZSCA σ_z	Scale the z-coordinate

where

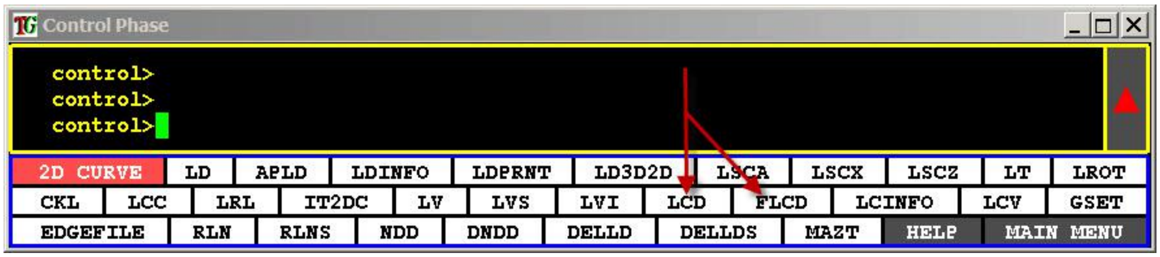
$x_i\ y_i\ z_i$ are 3D coordinates $\qquad$ δ_x is a distance in the x-direction

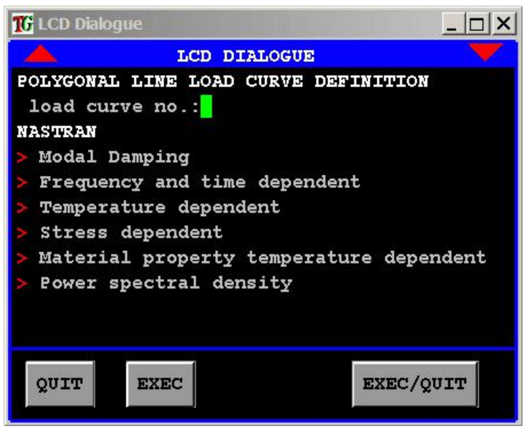
δ_y is a distance in the y-direction
δ_z is a distance in the z-direction
θ_x is a rotation about the x-axis (degrees)
θ_y is a rotation about the y-axis (degrees)
θ_z is a rotation about the z-axis (degrees)
θ is a rotation about an axis (degrees)
$x_n\ y_n\ z_n$ is a direction vector
σ is a global scale factor
σ_x is a scale factor in the x-coordinate
σ_y is a scale factor in the y-coordinate
σ_z is a scale factor in the z-coordinate

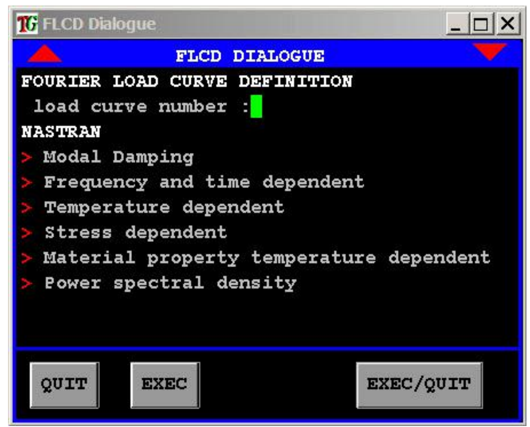
A frame of reference, defined by 3 points, forms a coordinate transformation that maps the global coordinate system to a local coordinate system. The first point serves as the local origin, the first point connects to the second point to form the local x-axis. The third point is in the positive half of the local xy-plane. The local z-axis is obtained from a cross product of the local x- and y-axis.

II. Tables

The **True*Grid*®** **LCD** (Load Curve Definition) and the **FLCD** (Fourier Load Curve Definition) commands will generate the following table cards in NASTRAN®: TABDMP1, TABLED1, TABLED2, TABLED3, TABLED4, TABLEM1, TABLEM2, TABLEM3, TABLEM4, TABLES1, TABLEST, and TABRND1. Both commands are found in all phases under the ***2D Curve*** menu item. Load curves (tables) are frequently used to define time or temperature dependent conditions. See the **True*Grid*®** manual to learn about all of the ways to generate these 2D curves.

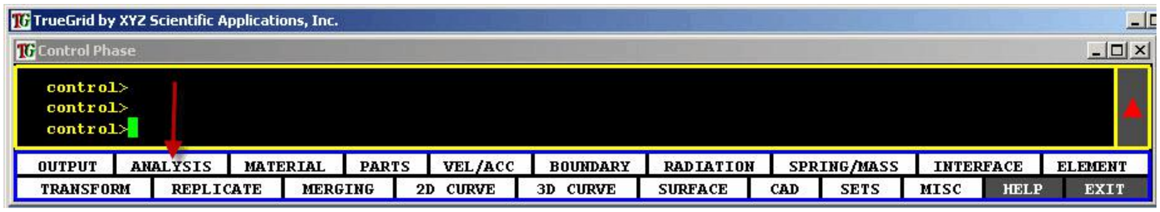


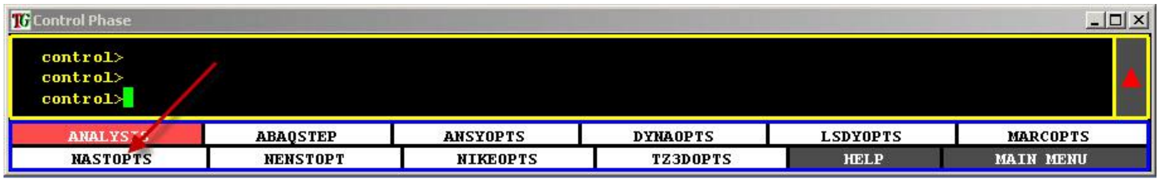




III. Solution Selection

The ***NASTOPTS*** command, which stands for NASTRAN® Analysis Options, can be used to select a solution method and a few other analysis options. The command is found in the ***CONTROL*** phase under the ***ANALYSIS*** menu.





No attempt is made in **True*Grid*®** to mimic the executive and case control features in NASTRAN®. It is anticipated that the control features of NASTRAN® will be frequently modified in a complex model using a text editor. Generating these control statements is contrary to **True*Grid*®**'s primary goal which is to generate the bulk data. The **NASTOPTS** command is an exception to the primary goal, to make it possible to run a simple model directly from **True*Grid*®** to NASTRAN® without using a text editor. If it is essential to run a complex model from **True*Grid*®** to NASTRAN® without using a text editor, then insert the appropriate control statements using the **True*Grid*® VERBATIM** command.

There are many ways to create a node or element set in **True*Grid*®** and for various reasons. Typically, they are used to easily assign conditions or loads on portions of the mesh. Be sure to delete the set after you have used it to assign the condition or loads it was created for. Any node or element sets that remain when to write the output file will be recorded in the file using the case control **set** card.

NASTOPTS Command

NASTOPTS *options* ;

where an *option* can be

SOL *solution_#*	for solution sequence number
where *solution_#* can be	
101	for Statics
103	for Normal Modes
105	for Buckling
106	for Nonlinear Statics
107	for Direct Complex Eigenvalues
108	for Direct Frequency Response
109	for Direct Transient Response
110	for Modal Complex Eigenvalues
111	for Modal Frequency Response
112	for Modal Transient Response
114	for Cyclic Statics
115	for Cyclic Normal Modes
116	for Cyclic Buckling.
118	for Cyclic Direct Frequency Response
128	for Nonlinear Harmonic Response
129	for Nonlinear Transient Response
144	for Static Aeroelastic Response
145	for Aerodynamic Response
146	for Aeroelastic Response
153	for Steady Nonlinear Heat Transfer
159	for Transient Heat Transfer
190	for Database Transfer
200	for Design Optimization
400	for Nonlinear Static and Implicit Transient Analysis
600	for MSC NASTRAN® API into MSC MARC nonlinear
700	for Nonlinear Explicit Transient Analysis
GRAV *id scale xc yc zc*	for Gravity Vector
NLPARM *id #_increments items* ;	for Nonlinear static analysis control
where an *item* can be	
AUTO	for Automatic Stiffness control
ITER *#_iterations*	for Iterative Stiffness control
SEMI	for Semi-iterative Stiffness control
MAXITER *#_iterations*	for Maximum iteration for load increment
U	for Displacement Convergence criterion

EPSU *tol*	for Displacement error tolerance
P	for Load Convergence criterion
EPSP *tol*	for Load error tolerance
W	for Work Convergence criterion
EPSW *tol*	for Work error tolerance
YES	for Every computed load increment Output
NO	for The last load of the subcase Output
ALL	for Every computed and user-specified load increment Output
MAXDIV *limit*	for Divergence condition limit
FSTRESS *fraction*	for Fraction of effective stress
MAXBIS *#_bisections*	for Maximum number of bisections
RTOLB *increment*	for Maximum rotation increment per iteration
MAXQN *max*	for Maximum number of quasi-Newton vectors
MAXLS *max*	for Maximum number of line searches
LSTOL *tolerance*	for Line search tolerance
MAXR *fraction*	for Maximum ratio for arc length adjustments
DT *time*	for Incremental time interval for creep

RANDPS *id sid_1 sid_2 lcd_1 x y*	for Power spectral density specification
RANDT1 *id intervals start max*	for Auto correlation time lag
TSTEP *id steps increments factor*	for Time step definition

TSTEPNL *id increments factor options* ; for Transient analysis controls

where a Nonlinear transient analysis *option* can be

NO *factor*	for Time step increment for output
METHOD *update_opts*	for Method for Dynamic matrix update

where *update_opts* can be

1	for Automatically update stiffness
2	to Update every k steps
3	for Automatically adjust and bisection
KSTEP *#_iterations*	for Iterations before stiffness update
ADJUST *factor*	for Time step automatic adjustment skip factor
MSTEP *#_steps*	for Number of steps for dominant response
RB *time*	for define bounds to keep same time step

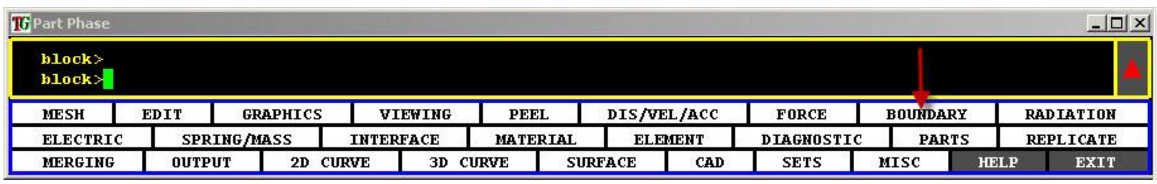
MAXITER *#_iterations*	for Limit of iterations per time step
U	for Displacement Convergence criterion
EPSU *tol*	for Displacement error tolerance
P	for Load Convergence criterion
EPSP *tol*	for Load error tolerance
W	for Work Convergence criterion
EPSW *tol*	for Work error tolerance
MAXDIV *limit*	for Divergence condition limit
FSTRESS *fraction*	for Fraction of effective stress

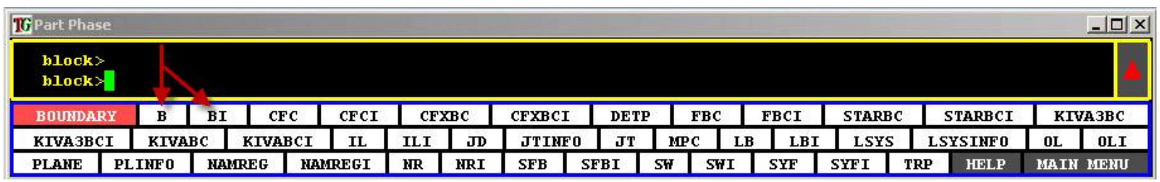
MAXBIS *#_bisections*	for Maximum number of bisections
MAXR *fraction*	for Maximum ratio for arc length adjustments
UTOL *tolerance*	for Displacement increment tolerance
RTOLB *increment*	for Maximum rotation increment per iteration
MAXQN *max*	for Maximum number of quasi-Newton vectors
MAXLS *max*	for Maximum number of line searches

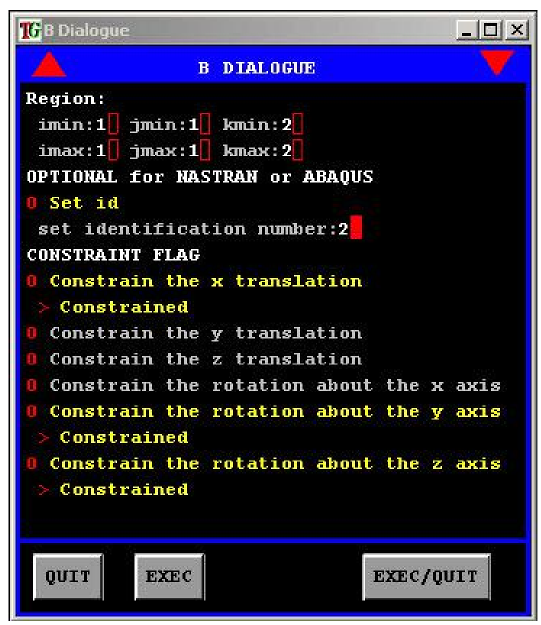
IV. Nodal Constraints

There are several methods to specify nodal constraints. These methods are divided into methods that define single point constraints and multiple point constraints.

Single Point Constraints







The most common form of nodal constraint is the one that is always effective though out the analysis. In the part phase, one can select one or more regions with the B and BI commands respectively. Each of the nodes in the selected region(s) will be assign the specified nodal constraints. For example,

B 1 1 1 1 2 2 dx ry rz

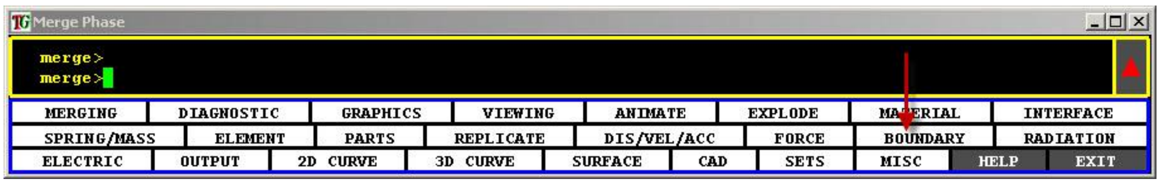
will assign the nodes on the i-face 1 with the symmetry nodal constraint of displacement in x and the rotation in y and z. This type of constraint will show up in the GRID card for NASTRAN® .
If a case is assigned to the constraint by using an additional number before the list of constraint flags,

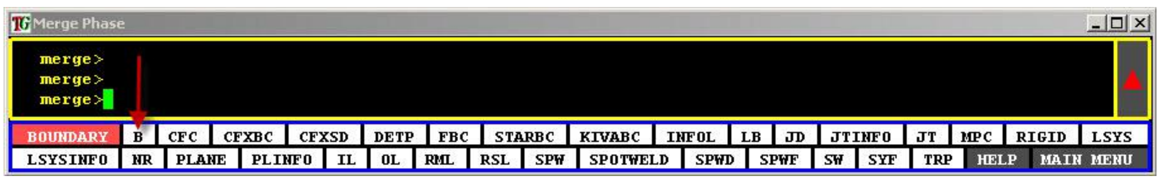
B 1 1 1 1 2 2 sid 5 dx

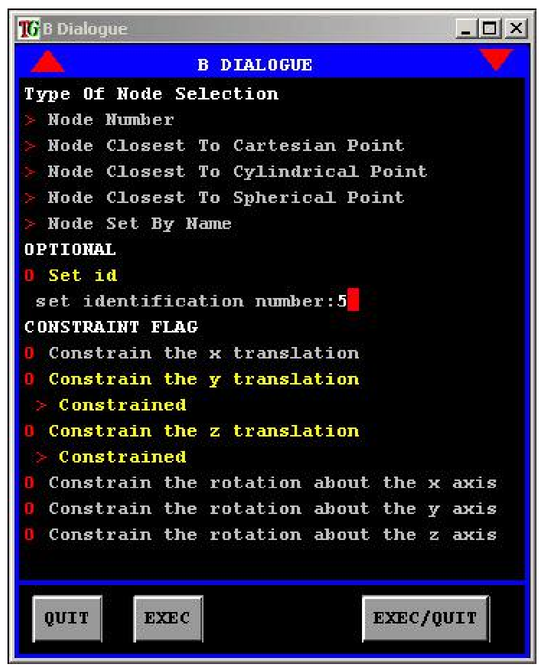
then a SPC1 command will be used in the NASTRAN® output file with a SID of 5. A same holds for the part phase command BI, except the nodes selected for the constraint condition is from a set of regions instead of one region. For example,

BI -1 0 -2;1 2;1 2; dx
BI 1 2;-1 0 -2;1 2; sid 3 dy

where in the first case, the constraints are always effective while in the second case, they are only effective in the subcases that include "SPC = 3".







The B command in the merge phase has 5 options when selecting nodes. Most of the options are self explanatory, except for the "Node Set by Name". A node set can be formed in many ways. One should spend some time exploring the different ways to form a node set. If "ns1" is the name of a node set, then

B nset ns1 rz dx;

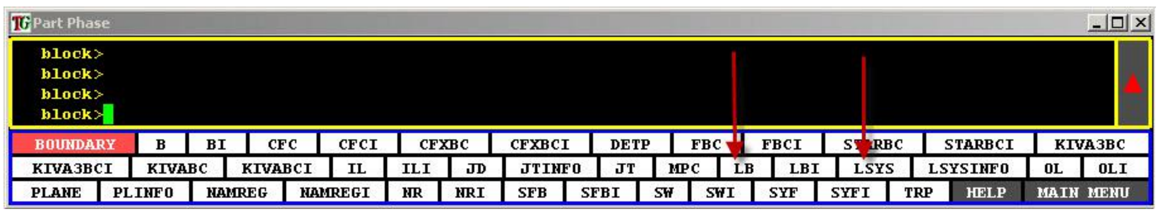
will assign the nodal constraints to each node in the node set called ns1. As in the case of the nodal constraints in the part phase, these constraints are not associated with a single point constraint set identification number, so these constraints will be found as part of the node definition using the GRID command. Alternatively, these constraints can be associated with a set using the sid option. For example

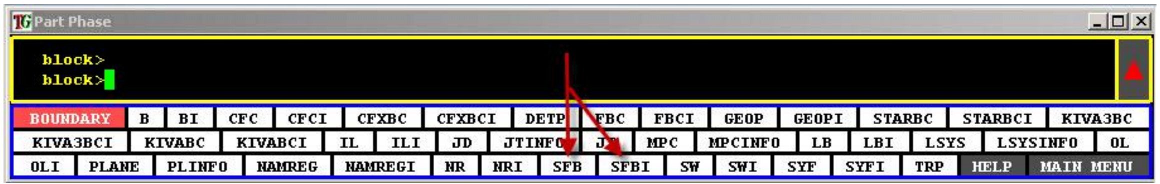
B nset ns1 sid 23 rz 1 dx 1;

will create a SPC1 card in the NASTRAN® output file instead of incorporating the constraint in the GRID card. These constraints will only be active if the “SPC = 23" appears in a subcase.

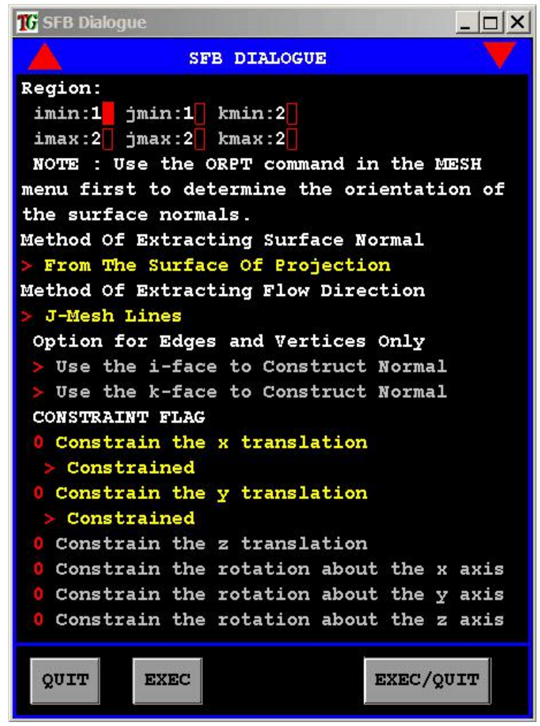
Use the B option of the **CONDITION** command to view graphically the nodes that have been constrained.

To constrain a node in a local coordinate system, use the **LSYS** command to define the local coordinate system. This becomes a CORD2R card in the NASTRAN® file. Then use the **LB** command to assign the local constraint. This constraint will be incorporated into the GRID card for the selected nodes. There are similar commands in the merge phase.





The **SFB/SFBI** commands combine the two steps into one by automatically extracting the local coordinate system from the surface of projection. This is available only in the part phase. In the merge phase, use the SFB option on the **CONDITION** command to see the constraints in the local coordinate systems.



.

Multiple Point Constraints (MPC)

A linear constraint equation defines the relation of degrees of freedom of multiple nodes. Suppose k is the number of nodes included in the equation. The **True*Grid*®** form of this equation is:

$$N_1^{dof} = C_0 + \sum_{i=2}^{k} N_i^{dof} \times C_i \quad \text{where} \quad N_i^{dof} \text{ and } C_i$$

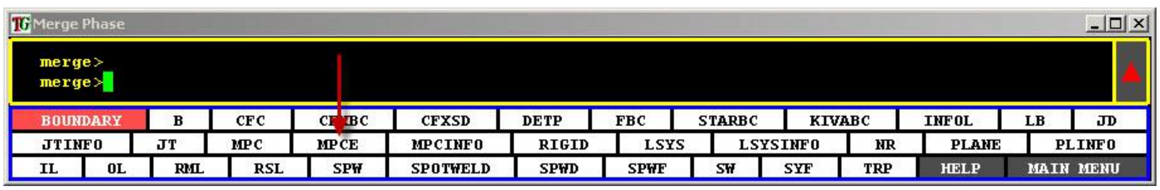
are a degree of freedom of node i and a coefficient of node i, respectively. Note that the coefficient for the first node is 1. The first node is the dependent node and all of the rest of the nodes are the independent nodes. NASTRAN® has the form:

$$\sum_{i=1}^{k} N_i^{dof} \times A_i = 0$$

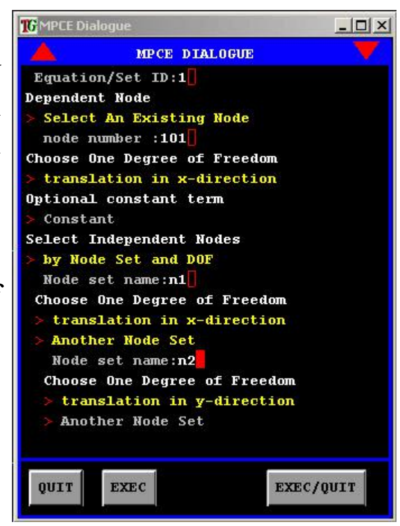
where A_i are the coefficients. So if you know what the coefficients are in the NASTRAN® form, then the coefficients for the **True*Grid*®** form are:

$$C_i = -\frac{A_i}{A_1} \quad \text{and} \quad C_0 = 0$$

There are 3 methods to define a MPC in **True*Grid*®**.



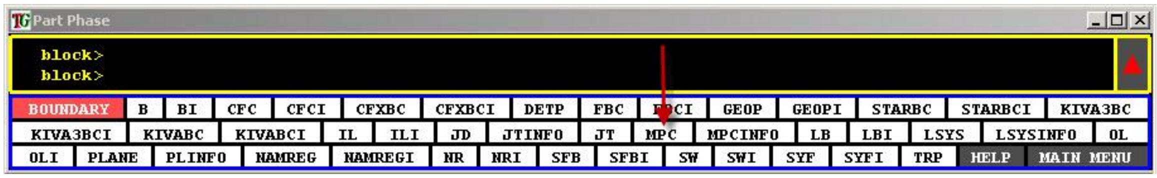
The first method is to use the **MPCE** command in the merge phase. This command is the most complicated because it supports many features. The set identification of the MPC and both coefficient and degree of freedom can be chosen for each node. This method requires either a node set or a list of nodes to form the MPC.



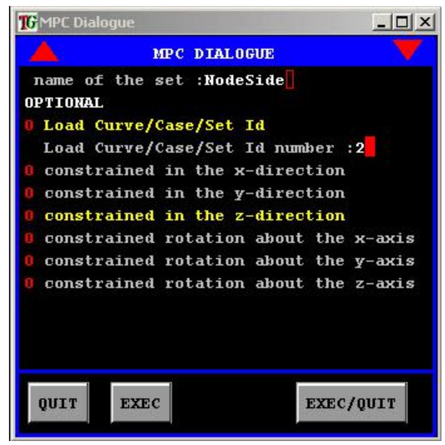
In the example on the right, we start with the dependent node number 101 with x-displacement DOF. Then the node set N1 of independent nodes with x-displacement DOF are added to the equation. Finally, the node set n2 of independent nodes with y-displacement DOF completes the equation. All coefficients are assumed to be 1.

To view the MPCEs, use the **CONDITION** command follwoed by the MPCE option.

The second method is to create a node set. The first node in the set becomes the dependent node. In order to get the dependent node as the first node in the set, use the LA command with the onset command (Label Nodes In An Ordered Node Set) in the merge phase. If the dependent node is not the first node in the set, then use the **MVNSET** command to change the ordering of the nodes.



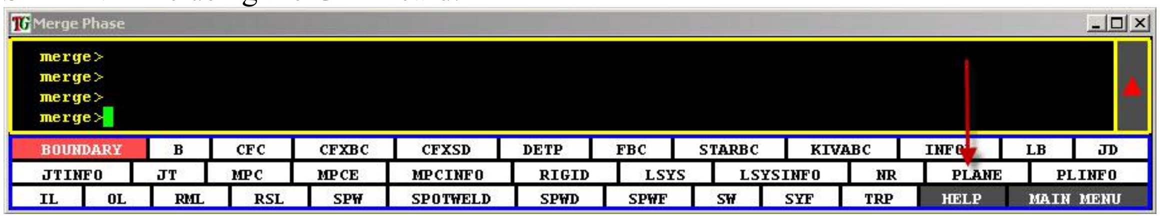
Then use the **MPC** command. The only option in this command is the degrees of freedom. For each degree of freedom selected, there will be a constraint equation. The constant term is 0, the dependent node will have a coefficient of 1.0, and the other coefficients are -1.0. Since it is possible to not assign a set id and it is possible to constrain more than one DOF, set ids will be generated for the **MPC** commands without set id and for the extra DOF.



The third method is to choose the "Shared Degree of Freedoms" in the **JD** command (Joint Definition). Then use the **JT** command (Joint) to specify the node numbers, sequenced from 1 up to 16, that identifies the MPC nodes with the first node being the dependent node. The set identification for these MPCs will be the Joint number that defines the MPC. For each degree of freedom selected, there will be a constraint equation. The constant term is 0 and the other coefficients are 1. The first node in the list becomes the dependent node. In the merge phase, use the **CONDITION** command with the option JT followed by the joint number to see the nodes selected for that MPC.

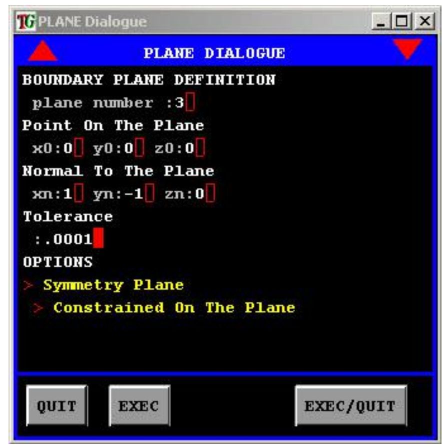
Symmetry Planes

Use the **PLANE** command found in the ***Boundary*** menu to assign nodal constraints to nodes along a symmetry plane. These constraints are not associated with a SID. They are assigned to nodes in the NASTRAN® file using the GRID card.



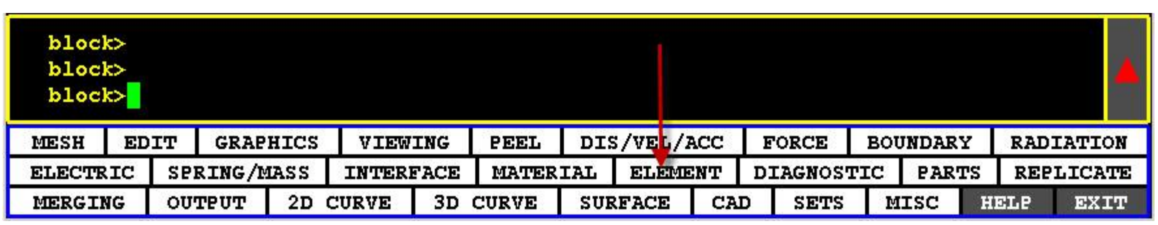
In this example, the symmetry plane is the XY-diagonal through the origin of the global coordinate system.

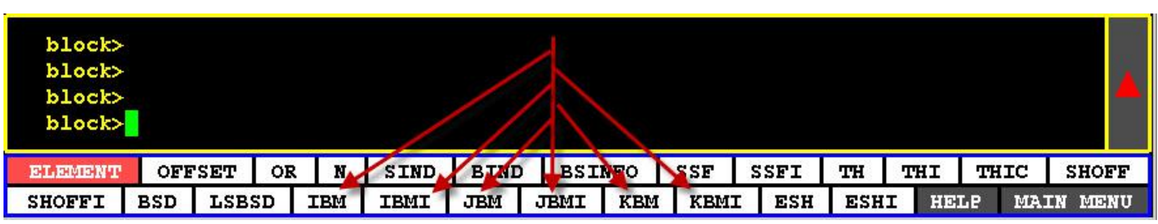
Use the **CONDITION** command with the SY option followed by the plane number to view the nodes that fall within the specified tolerance. This can only be done in the merge phase.



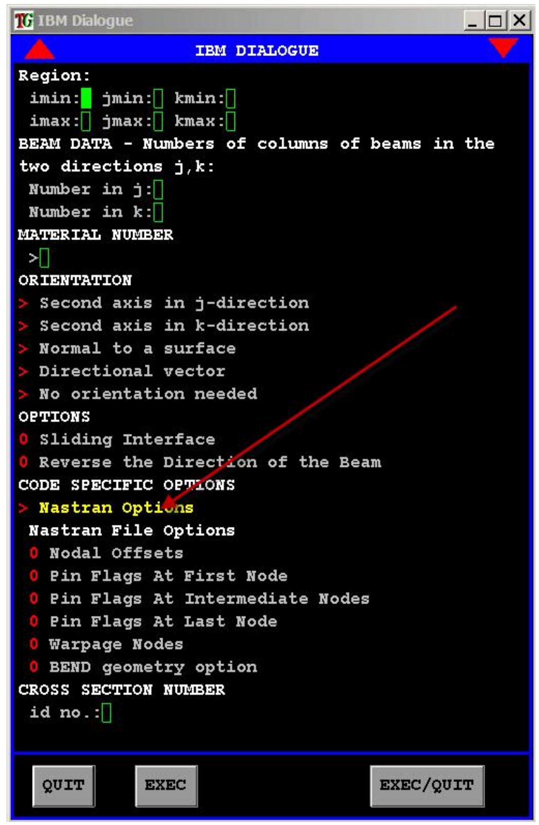
V. 1D Element Properties

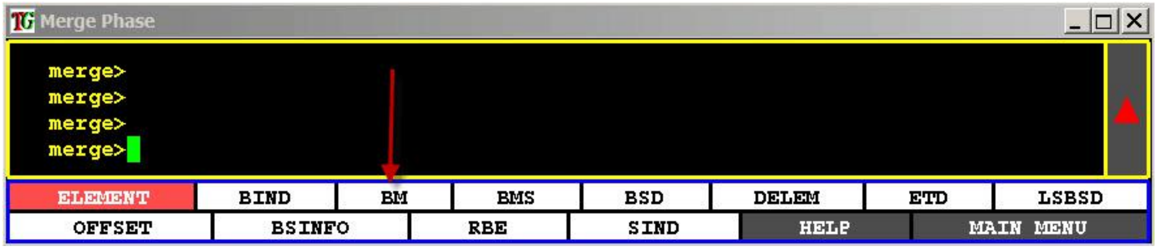
Beams, bends, rods, bars, tubes, and curved pipes (1D elements) can be generated using the **IBM**, **IBMI**, **JBM**, **JBMI**, **KBM**, and **KBMI** commands in the part phase or the **BM** command in the merge phase. Both a material number (see **NASTMATS**) and a cross section number (see **BSD** below) are required when generating these elements.

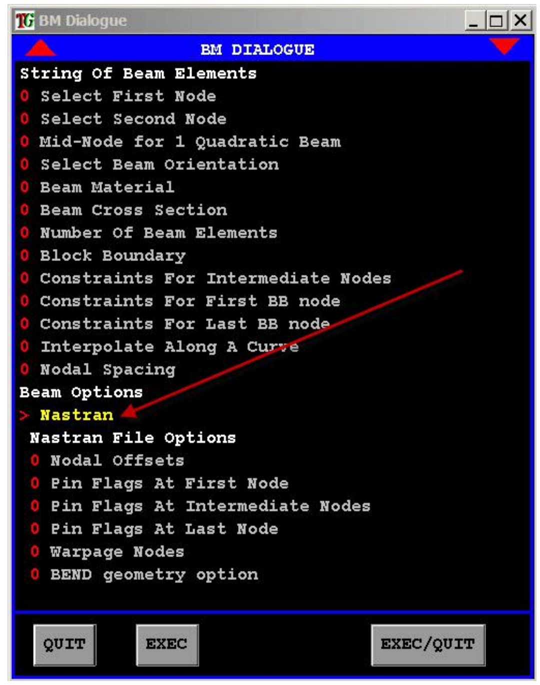




The beam commands in the part phase can assign some properties to the 1D elements being generated. In particular, if the 1D element is a beam, then nodal offsets, pin flags, and warpage points can be specified. If the 1D element is a bend, then the geometry option can be selected. These properties are only assigned to the 1D elements being generated by this instance of the beam command.







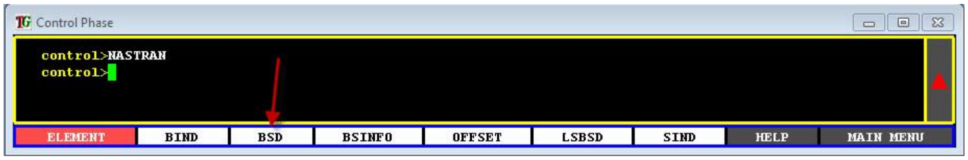
The BM command in the merge phase, used to create 1D elements, also has the options to specify 1D element properties, identical to the beam commands in the part phase.

A cross section is referenced by every beam. Each linear beam is defined by two end nodes and, in the cases for a beam, bend, or bar, by either a third node or a coordinate triple to define the orientation of the cross-sectional local coordinate system of the element.

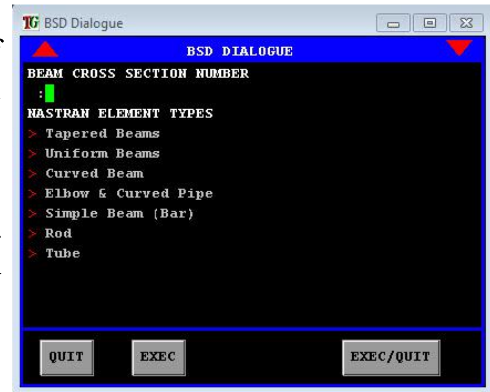
If the **QUADRATIC** or **TRIQUADRATIC** command was issued first, then the 1D elements will be created with an additional node along the mid-section of the element. This intermediate node will be used to define the third node in a CBEAM3 card. In other words, if the property of the quadratic beam is a quadratic beam (PBEAM3), the quadratic feature of the beam will be preserved. Since this is the only 1D element type that can possess a mid node, all other 1D elements created under one of the second order commands (**QUADRATIC** or **TRIQUADRATIC**) will be treated as two 1D elements. The first 1D element will start at the first node and connect to the mid node. The second 1D element will start at the mid node and connect to the last node.

Throughout **True*Grid*®**, the local coordinate system for the beam element is referred to by x, y, and z. They correspond to, respectively, the longitudinal axis of the beam stretched between the first two nodes, the orthogonal component from the first node to the third node/coordinate, and the third direction orthogonal to the first two directions.

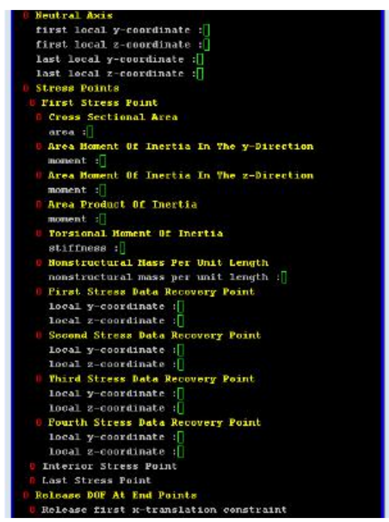
BSD Command



Use the BSD command, within the Element menu, to define the cross sectional properties of 1D elements.



The menu for the BSD command shows there are 7 types of 1D elements supported. If you select the tapered or uniform beam type, you will be defining properties using the NASTRAN® PBEAM card. Selecting the curved beam, elbow, or curved pipe, you will be selecting PBEND. Selecting Bar, Rod, or Tube defines the PBAR, PROD, or the PTUBE respectively. Things get complicated quickly because there are numerous parameters and there are keywords associated with these parameters. Use the Control-V to see the keywords as shown below.



Notice, for example, the keyword **BNA1** in square brackets in the picture in the upper left, which selects tapered beam properties. Also, the stress points parameters (see the picture of the menu cut-out on the above right), which are numerous, are hidden until one selects one of the stress point options. All of the keywords, their meaning, and the associated parameters for the **BSD** command are listed below.

BSD cross_section_# options ;
where an *option* can be:

Tapered beam

BNA1 *t_options* ;	Tapered beam cross section
where *t_options* can be	
SHSTF *y-shear z-shear*	Shear stiffness in y and z
SHRLF *y-shear z-shear*	Shear relief in y and z
NSMMI *moment*	Nonstructural mass at A & B
NSMMI1 *moment*	Nonstructural mass at A
NSMMI2 *moment*	Nonstructural mass at B
WARP *coefficient*	Warping coef. at A and B
WARP1 *coefficient*	Warping coef. at A
WARP2 *coefficient*	Warping coef. at B
CENGRAV y_1 z_1 y_2 z_2	Center of gravity coordinates
NEUAXIS y_1 z_1 y_2 z_2	Neutral axis coordinates
SPCSD *position c_options* ;	Repeat for each location along the beam
where *position* is the relative position from 0 to 1	
where *c_options* can be	
AREA *area*	Cross section area
IYY *moment*	Area moment of inertia
IZZ *moment*	Area moment of inertia
IYZ *moment*	Area moment of inertia
IXX *moment*	Torsional stiffness
MASS *mass*	Nonstructural mass
SDR1 y_1 z_1	y and z for location 1 in cross section
SDR2 y_2 z_2	y and z for location 2 in cross section
SDR3 y_3 z_3	y and z for location 3 in cross section
SDR4 y_4 z_4	y and z for location 4 in cross section
DX1	Release x-displacement at first node
DY1	Release y-displacement at first node
DZ1	Release z-displacement at first node
RX1	Release x-rotation at first node
RY1	Release y-rotation at first node

RZ1	Release z-rotation at first node
DX2	Release x-displacement at last node
DY2	Release y-displacement at last node
DZ2	Release z-displacement at last node
RX2	Release x-rotation at last node
RY2	Release y-rotation at last node
RZ2	Release z-rotation at last node

Beam PBCOMP

BNA2 *t_options* ;

where *t_options* can be	Beam PBCOMP
AREA *area*	Area of beam cross section
IYY *moment*	Area moment of inertia in plane 1
IZZ *moment*	Area moment of inertia in plane 2
IYZ *moment*	Area product of inertia
IXX *moment*	Torsional stiffness parameter.
MASS *mass*	Nonstructural mass per unit length
SHSTF *y z*	Shear stiffness factor in planes 1 & 2
CENGRAV *y z*	The (y,z) coordinates of center of gravity
NEUAXIS *y z*	The (y,z) coordinates of neutral axis.
CSTYPE *type c_options* ;	Symmetry option
where *type* can be	
1	for the default elliptic
2	for symmetry about y and z
3	for symmetry about y
4	for symmetry about z
5	for symmetry about y=z=0
6	for arbitrary
where *c_options* can be	
CSCRV y_1 z_1 ... y_n z_n ;	The (y,z) coordinates of the lumped areas
CSSAR $area_1$ ... $area_n$;	Fraction of total area lumped areas
CSSMAT mat_1 ... mat_n ;	Mat Id of integration point of lumped areas
DX1	Release x-displacement at first node
DY1	Release y-displacement at first node
DZ1	Release z-displacement at first node
RX1	Release x-rotation at first node
RY1	Release y-rotation at first node
RZ1	Release z-rotation at first node
DX2	Release x-displacement at last node
DY2	Release y-displacement at last node
DZ2	Release z-displacement at last node

RX2	Release x-rotation at last node
RY2	Release y-rotation at last node
RZ2	Release z-rotation at last node

Curved beam

BNA3 *t_options* ;	Curved beam
where *t_options* can be	
AREA *area*	Area of the beam cross section
IYY *moment*	Area moments of inertia in plane 1
IZZ *moment*	Area moments of inertia in plane2
IXX *moment*	Torsional stiffness
RB *radius*	Bend radius of the line of centroids.
THETAB *angle*	Arc angle of element
SHSTF *y-shear z-shear*	Shear stiffness factor
SDR1 y_1 z_1	Location 1 for stress data recovery
SDR2 y_2 z_2	Location 2 for stress data recovery
SDR3 y_3 z_3	Location 3 for stress data recovery
SDR4 y_4 z_4	Location 4 for stress data recovery
MASS *mass*	Nonstructural mass
RC *offset*	Radial offset of the geometric centroid
ZC *offset*	Perpendicular offset of the geometric centroid
DELTAN *offset*	Radial offset of the neutral axis

Elbow and curved pipe

BNA4 *t_options* ;	Elbow and curved pipe
where *t_options* can be	
FSI *intensification*	Flag selecting the intensification factors
where *intensification* can be	
1	Formulas for planes 1 and 2
2	ASME code
3	Empirical factors, Welding Research Council
MCSR *radius*	Mean cross-sectional radius
WTH *thickness*	Wall thickness of the curved pipe
IP *pressure*	Internal pressure
RB *radius*	Bend radius
THETAB *angle*	Arc angle of element.
MASS *mass*	Nonstructural mass
RC *offset*	Radial offset of the geometric centroid
ZC *offset*	Perpendicular offset of the geometric centroid

Bar

BNA5 *t_options* ;	Bar
where *t_options* can be	
AREA *area*	Area of bar cross section
IYY *moment*	Area moment of inertia
IZZ *moment*	Area moment of inertia
IYZ *moment*	Area moment of inertia
IXX *moment*	Torsional constant
MASS *mass*	Nonstructural mass
SHSTF *y-shear z-shear*	Area factor for shear
SDR1 y_1 z_1	Location 1 for stress data recovery
SDR2 y_2 z_2	Location 2 for stress data recovery
SDR3 y_3 z_3	Location 3 for stress data recovery
SDR4 y_4 z_4	Location 4 for stress data recovery

Rod

BNA6 *t_options* ;	Rod
where *t_options* can be	
AREA *area*	Area of the rod
IXX *moment*	Torsional constant.
TSC *coefficient*	Coefficient to determine torsional stress.
MASS *mass*	Nonstructural mass

Tube

BNA7 *t_options* ;	Tube
where *t_options* can be	
OD *diameter*	Outside diameter of tube
WTH *thickness*	Thickness of tube
MASS *mass*	Nonstructural mass
OD2 *diameter*	Diameter of tube at second grid point

Quadratic beam

BNA9 *t_options* ;	Tapered beam cross section
where *t_options* can be	
KY f*actor*	Shear effectiveness y-directions
KZ *factor*	Shear effectiveness z-directions
STRESS *location*	Location for stress, strain, force

where *location* can be **GRID** or **GAUSS**

SOB *option*	stress output request at node B
SOC *option*	stress output request at node C

where *option* can be **YES** or **YESA**

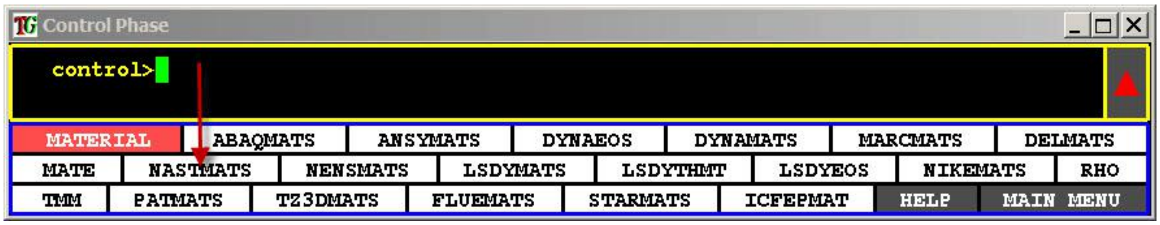
CSDA *parameters* ;	cross section data at node A
CSDB *parameters* ;	cross section data at node B
CSDC *parameters* ;	cross section data at node C

where *parameters* can be any of

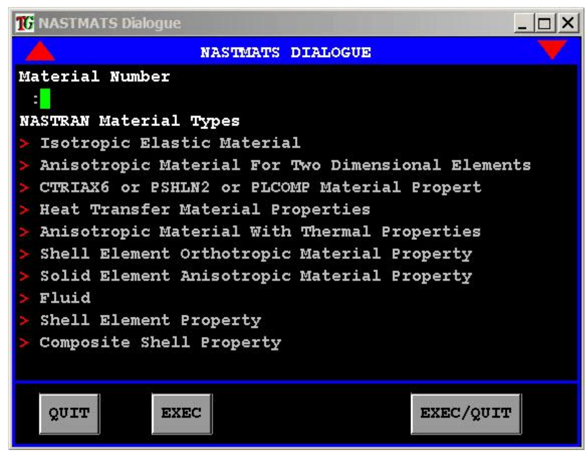
A *area*	area of beam cross section
IZ *moment*	area moment of inertia about z-axis
IY *moment*	area moment of inertia about y-axis
IYZ *inertia*	area product of inertia
J *stiffness*	torsional stiffness
NSM *mass*	nonstructural mass
CY *y*	Stress output y-coordinate at point 1
CZ *z*	Stress output z-coordinate at point 1
DY *y*	Stress output y-coordinate at point 2
DZ *z*	Stress output z-coordinate at point 2
EY *y*	Stress output y-coordinate at point 3
EZ *z*	Stress output z-coordinate at point 3
FY *y*	Stress output y-coordinate at point 4
FZ *z*	Stress output z-coordinate at point 4
NY *y*	Neutral axis y-coordinate
NZ *z*	Neutral axis z-coordinate
MY *y*	Center of gravity y-coordinate
MZ *z*	Center of gravity z-coordinate
NSIY *momemt*	nonstructural mass moments of inertia y-axis
NSIZ *momemt*	nonstructural mass moments of inertia z-axis
NSIYZ *inertia*	nonstructural mass product of inertia
CW *coef*	warping coefficient
W1 *value*	value of warping function at point 1
W2 *value*	value of warping function at point 2
W3 *value*	value of warping function at point 3
W4 *value*	value of warping function at point 4
WY1 *dy*	y-gradient of warping function at point 1
WY2 *dy*	y-gradient of warping function at point 2
WY3 *dy*	y-gradient of warping function at point 3
WY4 *dy*	y-gradient of warping function at point 4
WZ1 *dz*	z-gradient of warping function at point 1
WZ2 *dz*	z-gradient of warping function at point 2
WZ3 *dz*	z-gradient of warping function at point 3
WZ4 *dz*	z-gradient of warping function at point 4

VI. Materials and Element Properties

The command to define a material and element model, **nastmats**, can be found in the MATERIAL menu. Each model below is assigned a number to identify it. Every element type is associated with one or more materials. There are various ways to make this association, which is explained for each model below. Then the number assigned to this association is used to prescribe both the material model and element type to regions in the mesh using the MT, MTI, and MATE commands..



The following is an example of using this dialogue to define a material type 8 followed by defining a shell property, referencing the type 8 material model three times. Since material and element properties a quite complex, it is best to use the dialogue to builds these models.




```
nastmats 3 8 e1 54782 e2 54692 v12 .3 g12 21067
             rho 1.8e+03 a1 -1.2e-06 a2 -1.501e-06 ;
nastmats 4 pshell mid1 3 t .1 mid2 3 t3 234.5
           mid3 3 tst .1 nsm 1.2e+02 z1 .025 z2 .0275 mid4 3 ;
```

which produces the following cards in the output file from **True*Grid***®, in large format.

```
MAT8*                  3              54782.          54692.             0.3
*                 21067.                                               1800.
*             -0.0000012    -0.000001501
PSHELL*                4               3             0.1               3
*                  234.5               3             0.1            120.
*                  0.025          0.0275               3
```

The following defines the syntax for the NASTMATS command.

NASTMATS *material_# material_type options_list* ;

where material type can be 1 through 5, and 8 through 10, PSHELL, and PCOMP followed by an *options_list*.

Material 1 - Isotropic Elastic

When this material model is to be associated with solid elements, select the PSOLID option below and assign values to the parameters that accompany it.

When specifying the material model for a shear panel, select the PSHEAR option below and assign values to the parameters that accompany it. When you are creating other types of shell elements, either use the PSHELL or PCOMP option below, as well as this material model (i.e. use the nastmats command multiple times: once for the element properties and as many times as needed to define all of the material models needed). Note that the material number for this material model will differ from the material number associated with the PSHELL or PCOMP option. This gives you the option to use different material models in defining the PSHELL or PCOMP shell element. Use the material number associated with the PSHELL or PCOMP option when assigning material and element properties to a region of the shell mesh.

When a 1D element is to be associated with this material model, use the BSD command to specify the beam cross section properties. When generating beam elements using the IBM/IBMI, JBM/JMBI, and KMB/KBMI commands in the part phase or the BM command in the merge phase, specify both this material number and the beam cross section number (both material and beam cross section numbers are required in all of the beam generation commands).

Below is the syntax for this material model starting with the material type.

1 for isotropic elastic material type 1

E *modulus*	for Young's Modulus
G modulus	for shear modulus
NU *ratio*	for Poisson's Ratio
RH0 *density*	for mass density
A *expansion*	for thermal expansion coefficient
TREF *temperature*	for reference temperature
GE *damping*	for damping coefficient
ST *stress*	for stress limit for tension
SC *stress*	for stress limit for compression
SS *stress*	for stress limit for shear
MCSID *trans* ;	for material coordinate system

The following are stress dependency parameters.

NLELAST	for non-linear elastic
PLASTIC	for plastic
TID *table_id*	for replacement table number
HSLOPE *slope*	for work hardening slope
HRULE *flag*	for the hardening rule
where *flag* can be	
1	for isotropic
2	for kinematic
3	for isotropic and kinematic hardening
YLDFUN *flag*	for yield function criterion
where *flag* can be	
1	for von Mises
2	for Tresca
3	for Mohr-Coulomb
4	for Drucker-Prager
INYLDP *yield*	for initial yield point
IFANGL *angle*	for internal friction angle

The following specify temperature dependent load curves.

R1 *lcn*	Young's modulus Load Curve
R2 *lcn*	shear modulus Load Curve
R3 *lcn*	Poisson's ratio Load Curve
R4 *lcn*	mass density Load Curve
R5 *lcn*	thermal expansion coefficient Load Curve
R7 *lcn*	damping coefficient Load Curve
R8 *lcn*	stress limit for tension Load Curve
R9 *lcn*	stress limit for compression Load Curve
R10 *lcn*	stress limit for shear Load Curve

The following specifies element properties associated with this material. Specify the element properties if this material will be used with one or more of the following element types.

PSOLID *options*

where the *options* are

CORDM *trans* ;

IN *intensity*

where *intensity* can be

0	for bubble
2	for two
3	for three
STRESS 1	for Gaussian
ISOP 1	for full integration

FCTN *flag*

where *flag* can be

0	for structural element (default)
1	for fluid element
2	for fluid element with frequency
COROT *flag*	Corotational request
where *flag* can be	
0	Do not rotate (default)
1	Local coordinate system rotates with element
PSHEAR *options*	Shear panel properties
where the *options* are	
T *thickness*	Thickness of shear panel
NSM *mass*	Nonstructural mass per unit area
F1 *factor*	Effectiveness factor for extensional stiffness along edges 1-2 and 3-4.
F2 *factor*	Effectiveness factor for extensional stiffness along edges 2-3 and 1-4.

Material 2 - Shell Element Anisotropic

When using this material model, use the PSHELL option below to specify the shell element properties (i.e. use the nastmats command multiple times: once for the element properties and as many times as needed to define all of the material models needed). Note that the material number for this material model will differ from the material number associated with the PSHELL option. This gives you the option to use different material models in defining the PSHELL shell element. Use the material number associated with the PSHELL option when assigning material and element properties to a region of the shell mesh.

Below is the syntax for this material model starting with the material type.

2	for anisotropic material type 2	
	G *g11 g12 g13 g22 g23 g33*	Material matrix terms
	RHO density	Mass density
	A1 expansion	Thermal expansion coefficient
	A2 expansion	Thermal expansion coefficient
	A3 expansion	Thermal expansion coefficient
	TREF temperature	Reference temperature
	GE damping	Damping coefficient
	ST stress	Stress limit for tension
	SC stress	Stress limit for compression
	SS stress	Stress limit for shear
	MCSID *trans* ;	for material coordinate system

The following specify temperature dependent load curves.

R1 *lcn*	Material matrix term G11 load curve
R2 *lcn*	Material matrix term G12 load curve
R3 *lcn*	Material matrix term G13 load curve
R4 *lcn*	Material matrix term G22 load curve
R5 *lcn*	Material matrix term G23 load curve
R6 *lcn*	Material matrix term G33 load curve
R7 *lcn*	Mass density RHO load curve
R8 *lcn*	Thermal coefficient A1 load curve
R9 *lcn*	Thermal coefficient A2 load curve
R10 *lcn*	Thermal coefficient A3 load curve
R12 *lcn*	Damping coefficient GE load curve
R13 *lcn*	Stress limit for tension ST load curve
R14 *lcn*	Stress limit for compression SC load curve
R15 *lcn*	Stress limit for shear SS load curve

Material 3 - Orthotropic 2D Axisymmetric

There is no application for this material model at this time.

Below is the syntax for this material model starting with the material type.

3 for orthotropic material type 3

EX *modulus*	Young's Modulus in x-direction
ETHETA *modulus*	Young's Modulus in θ-direction
EZ *modulus*	Young's Modulus in z-direction
NUXTHETA *ratio*	Poisson's Ratio in xθ-direction
NUTHETAZ *ratio*	Poisson's Ratio in θz-direction
NUZX *ratio*	Poisson's Ratio in zx-direction
RHO *density*	Mass density
GZX *modulus*	Shear modulus
AX *expansion*	Thermal expansion coefficient in x-direction
ATHETA *expansion*	Thermal expansion coefficient in θ-direction
AZ *expansion*	Thermal expansion coefficient in z-direction
TREF *temperature*	Reference temperature
GE *damping*	Damping coefficient

The following specify temperature dependent load curves.

R1 *lcn*	Young's modulus in x-direction load curve
R2 *lcn*	Young's modulus in θ-direction load curve
R3 *lcn*	Young's modulus in z-direction load curve
R4 *lcn*	Shear modulus load curve

R5 *lcn*	Poisson's ratio xθ-direction load curve
R6 *lcn*	Poisson's ratio θz-direction load curve
R7 *lcn*	Poisson's ratio zx-direction load curve
R10 *lcn*	Mass density load curve
R11 *lcn*	Thermal expansion in x-direction load curve
R12 *lcn*	Thermal expansion in θ-direction load curve
R13 *lcn*	Thermal expansion in z-direction load curve
R15 *lcn*	Damping coefficient load curve

The following specify advanced orthotropic, nonlinear elastic load curves.

TEX *lcn*	Young's Molus x-direction EX load curve
TETH *lcn*	Young's Modulus θ-direction ETH load curve
TEZ *lcn*	Young's Modulus z-direction EZ load curve
TNUXTH *lcn*	Poisson's Ratio xθ-direction NUXTH load curve
TNUTHZ *lcn*	Poisson's Ratio θz-direction NUTHZ load curve
TNUZX *lcn*	Poisson's Ratio zx-direction NUZX load curve
TRHO *lcn*	Mass density RHO load curve
TGZX *lcn*	Shear modulus GZX load curve
TAX *lcn*	Thermal expansion coefficient x-direction AX load curve
TATH *lcn*	Thermal expansion coefficient θ-direction ATH load curve
TAZ *lcn*	Thermal expansion coefficient z-direction AZ load curve

Material 4 - Isotropic Thermal

When this material model is to be associated with solid elements, select the PSOLID option below and assign values to the parameters that accompany it.

When a 1D element is to be associated with this material model, use the BSD command to specify the beam cross section properties. When generating beam elements using the IBM/IBMI, JBM/JMBI, and KMB/KBMI commands in the part phase or the BM command in the merge phase, specify both this material number and the beam cross section number (both material and beam cross section numbers are required in all of the beam generation commands).

Below is the syntax for this material model starting with the material type.

4 for isotropic material type 4

K *conductivity*	Thermal conductivity

CP *capacity*	Heat capacity per unit mass at constant pressure
RHO *density*	Density
H *coefficient*	Free convection heat transfer coefficient
MU *viscosity*	Dynamic viscosity
HGEN *coefficient*	Heat generation capability
REFENTH *enthalpy*	Reference enthalpy
TCH *temperature*	Lower temperature limit
TDELTA *temperature*	Total temperature change range
QLAY *heat*	Latent heat of fusion per unit mass

The following specify temperature dependent load curves.

R1 *lcn*	Thermal conductivity load curve
R2 *lcn*	Thermal heat capacity load curve
R4 *lcn*	Free convection heat transfer coefficient load curve
R5 *lcn*	Dynamic viscosity load curve
R6 *lcn*	Internal heat generation load curve

The following specifies the element properties associated with this material. Specify the element properties if the material being defined above will be used with the following element type.

PSOLID *options*

where the *options* are

CORDM *trans* ;

in *intensity*

where *intensity* can be

0	for bubble
2	for two
3	for three
STRESS 1	for Gaussian
ISOP 1	for full integration

FCTN *flag*

where *flag* can be

0	for structural element (default)
1	for fluid element
2	for fluid element with frequency

COROT *flag* — Corotational request

where *flag* can be

0	Do not rotate (default)
1	Local coordinate system rotates with element

Material 5 - Anisotropic Thermal

When this material model is to be associated with solid elements, select the PSOLID option below and assign values to the parameters that accompany it.

When a 1D element is to be associated with this material model, use the BSD command to specify the beam cross section properties. When generating beam elements using the IBM/IBMI, JBM/JMBI, and KMB/KBMI commands in the part phase or the BM command in the merge phase, specify both this material number and the beam cross section number (both material and beam cross section numbers are required in all of the beam generation commands).

Below is the syntax for this material model starting with the material type.

5 for anisotropic material type 5

KXX conductivity	Thermal conductivity KXX
KXY conductivity	Thermal conductivity KXY
KXZ conductivity	Thermal conductivity KXZ
KYY conductivity	Thermal conductivity KYY
KYZ conductivity	Thermal conductivity KYZ
KZZ conductivity	Thermal conductivity KZZ
CP capacity	Heat capacity
RHO density	Density
HGEN heat	Heat generation capability

The following specify temperature dependent load curves.

TKXX conductivity	Thermal conductivity KXX load curve
TKXY conductivity	Thermal conductivity KXY load curve
TKXZ conductivity	Thermal conductivity KXZ load curve
TKYY conductivity	Thermal conductivity KYY load curve
TKYZ conductivity	Thermal conductivity KYZ load curve
TKZZ conductivity	Thermal conductivity KZZ load curve
TCP capacity	Heat capacity load curve
THGEN heat	Heat generation capability load curve

The following specifies the element properties associated with this material. Specify the element properties if the material being defined above will be used with the following element type.

PSOLID *options*

where the *options* are

CORDM *trans* ;

in *intensity*

where *intensity* can be

0 for bubble

2	for two
3	for three
STRESS 1	for Gaussian
ISOP 1	for full integration
FCTN *flag*	
where *flag* can be	
0	for structural element (default)
1	for fluid element
2	for fluid element with frequency
COROT *flag*	Corotational request
where *flag* can be	
0	Do not rotate (default)
1	Local coordinate system rotates with element

Material 8 - Shell Element Orthotropic

When defining this material model, use the PSHELL or PCOMP option below (i.e. use the nastmats command multiple times: once for the element properties and as many times as needed to define all of the material models needed). Note that the material number for this material model will differ from the material number associated with the PSHELL or PCOMP option. This gives you the option to use different material models in defining the PSHELL or PCOMP shell element. Use the material number associated with the PSHELL or PCOMP option when assigning material and element properties to a region of the shell mesh.

Below is the syntax for this material model starting with the material type.

8 for orthotropic material type 8

E1 *modulus*	Modulus of elasticity in longitudinal direction
E2 *modulus*	Modulus of elasticity in lateral direction
V12 *ratio*	Poisson's ratio for uniaxial loading in 1-direction
G12 *modulus*	In-plane shear modulus
G1Z *modulus*	Transverse shear modulus for shear in 1-Z plane
G2Z *modulus*	Transverse shear modulus for shear in 2-Z plane
RHO *density*	Mass density
A1 *expansion*	Thermal expansion coefficient in 1-direction
A2 *expansion*	Thermal expansion coefficient in 2-direction

TREF *temperature*	Reference temperature for calculation of thermal loads
XT *stress/strain*	Allowable stresses or strains in tension in the longitudinal direction
XC *stress/strain*	Allowable stresses or strains in compression in the longitudinal direction
YT *stress/strain*	Allowable stresses or strains in tension in the lateral direction
YC *stress/strain*	Allowable stresses or strains in compression in the lateral direction
S *stress/strain*	Allowable stress or strain for in-plane shear
GE *damping*	Structural damping coefficient
f12 *interaction*	Interaction term in the tensor polynomial theory of Tsai-Wu
STRN	Indicates Xt, Xc, Yt, Yc, and S are strain allowables

The following specify temperature dependent load curves.

R1 *lcn*	Modulus of elasticity in longitudinal direction load curve
R2 *lcn*	Modulus of elasticity in lateral direction load curve
R3 *lcn*	Poisson's ratio for uniaxial loading in 1-direction load curve
R4 *lcn*	In-plane shear modulus load curve
R5 *lcn*	Transverse shear modulus for shear in 1-Z plane load curve
R6 *lcn*	Transverse shear modulus for shear in 2-Z plane load curve
R7 *lcn*	Mass density RHO load curve
R8 *lcn*	Thermal expansion coefficient in 1-direction load curve
R9 *lcn*	Thermal expansion coefficient in 2-direction load curve
R11 *lcn*	Allowable stresses or strains in tension in the longitudinal direction load curve
R12 *lcn*	Allowable stresses or strains in compression in the longitudinal direction load curve
R13 *lcn*	Allowable stresses or strains in tension in the lateral direction load curve
R14 *lcn*	Allowable stresses or strains in compression in the lateral direction load curve

R15 *lcn*	Allowable stress or strain for in-plane shear load curve
R16 *lcn*	Structural damping coefficient load curve
R17 *lcn*	Interaction term in the tensor polynomial theory of Tsai-Wu load curve

The following specify Advanced Orthotropic, Nonlinear Elastic Material

TE1 *lcn*	Modulus of elasticity in longitudinal direction load curve
TE2 *lcn*	Modulus of elasticity in lateral direction load curve
TNU12 *lcn*	Poisson's ratio for uniaxial loading in 1-direction load curve
TG12 *lcn*	In-plane shear modulus load curve
TG1Z *lcn*	Transverse shear modulus for shear in 1-Z plane load curve
TG2Z *lcn*	Transverse shear modulus for shear in 2-Z plane load curve
TRHO *lcn*	Mass density load curve
TA1 *lcn*	Thermal expansion coefficient in 1-direction load curve
TA2 *lcn*	Thermal expansion coefficient in 2-direction load curve

Material 9 - Solid Element Anisotropic

This material model is to be associated with solid elements. Select the PSOLID option below and assign values to the parameters that accompany it.

Below is the syntax for this material model starting with the material type.

9 for anisotropic material type 9

G11 *term*	Matrix element
G12 *term*	Matrix element
G13 *term*	Matrix element
G14 *term*	Matrix element
G15 *term*	Matrix element
G16 *term*	Matrix element
G22 *term*	Matrix element
G23 *term*	Matrix element
G24 *term*	Matrix element
G25 *term*	Matrix element

G26 *term*	Matrix element
G33 *term*	Matrix element
G34 *term*	Matrix element
G35 *term*	Matrix element
G36 *term*	Matrix element
G44 *term*	Matrix element
G45 *term*	Matrix element
G46 *term*	Matrix element
G55 *term*	Matrix element
G56 *term*	Matrix element
G66 *term*	Matrix element
RHO *density*	Mass density
A1 *expansion*	Thermal expansion x-coefficient
A2 *expansion*	Thermal expansion y-coefficient
A3 *expansion*	Thermal expansion z-coefficient
A4 *expansion*	Thermal expansion xy-coefficient
A5 *expansion*	Thermal expansion yz-coefficient
A6 *expansion*	Thermal expansion zx-coefficient
TREF *temperature*	Reference temperature
GE *damping*	Structural element damping coefficient

The following specify temperature dependent load curves.

R1 *lcd*	Load curve for Matrix element G11
R2 *lcd*	Load curve for Matrix element G12
R3 *lcd*	Load curve for Matrix element G13
R4 *lcd*	Load curve for Matrix element G14
R5 *lcd*	Load curve for Matrix element G15
R6 *lcd*	Load curve for Matrix element G16
R7 *lcd*	Load curve for Matrix element G22
R8 *lcd*	Load curve for Matrix element G23
R9 *lcd*	Load curve for Matrix element G24
R10 *lcd*	Load curve for Matrix element G25
R11 *lcd*	Load curve for Matrix element G26
R12 *lcd*	Load curve for Matrix element G33
R13 *lcd*	Load curve for Matrix element G34
R14 *lcd*	Load curve for Matrix element G35
R15 *lcd*	Load curve for Matrix element G36
R16 *lcd*	Load curve for Matrix element G44
R17 *lcd*	Load curve for Matrix element G45
R18 *lcd*	Load curve for Matrix element G46
R19 *lcd*	Load curve for Matrix element G55
R20 *lcd*	Load curve for Matrix element G56
R21 *lcd*	Load curve for Matrix element G66

R22 *lcd*	Load curve for the mass density
R23 *lcd*	Load curve for thermal expansion x-coef. A1
R24 *lcd*	Load curve for thermal expansion y-coef. A2
R25 *lcd*	Load curve for thermal expansion z-coef. A3
R26 *lcd*	Load curve for thermal expansion xy-coef. A4
R27 *lcd*	Load curve for thermal expansion yz-coef. A5
R28 *lcd*	Load curve for thermal expansion zx-coef. A6
R30 *lcd*	Load curve for damping coef. GE

PSOLID *options*

where the *options* are

CORDM *trans* ;

in *intensity*

where *intensity* can be

0	for bubble
2	for two
3	for three
STRESS 1	for Gaussian
ISOP 1	for full integration

FCTN *flag*

where *flag* can be

0	for structural element (default)
1	for fluid element
2	for fluid element with frequency

COROT *flag* — Corotational request

where *flag* can be

0	Do not rotate (default)
1	Local coordinate system rotates with element

Material 10 - Fluid

This material model is to be associated with solid elements. Select the PSOLID option below and assign values to the parameters that accompany it.

Below is the syntax for this material model starting with the material type.

10	for fluid material type 10	
	BULK *modulus*	Bulk modulus
	RHO *density*	Mass density
	SS *speed*	Speed of sound
	GE *coefficient*	Fluid element damping coefficient
	ALPHA *coefficient*	Normalized admittance coefficient for porous material
	PSOLID *options*	
	where the *options* are	
	CORDM *trans* ;	
	in *intensity*	
	where *intensity* can be	
	0	for bubble
	2	for two
	3	for three
	STRESS 1	for Gaussian
	ISOP 1	for full integration
	FCTN *flag*	
	where *flag* can be	
	0	for structural element (default)
	1	for fluid element
	2	for fluid element with frequency
	COROT *flag*	Corotational request
	where *flag* can be	
	0	Do not rotate (default)
	1	Local coordinate system rotates with element

PSHELL Shell Element Property

When defining this multi-material element model, use the nastmats command multiple times: once for the element properties and as many times as needed to define all of the material models for this element model. Note that the material number for the material models will differ from the number for element model associated with this PSHELL option. Use this element model number when assigning material and element properties to a region of the shell mesh.

Below is the syntax for this multi-material element model starting with the element type.

PSHELL	for shell element property	
	MID1	Material ID for the membrane
	T *thickness*	Default membrane thickness
	MID2 *material_id*	Material ID for bending
	T3 *stiffness*	Bending moment of inertia ratio,
	MID3 *material_id*	Material ID for transverse shear
	TST *ratio*	Transverse shear thickness ratio
	NSM *ratio*	Nonstructural mass per unit area
	Z1 *distance*	First fiber distance for stress calculations
	Z2 *distance*	Second fiber distance for stress calculations
	MID4 *material_id*	Material ID for membrane-bending coupling.

PCOMP Shell Element Property

When defining this multi-material element model, use the nastmats command multiple times: once for the element properties and as many times as needed to define all of the material models for this element model. Note that the material number for the material models will differ from the number for element model associated with this PCOMP option. Use this element model number when assigning material and element properties to a region of the shell mesh.

Below is the syntax for this multi-material element model starting with the element type.

PCOMP	for shell element property	
	PCZ0 *distance*	Distance from the reference plane to the bottom surface
	PCNSM *mass*	Nonstructural mass per unit area
	PCSB *stress*	Allowable shear stress of the bonding material
	PCFT *flag*	Failure theory
	where the *flag* can be	
	0	for Hill
	1	for Hoffman

2	for Tsai-Wu
3	for Maximum strain
PCTREF *temperature*	Reference temperature.
PCGE *coefficient*	Damping coefficient
PCLAM *lamination*	Laminate Options
where *lamination* can be	
0	All plies must be specified
1	Only plies on one side of the element centerline are specified
2	All plies must be specified, but only membrane terms (MID1 on the derived PSHELL entry) are computed.
3	All plies must be specified, but only bending terms (MID2 on the derived PSHELL entry) are computed
4	All plies must be specified, stacking sequence is ignored
5	All plies must be specified, with the last ply specifying core properties and the previous plies specifying face sheet properties
PCLIST *list*	Ply list

where *list* is

$mat_1\ t_1\ orient_1\ flag_1\ \ldots\ mat_n\ t_n\ orient_n\ flag_n$

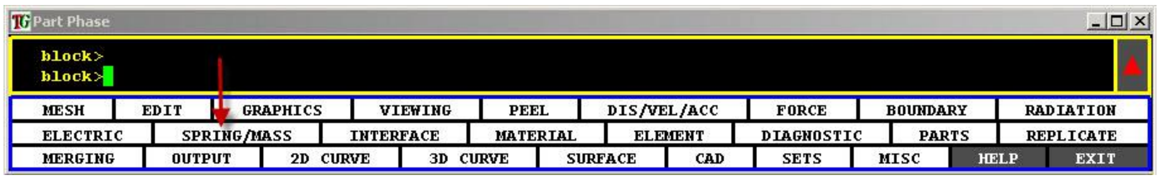
where mat_i is a material number for ply i
where t_i is the thickness of ply i
where $orient_i$ is the orientation of ply i
where the $flag_i$ is for ply i and can be

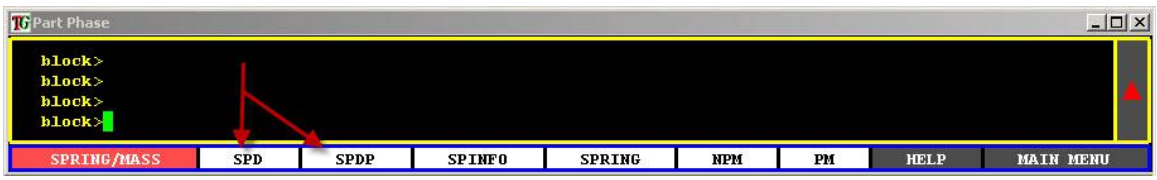
0	for no output (default)
1	for write the output

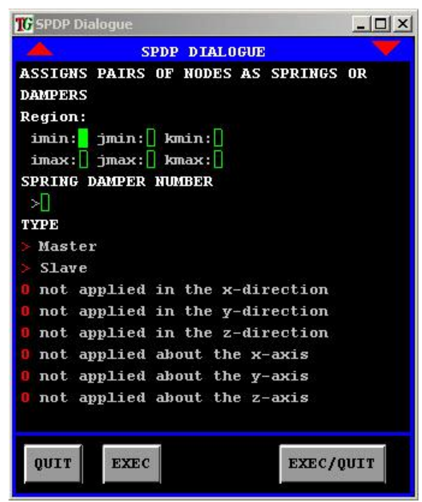
VII. Springs, Dampers, and Point Masses

Springs and Dampers

The **SPD** command assigns properties to a set of springs/dampers. It can be found in the ***SPRING/MASS*** menu. The **SPDP** command creates a set of springs/dampers to connect two independent parts at an interface. Properties of these springs/dampers are set using the **SPD** command. The two faces of the interface are arbitrarily identified as master and slave. This command is found only in the ***SPRING/MASS*** menu in the part phase.







NOTE: The *id_#* in the **SPDP** command and the *spring_#* in the **SPRING** command are not related.

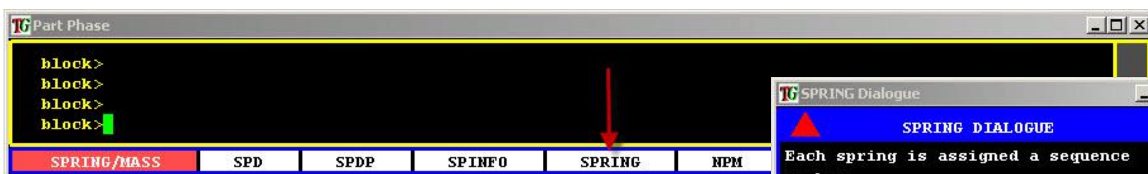
SPD *spring/damper_#* **SE** *elastic_property damping stress*

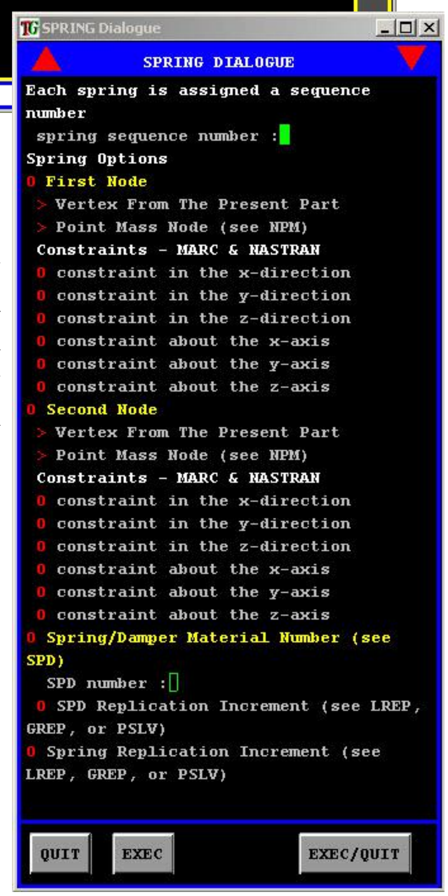
SPDP *i1 j1 k1 i2 j2 k2 id_#* **M** *scale options* ;	Defines the master side of the interface
or	
SPDP *i1 j1 k1 i2 j2 k2 id_#* **S** *options* ;	Defines the slave side of the interface

where an *option* can be

DX	Release displacement in the x-direction
DY	Release displacement in the y-direction
DZ	Release displacement in the z-direction
RX	Release rotation about the x-axis
RY	Release rotation about the y-axis
RZ	Release rotation about the z-axis

The **SPRING** command is used to define a single spring/damper. It is found in the ***SPRING/MASS*** menu. The **SPRING** command is found in both the part and merge phase. Nodes can be selected for the spring/damper either using the reduced indices of the part phase or the node number of any node in the merge phase. This is why there are two versions of the **SPRING** command below.





A numbered point mass can be used as a node in a spring definition. Since a spring defined in the part phase is a component of the part being generated, the spring is replicated if the part is replicated. The spring number and the spring material number of the replicated spring is incremented and these increments can be specified in this dialogue.

Use the SP option of the CONDITION command to view the springs and dampers in the merge phase.

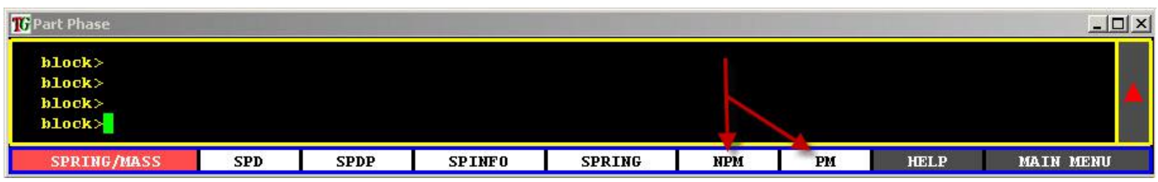
SPRING *spring_# options* ;	**SPRING** command in the part phase
where an *option* can be:	
SINC *increment*	Spring number increment in part replication
V1 *i j k*	Vertex indices for node 1 of spring
PM1 *pointmass_#*	Point mass number for node 1 of spring
PMINC1 *increment*	Point mass increment for node 1 in part replication
DX1	Release displacement for node 1 in the x-direction
DY1	Release displacement for node 1 in the y-direction
DZ1	Release displacement for node 1 in the z-direction
RX1	Release rotation for node 1 about the x-axis
RY1	Release rotation for node 1 about the y-axis
RZ1	Release rotation for node 1 about the z-axis
V2 *i j k*	Vertex indices for node 2 of spring
PM2 *pointmass_#*	Point mass number of node 2 of spring
PMINC2 *increment*	Point mass increment for node 2 in part replication
DX2	Release displacement for node 2 in the x-direction
DY2	Release displacement for node 2 in the y-direction
DZ2	Release displacement for node 2 in the z-direction
RX2	Release rotation for node 2 about the x-axis
RY2	Release rotation for node 2 about the y-axis
RZ2	Release rotation for node 2 about the z-axis
SDDN *spd_#*	Material *spring/damper_#* from the **SPD** command
SMINC *increment*	*spring/damper_#* increment in part replication

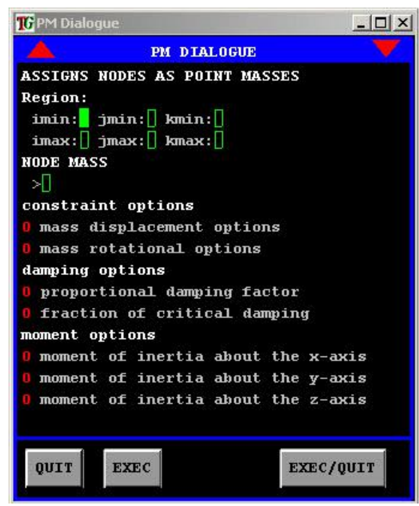
SPRING *spring_# options* ;	**SPRING** command in the merge phase
where an *option* can be:	
N1 *node_#*	Node 1 of spring
PM1 *pointmass_#*	Point mass number for node 1 of spring
DX1	Release displacement for node 1 in the x-direction
DY1	Release displacement for node 1 in the y-direction
DZ1	Release displacement for node 1 in the z-direction
RX1	Release rotation for node 1 about the x-axis
RY1	Release rotation for node 1 about the y-axis
RZ1	Release rotation for node 1 about the z-axis
N2 *node_#*	Node 2 of spring
PM2 *pointmass_#*	Point mass number of node 2 of spring
DX2	Release displacement for node 2 in the x-direction
DY2	Release displacement for node 2 in the y-direction
DZ2	Release displacement for node 2 in the z-direction
RX2	Release rotation for node 2 about the x-axis
RY2	Release rotation for node 2 about the y-axis

RZ2 Release rotation for node 2 about the z-axis

Point Masses

There are two ways to define a point mass in **True*Grid*®**. The **PM** command selects a node in the mesh and assigns a mass to that node. The **NPM** command creates a new node and assigns a mass to that node. It is assumed that the new node from the **NPM** command will be tied to the global mesh in some manner, such as with a beam or spring. The end result is a PMASS and CMASS1 card defining the point mass within NASTRAN®. Both commands are found in the ***SPRING/MASS*** menu in both the part and merge phases. The identification number in the **NPM** is used as an index into a table. Do not use arbitrarily large numbers since this will cause the table to grow very large and become inefficient.





PM *i1 j1 k1 i2 j2 k2 node_mass options* ; Part phase nodal point mass

where an *option* can be :

MDX Release mass DOF in x-displacement
MDY Release mass DOF in y-displacement
MDZ Release mass DOF in z-displacement
MRX Release mass DOF in x-rotation
MRY Release mass DOF in y-rotation
MRZ Release mass DOF in z-rotation

NPM *id_# x y z mass options* ;	Part phase NPM new node point mass
where an *option* can be :	
INC *increment*	Increment in the *id_#* when the part is replicated
DX	Nodal constraint in the x-direction
DY	Nodal constraint in the y-direction
DZ	Nodal constraint in the z-direction
RX	Nodal constraint about the x-axis
RY	Nodal constraint about the y-axis
RZ	Nodal constraint about the z-axis
MDX	Release mass DOF in x-displacement
MDY	Release mass DOF in y-displacement
MDZ	Release mass DOF in z-displacement
MRX	Release mass DOF in x-rotation
MRY	Release mass DOF in y-rotation
MRZ	Release mass DOF in z-rotation

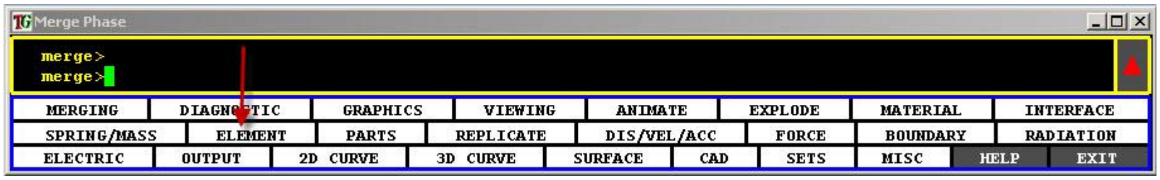
PM *node_# node_mass options* ;	Merge phase nodal point mass
where an *option* can be :	
MDX	Release mass DOF in x-displacement
MDY	Release mass DOF in y-displacement
MDZ	Release mass DOF in z-displacement
MRX	Release mass DOF in x-rotation
MRY	Release mass DOF in y-rotation
MRZ	Release mass DOF in z-rotation

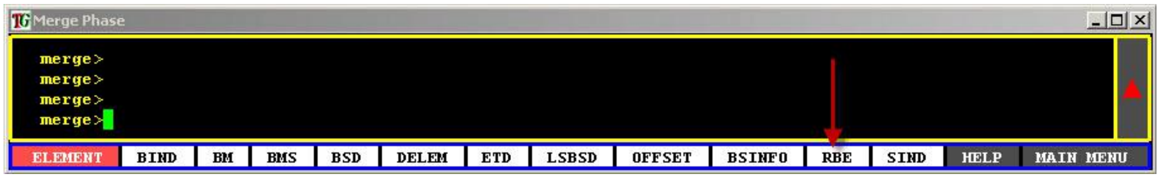
NPM *id_# x y z mass options* ;	Merge phase NPM new node point mass
where an *option* can be :	
DX	Nodal constraint in the x-direction
DY	Nodal constraint in the y-direction
DZ	Nodal constraint in the z-direction
RX	Nodal constraint about the x-axis
RY	Nodal constraint about the y-axis
RZ	Nodal constraint about the z-axis
MDX	Release mass DOF in x-displacement
MDY	Release mass DOF in y-displacement
MDZ	Release mass DOF in z-displacement
MRX	Release mass DOF in x-rotation
MRY	Release mass DOF in y-rotation
MRZ	Release mass DOF in z-rotation

Use the PM option of the **CONDITION** command to view the point masses while in the merge phase.

VIII. Rigid Bodies

The **RBE** command in the merge phase under ***ELEMENTS*** can be used to generate rigid bodies (Rigid Body Element - RBE2, Rigid Triangular Plate - RTRPLT, Interpolation Constraint Element - RBE3, Rigid Bar - RBAR, and Rigid Pin-Ended Element Connection - RROD).





RBE *element_# type parameters*
where the *parameters* vary depending on the *type*

The *type* and associated *parameters* are listed below.

RROD - Rigid Pin-Ended Element Connection

RROD *option node dof ; node dof ;* Rigid Pin-Ended Element Connection
where the *option* can be
ALPHA *α* thermal expansion coefficient
where only one *dof* can be selected from

MDX	dependent translation in x
MDY	dependent translation in y
MDZ	dependent translation in z

RBAR - Rigid Bar

RBAR *option node dof ; node dof* ; Rigid Bar

where the *option* can be

ALPHA *α*	thermal expansion coefficient

where a constraint can be formed using

NDX	independent translation in x
NDY	independent translation in y
NDZ	independent translation in z
NRX	independent rotation in x
NRY	independent rotation in y
NRZ	independent rotation in z
MDX	dependent translation in x
MDY	dependent translation in y
MDZ	dependent translation in z
MRX	dependent rotation in x
MRY	dependent rotation in y
MRZ	dependent rotation in z

RTRPLT - Rigid Triangular Plate

RTRPLT *option node dof ; node dof ; node dof* ; Rigid Triangular Plate

where the *option* can be

ALPHA *α*	thermal expansion coefficient

where a *dof* can be formed using

NDX	independent translation in x
NDY	independent translation in y
NDZ	independent translation in z
NRX	independent rotation in x
NRY	independent rotation in y
NRZ	independent rotation in z
MDX	dependent translation in x
MDY	dependent translation in y
MDZ	dependent translation in z
MRX	dependent rotation in x
MRY	dependent rotation in y
MRZ	dependent rotation in z

RBE2 - Rigid Body Element, Form 3

RBE2 *option node dof ; list_nodes* ; Rigid Body Element, Form 3

where the *option* can be

ALPHA *α*	thermal expansion coefficient

where a *dof* can be formed using

MDX	dependent translation in x
MDY	dependent translation in y
MDZ	dependent translation in z
MRX	dependent rotation in x
MRY	dependent rotation in y
MRZ	dependent rotation in z

RBE3 - Interpolation Constraint Element

RBE3 *option node dof ;[weight dof ; list_nodes ;];* Interpolation Constraint Element

where the *option* can be

ALPHA *α*	thermal expansion coefficient

where a *dof* can be formed using

MDX	DOF translation in x
MDY	DOF translation in y
MDZ	DOF translation in z
MRX	DOF rotation in x
MRY	DOF rotation in y
MRZ	DOF rotation in z

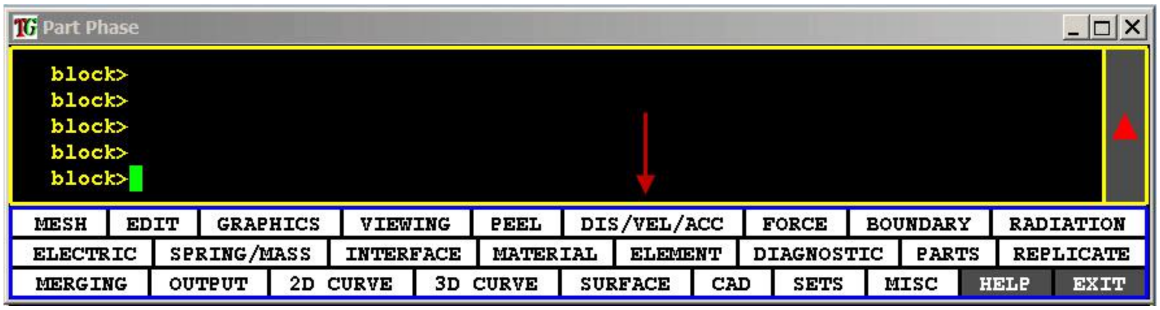
where a *node* can be selected by it's number or by coordinates

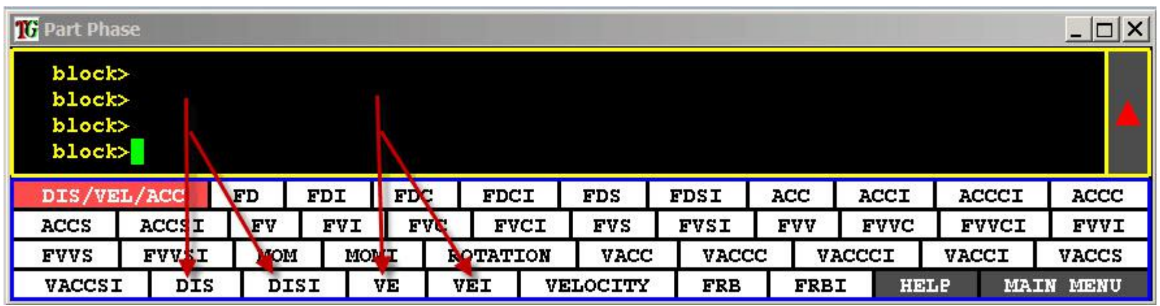
NODE *node_#*	Choose node by node number
RT *x y z*	Choose the node closest to Cartesian coordinates
CY *ρ θ z*	Choose the node closest to cylindrical coordinates
SP *ρ θ φ*	Choose the node closest to spherical coordinates

IX. Initial Conditions and Loads

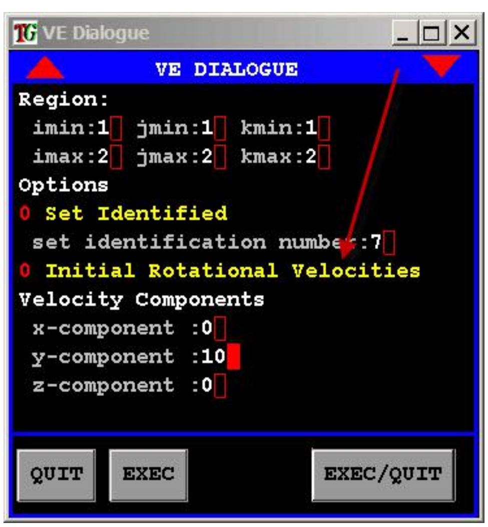
Initial Velocities and Displacements

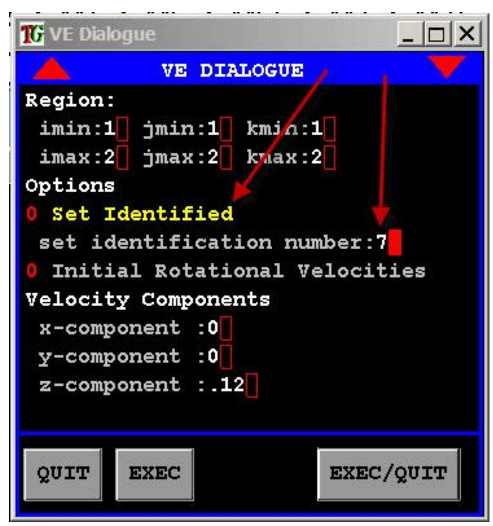
Initial velocities and initial displacements can be assigned to portions of the mesh using the VE or VEI and the DIS or DISI commands, respectively, found under the DIS/VEL/ACC menu.





These **True*Grid*®** commands produce the NASTRAN® TIC cards. Be sure to select a Set ID in the dialogue box that pops up, or the Set ID will default to zero, which may not have meaning in NASTRAN® (right).



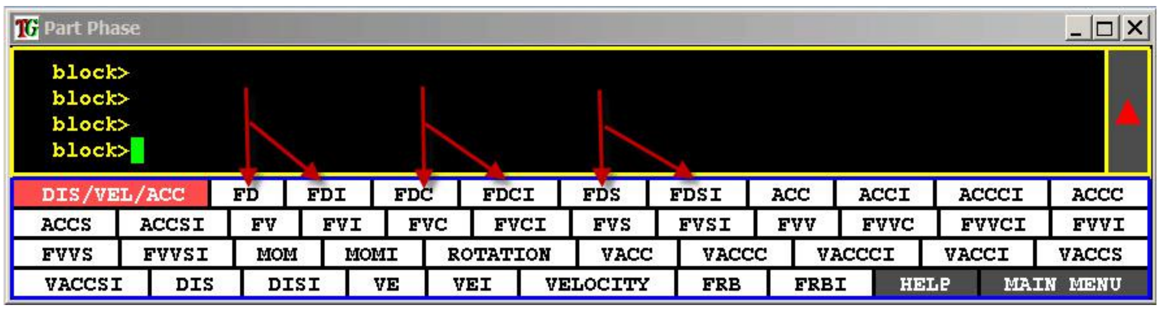


To specify rotational initial conditions, click on "Initial Rotational Velocities (Displacements)" so that it is yellow (left). The ve and dis commands are available in the merge phase using nodal sets instead of regions.

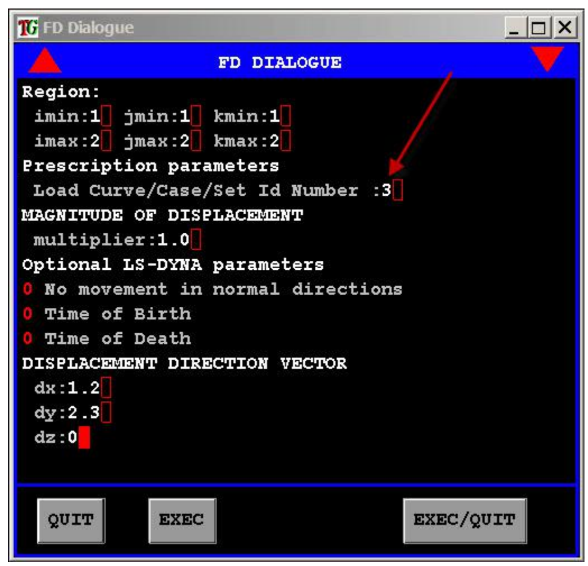
Use the VE or DIS option of the **CONDITION** command in the merge phase to view the initial velocities or initial displacements in the merge phase.

Fixed Displacement

Nodal displacement (SPC card) can be specified in the part phase using the **True*Grid*®** FD/FDI, FDC/FDCI, and FDS/FDSI commands. They are found in the DIS/VEL/ACC menu.



There are 6 variations of this command used to define fixed nodal displacements. On the right is the FD dialogue as an example. Be sure to select a Set ID. You can ignore the 3 optional LS-DYNA parameters.



The FD command is the simplest form. The FDI will require understanding index progressions which allow you to select multiple regions of the mesh in one command.

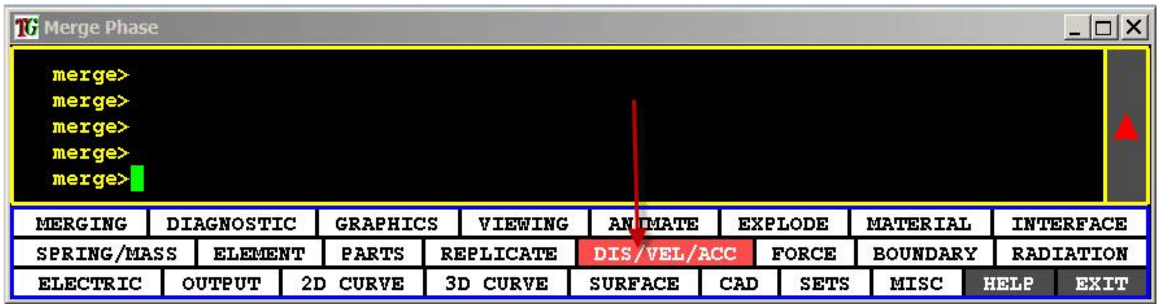
The FDC/FDCI commands define the displacement in cylindrical coordinates. The FDS/FDSI commands define the displacement in spherical coordinates.

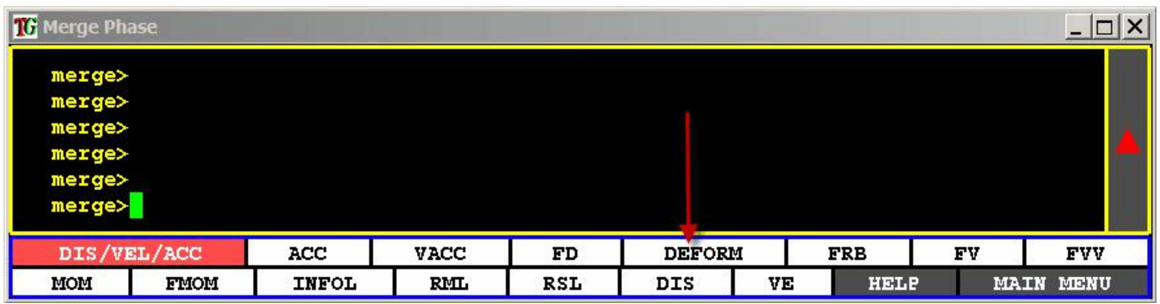
The FD command can also be found in the merge phase. Different methods are available in choosing the nodes to be displaced, including the need to create of a node set.

Use the FD option of the **CONDITION** command to view the fixed displacements in the merge phase.

Beam Element Displacement

The DEFORM command in **True*Grid*®** will generate the NASTRAN® DEFORM card to prescribe deformation to beam elements. This command is only found in the merge phase, under the DIS/VEL/ACC menu. This command usually requires building a beam element set.

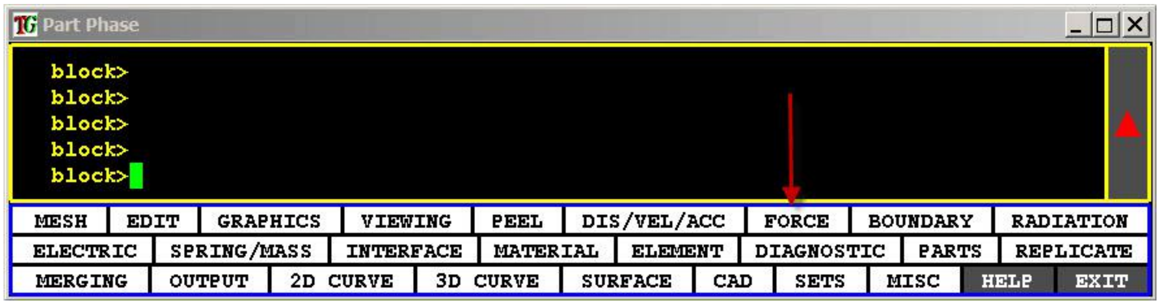


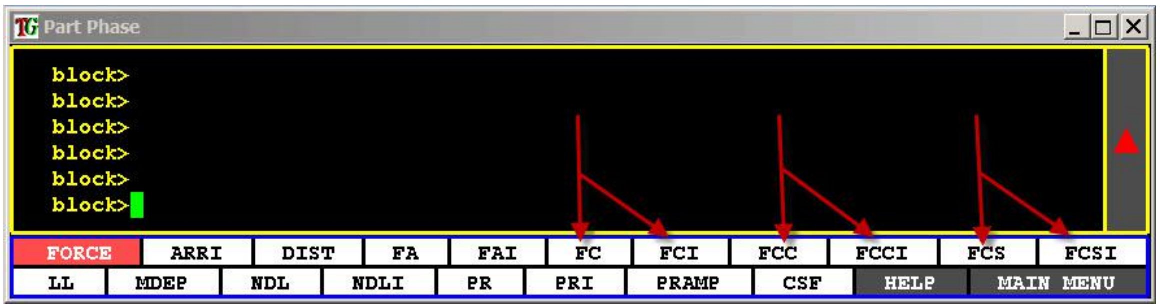


This command is straight forward. This dialogue is incomplete because the beam elements have not been selected. It was left unselected in this example so that you can see there are 2 choices: a single beam or a beam element set.

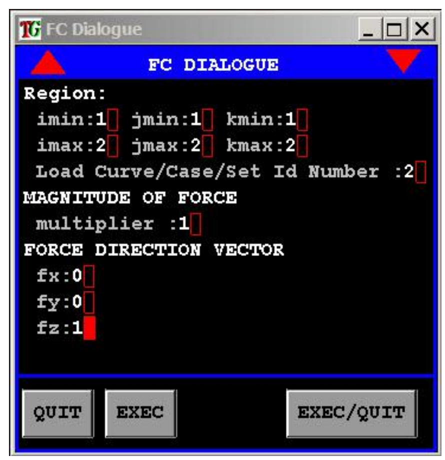
Force

Nodal force (FORCE card) can be specified in the part phase using the FC or FCI command. It is found in the FORCE menu.





There are 6 variations of this command used to define nodal forces. On the right is the FC dialogue representing all 6 commands. Be sure to select a Set ID.



The FC command is the simplest form. The FCI will require understanding index progressions which allow you to select multiple regions of the mesh in one command.

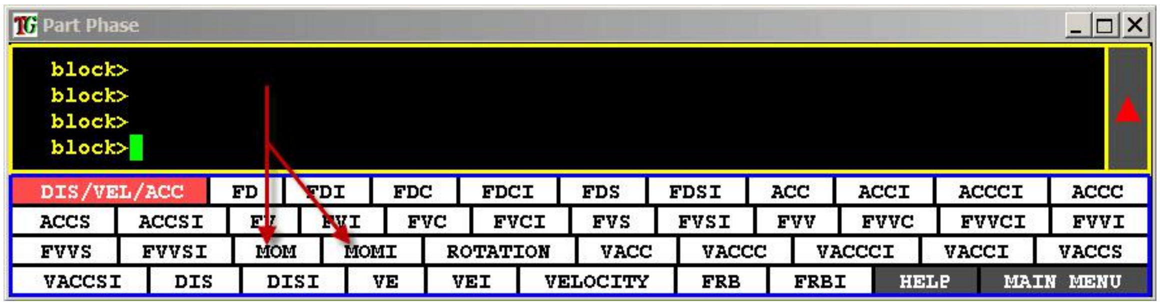
The FCC/FCCI commands define nodal forces in cylindrical coordinates. The FCS/FCSI commands define nodal forces in spherical coordinates.

The FC command can also be found in the merge phase. Different methods are available in choosing the nodes for nodal forces, including the creation of a node set.

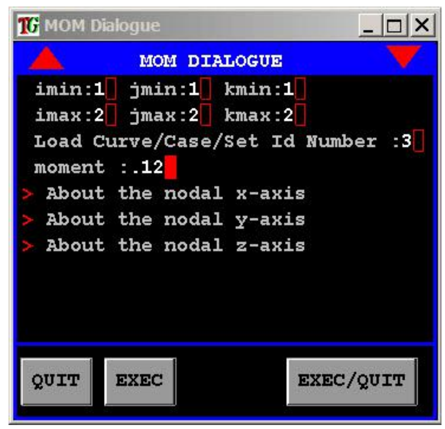
Use the FC option of the **CONDITION** command to view the nodal forces in the merge phase.

Nodal Moments

A static nodal moment can be assigned to a region in the mesh by using the **True*Grid*®** MOM/MOMI commands. These two commands are found in the DIS/VEL/ACC menu. This produces the NASTRAN® MOMENT card.



On the right is the dialogue for the MOM command in the part phase. The dialogue for the MOMI is similar. If the prescription is about a general axis, this command will be needed three times, one for a component about each axis.

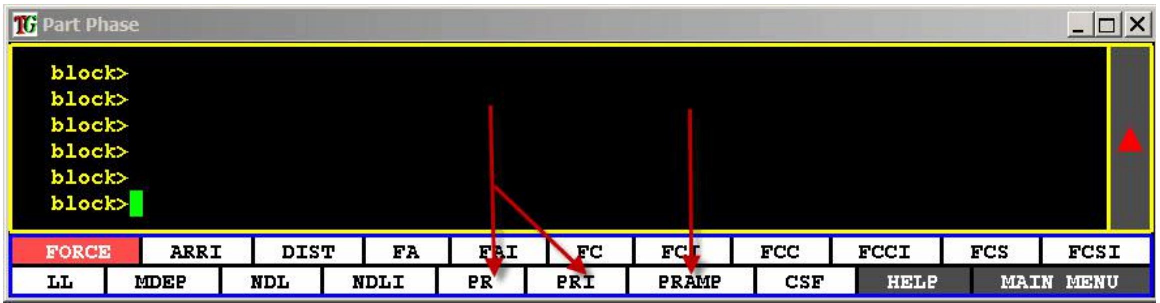


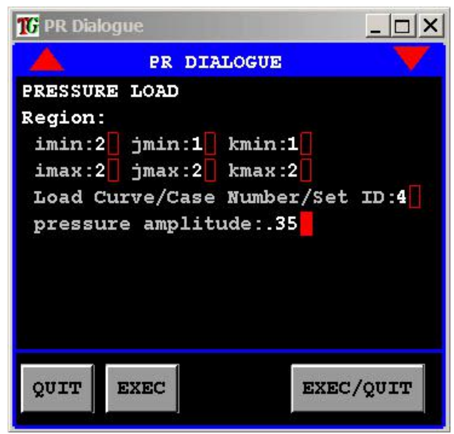
A similar dialogue is available in the merge phase for the MOM command. One of the options requires creating a node set.

Use the MOM option of the **CONDITION** command to view the static nodal moments in the merge phase.

Pressure

Faces of the mesh can be assigned a pressure load using the **True*Grid*®** PR/PRI commands in the part phase. This will produce the NASTRAN® PLOAD4 card. These commands are found under the FORCE menu. In addition, the **True*Grid*®** PRAMP command (Pressure Amplitude) can be applied by an algebraic formula which is used to calculate the pressure on any face that has a zero pressure which was assigned by the PR/PRI commands.





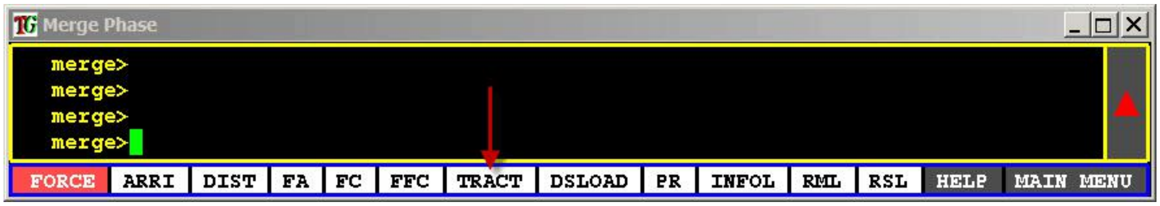
The dialogue on the right is for the PR command. The PRI is more complicated because it uses index progressions to select the faces. Otherwise, it accomplishes the same thing.

The PR command is also in the merge phase. A face set is required for the merge phase version of this command.

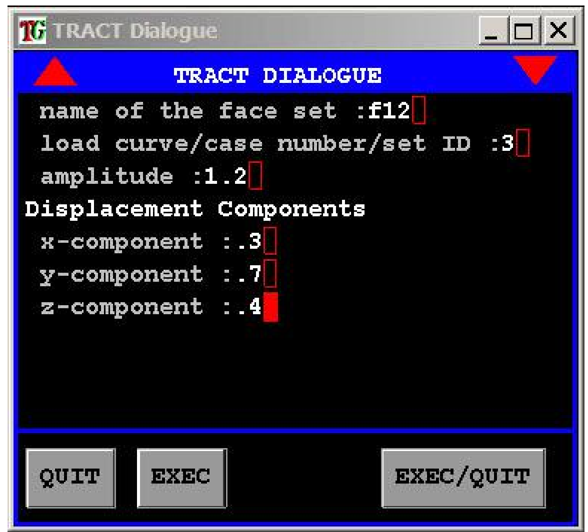
Use the PR option to the **CONDITION** command to view the facial pressures assigned to the mesh in the merge phase.

Traction

The **True*Grid*®** TRACT command applies traction to the selected faces of the mesh. This command produces the NASTRAN® PLOAD4 card. This command, at this time, is only found in the merge phase under the FORCE menu.

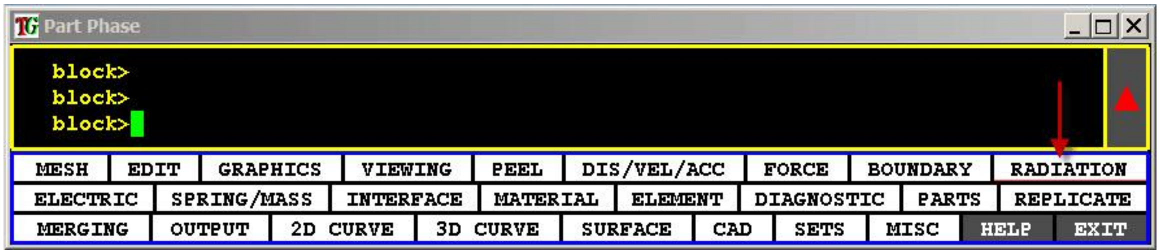


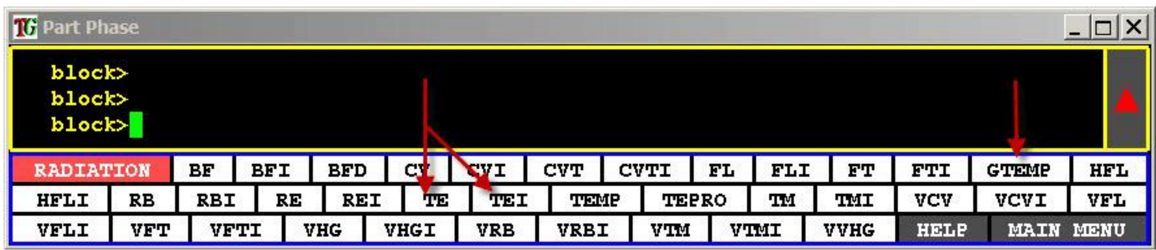
The dialogue to the right is an example of the tract command. This command requires a face set.

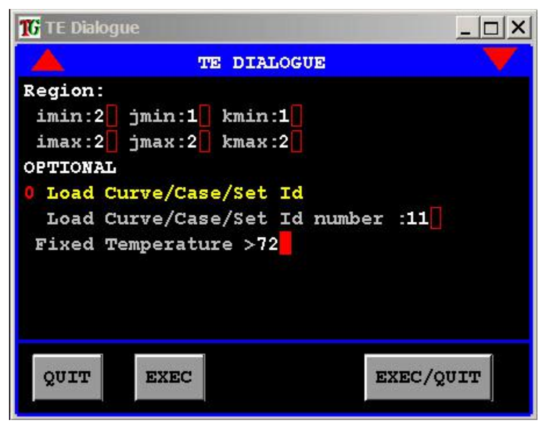


Fixed Nodal Temperatures

Nodal fixed temperatures used in the material models can be set using the **True*Grid*®** TE/TEI commands. These commands are in the part phase under the Radiation menu. They will produce the NASTRAN® TEMP card. The **True*Grid*®** GTEMP command, found in all 3 phases, sets the default global temperature, producing the NASTRAN® TEMPD card.







The dialogue to the right is an example of the TE command. Be sure to select the optional Set ID when completing this dialogue. The TEI command is the same except that it receives an index progression instead of a region and allows for multiple regions to be selected in one dialogue.

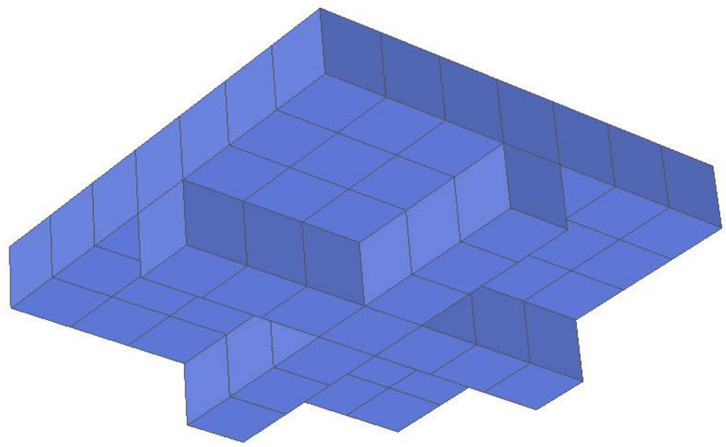
The TE command is also available in the merge phase. It differs in that it assigns temperatures to nodes in a node set.

Use the TE option of the **CONDITION** command to view the nodal fixed temperatures in the merge phase.

X. Simple Example

The following is the session file input to **True*Grid*®** in batch mode to create a simple panel model with a lattice structure using hexa elements.

```
nastran
nastmats 1 1 e 54792.  nu .3 g 21067.  rho 1.8e+03 psolid ;
title panel with lattice structure coupon stress test
block 1 4 5 8;1 4 5 8;1 2 3;1 4 5 8 1 4 5 8 1 2 3
dei 1 2 0 3 4; 1 2 0 3 4; 1 2;
bi -1 0 -4;2 3;-1;sid 1
dy 1 dz 1 ;
bi 2 3;-1 0 -4;-1;sid 1
dx 1 dz 1 ;
pr 2 2 3 3 3 3 1 20
merge
write
```



The following is the NASTRAN® bulk data file produced by **True*Grid*®**.

```
ID  TRUEGRID By XYZ Scientific Applications, Inc.
SOL     1
CEND
TITLE=panel with lattice structure coupon stress test
BEGIN BULK
MAT1    *              1          54792.          21067.             0.3
*                  1800.
PSOLID *               1               1
*                                                  SMECH
CHEXA          1       1       1       9      11       3       2      10
+             12       4
CHEXA          2       1       9      17      19      11      10      18
+             20      12
CHEXA          3       1      17      25      27      19      18      26
+             28      20
CHEXA          4       1       3      11      13       5       4      12
+             14       6
CHEXA          5       1      11      19      21      13      12      20
+             22      14
CHEXA          6       1      19      27      29      21      20      28
+             30      22
CHEXA          7       1       5      13      15       7       6      14
+             16       8
CHEXA          8       1      13      21      23      15      14      22
```

```
+             24      16
CHEXA          9       1      21      29      31      23      22      30
+             32      24
CHEXA         10       1      33      36      37      34       7      15
+             38      35
CHEXA         11       1      36      39      40      37      15      23
+             41      38
CHEXA         12       1      39      42      43      40      23      31
+             44      41
CHEXA         13       1       7      15      38      35       8      16
+             46      45
CHEXA         14       1      15      23      41      38      16      24
+             47      46
CHEXA         15       1      23      31      44      41      24      32
+             48      47
CHEXA         16       1      35      38      55      49      45      46
+             56      50
CHEXA         17       1      38      41      61      55      46      47
+             62      56
CHEXA         18       1      41      44      67      61      47      48
+             68      62
CHEXA         19       1      49      55      57      51      50      56
+             58      52
CHEXA         20       1      55      61      63      57      56      62
+             64      58
CHEXA         21       1      61      67      69      63      62      68
+             70      64
CHEXA         22       1      51      57      59      53      52      58
+             60      54
CHEXA         23       1      57      63      65      59      58      64
+             66      60
CHEXA         24       1      63      69      71      65      64      70
+             72      66
CHEXA         25       1      73      76      78      74      25      77
+             79      27
CHEXA         26       1      74      78      80      75      27      79
+             81      29
CHEXA         27       1      75      80      82      42      29      81
+             83      31
CHEXA         28       1      25      77      79      27      26      84
+             85      28
CHEXA         29       1      27      79      81      29      28      85
+             86      30
CHEXA         30       1      29      81      83      31      30      86
+             87      32
CHEXA         31       1      42      82      88      43      31      83
+             89      44
CHEXA         32       1      31      83      89      44      32      87
+             90      48
CHEXA         33       1      43      88      94      91      44      89
+             95      67
CHEXA         34       1      91      94      96      92      67      95
+             97      69
CHEXA         35       1      92      96      98      93      69      97
+             99      71
CHEXA         36       1      44      89      95      67      48      90
+            100      68
CHEXA         37       1      67      95      97      69      68     100
+            101      70
CHEXA         38       1      69      97      99      71      70     101
+            102      72
CHEXA         39       1      77     103     105      79      84     104
+            106      85
CHEXA         40       1     103     111     113     105     104     112
+            114     106
CHEXA         41       1     111     119     121     113     112     120
+            122     114
CHEXA         42       1      79     105     107      81      85     106
+            108      86
CHEXA         43       1     105     113     115     107     106     114
+            116     108
CHEXA         44       1     113     121     123     115     114     122
```

```
+             124      116
CHEXA          45        1       81      107      109       83       86      108
+             110       87
CHEXA          46        1      107      115      117      109      108      116
+             118      110
CHEXA          47        1      115      123      125      117      116      124
+             126      118
CHEXA          48        1       82      127      128       88       83      109
+             129       89
CHEXA          49        1      127      130      131      128      109      117
+             132      129
CHEXA          50        1      130      133      134      131      117      125
+             135      132
CHEXA          51        1       83      109      129       89       87      110
+             136       90
CHEXA          52        1      109      117      132      129      110      118
+             137      136
CHEXA          53        1      117      125      135      132      118      126
+             138      137
CHEXA          54        1       89      129      139       95       90      136
+             140      100
CHEXA          55        1      129      132      145      139      136      137
+             146      140
CHEXA          56        1      132      135      151      145      137      138
+             152      146
CHEXA          57        1       95      139      141       97      100      140
+             142      101
CHEXA          58        1      139      145      147      141      140      146
+             148      142
CHEXA          59        1      145      151      153      147      146      152
+             154      148
CHEXA          60        1       97      141      143       99      101      142
+             144      102
CHEXA          61        1      141      147      149      143      142      148
+             150      144
CHEXA          62        1      147      153      155      149      148      154
+             156      150
SPC1            2       32       33       34      133      134
SPC1            3       31       73       76       93       98
SPCADD          1        2        3
GRID*                    1                                 1.               1.
*                       2.
GRID*                    2                                 1.               1.
*                       3.
GRID*                    3                                 1.               2.
*                       2.
GRID*                    4                                 1.               2.
*                       3.
GRID*                    5                                 1.               3.
*                       2.
GRID*                    6                                 1.               3.
*                       3.
GRID*                    7                                 1.               4.
*                       2.
GRID*                    8                                 1.               4.
*                       3.
GRID*                    9                                 2.               1.
*                       2.
GRID*                   10                                 2.               1.
*                       3.
GRID*                   11                                 2.               2.
*                       2.
GRID*                   12                                 2.               2.
*                       3.
GRID*                   13                                 2.               3.
*                       2.
GRID*                   14                                 2.               3.
*                       3.
GRID*                   15                                 2.               4.
*                       2.
```

```
GRID*                16                            2.              4.
*                    3.
GRID*                17                            3.              1.
*                    2.
GRID*                18                            3.              1.
*                    3.
GRID*                19                            3.              2.
*                    2.
GRID*                20                            3.              2.
*                    3.
GRID*                21                            3.              3.
*                    2.
GRID*                22                            3.              3.
*                    3.
GRID*                23                            3.              4.
*                    2.
GRID*                24                            3.              4.
*                    3.
GRID*                25                            4.              1.
*                    2.
GRID*                26                            4.              1.
*                    3.
GRID*                27                            4.              2.
*                    2.
GRID*                28                            4.              2.
*                    3.
GRID*                29                            4.              3.
*                    2.
GRID*                30                            4.              3.
*                    3.
GRID*                31                            4.              4.
*                    2.
GRID*                32                            4.              4.
*                    3.
GRID*                33                            1.              4.
*                    1.
GRID*                34                            1.              5.
*                    1.
GRID*                35                            1.              5.
*                    2.
GRID*                36                            2.              4.
*                    1.
GRID*                37                            2.              5.
*                    1.
GRID*                38                            2.              5.
*                    2.
GRID*                39                            3.              4.
*                    1.
GRID*                40                            3.              5.
*                    1.
GRID*                41                            3.              5.
*                    2.
GRID*                42                            4.              4.
*                    1.
GRID*                43                            4.              5.
*                    1.
GRID*                44                            4.              5.
*                    2.
GRID*                45                            1.              5.
*                    3.
GRID*                46                            2.              5.
*                    3.
GRID*                47                            3.              5.
*                    3.
GRID*                48                            4.              5.
*                    3.
GRID*                49                            1.              6.
*                    2.
GRID*                50                            1.              6.
*                    3.
GRID*                51                            1.              7.
*                    2.
```

```
GRID*                   52                              1.              7.
*                       3.
GRID*                   53                              1.              8.
*                       2.
GRID*                   54                              1.              8.
*                       3.
GRID*                   55                              2.              6.
*                       2.
GRID*                   56                              2.              6.
*                       3.
GRID*                   57                              2.              7.
*                       2.
GRID*                   58                              2.              7.
*                       3.
GRID*                   59                              2.              8.
*                       2.
GRID*                   60                              2.              8.
*                       3.
GRID*                   61                              3.              6.
*                       2.
GRID*                   62                              3.              6.
*                       3.
GRID*                   63                              3.              7.
*                       2.
GRID*                   64                              3.              7.
*                       3.
GRID*                   65                              3.              8.
*                       2.
GRID*                   66                              3.              8.
*                       3.
GRID*                   67                              4.              6.
*                       2.
GRID*                   68                              4.              6.
*                       3.
GRID*                   69                              4.              7.
*                       2.
GRID*                   70                              4.              7.
*                       3.
GRID*                   71                              4.              8.
*                       2.
GRID*                   72                              4.              8.
*                       3.
GRID*                   73                              4.              1.
*                       1.
GRID*                   74                              4.              2.
*                       1.
GRID*                   75                              4.              3.
*                       1.
GRID*                   76                              5.              1.
*                       1.
GRID*                   77                              5.              1.
*                       2.
GRID*                   78                              5.              2.
*                       1.
GRID*                   79                              5.              2.
*                       2.
GRID*                   80                              5.              3.
*                       1.
GRID*                   81                              5.              3.
*                       2.
GRID*                   82                              5.              4.
*                       1.
GRID*                   83                              5.              4.
*                       2.
GRID*                   84                              5.              1.
*                       3.
GRID*                   85                              5.              2.
*                       3.
GRID*                   86                              5.              3.
*                       3.
```

```
GRID*                 87                              5.              4.
*                     3.
GRID*                 88                              5.              5.
*                     1.
GRID*                 89                              5.              5.
*                     2.
GRID*                 90                              5.              5.
*                     3.
GRID*                 91                              4.              6.
*                     1.
GRID*                 92                              4.              7.
*                     1.
GRID*                 93                              4.              8.
*                     1.
GRID*                 94                              5.              6.
*                     1.
GRID*                 95                              5.              6.
*                     2.
GRID*                 96                              5.              7.
*                     1.
GRID*                 97                              5.              7.
*                     2.
GRID*                 98                              5.              8.
*                     1.
GRID*                 99                              5.              8.
*                     2.
GRID*                100                              5.              6.
*                     3.
GRID*                101                              5.              7.
*                     3.
GRID*                102                              5.              8.
*                     3.
GRID*                103                              6.              1.
*                     2.
GRID*                104                              6.              1.
*                     3.
GRID*                105                              6.              2.
*                     2.
GRID*                106                              6.              2.
*                     3.
GRID*                107                              6.              3.
*                     2.
GRID*                108                              6.              3.
*                     3.
GRID*                109                              6.              4.
*                     2.
GRID*                110                              6.              4.
*                     3.
GRID*                111                              7.              1.
*                     2.
GRID*                112                              7.              1.
*                     3.
GRID*                113                              7.              2.
*                     2.
GRID*                114                              7.              2.
*                     3.
GRID*                115                              7.              3.
*                     2.
GRID*                116                              7.              3.
*                     3.
GRID*                117                              7.              4.
*                     2.
GRID*                118                              7.              4.
*                     3.
GRID*                119                              8.              1.
*                     2.
GRID*                120                              8.              1.
*                     3.
GRID*                121                              8.              2.
*                     2.
GRID*                122                              8.              2.
*                     3.
```

```
GRID*                  123                            8.              3.
*                       2.
GRID*                  124                            8.              3.
*                       3.
GRID*                  125                            8.              4.
*                       2.
GRID*                  126                            8.              4.
*                       3.
GRID*                  127                            6.              4.
*                       1.
GRID*                  128                            6.              5.
*                       1.
GRID*                  129                            6.              5.
*                       2.
GRID*                  130                            7.              4.
*                       1.
GRID*                  131                            7.              5.
*                       1.
GRID*                  132                            7.              5.
*                       2.
GRID*                  133                            8.              4.
*                       1.
GRID*                  134                            8.              5.
*                       1.
GRID*                  135                            8.              5.
*                       2.
GRID*                  136                            6.              5.
*                       3.
GRID*                  137                            7.              5.
*                       3.
GRID*                  138                            8.              5.
*                       3.
GRID*                  139                            6.              6.
*                       2.
GRID*                  140                            6.              6.
*                       3.
GRID*                  141                            6.              7.
*                       2.
GRID*                  142                            6.              7.
*                       3.
GRID*                  143                            6.              8.
*                       2.
GRID*                  144                            6.              8.
*                       3.
GRID*                  145                            7.              6.
*                       2.
GRID*                  146                            7.              6.
*                       3.
GRID*                  147                            7.              7.
*                       2.
GRID*                  148                            7.              7.
*                       3.
GRID*                  149                            7.              8.
*                       2.
GRID*                  150                            7.              8.
*                       3.
GRID*                  151                            8.              6.
*                       2.
GRID*                  152                            8.              6.
*                       3.
GRID*                  153                            8.              7.
*                       2.
GRID*                  154                            8.              7.
*                       3.
GRID*                  155                            8.              8.
*                       2.
GRID*                  156                            8.              8.
*                       3.
PLOAD4         1      32    20.0                              32      90
ENDDATA
```

XI. INDEX