

Nastran-GiD interface

Reference manual

Nastran reference manual

1 GiD-NASTRAN Interface Features	4
2 Constraints	6
2.1 Constraints (Points, Lines, Surfaces)	7
2.2 Enforced Displacements (Points, Lines, Surfaces)	8
3 Static Loads	9
3.1 Point Force Load	10
3.2 Moment	11
3.3 Line-Pressure-Load	12
3.4 Line-Projected-Pressure	13
3.5 Line-Triangular-Pressure	14
3.6 Surface-Pressure-Load	15
3.7 Normal-Surface-Load	16
4 Local Axes	17
5 Connections	18
6 Dynamic loads	19
6.1 Freq Dynamic Type 1	20
6.2 Freq Dynamic Type 2	21
6.3 Time Dynamic Type 1	22
6.4 Time Dynamic Type 2	23
6.5 Cosine/Sine Load	24
6.6 Pressure Freq Type 1 (lines and surfaces)	25
6.7 Pressure Freq Type 2 (lines and surfaces)	26
6.8 Pressure Time Type 1 (lines and surfaces)	27
6.9 Pressure Time Type 2 (lines and surfaces)	28
6.10 Initial Conditions (Points, lines and surfaces)	29
7 Thermal Loads	30
7.1 Initial Temperature	31
7.2 Heat Flux	32
7.3 Convection Boundary	33
7.4 Radiation Boundary	34
7.5 Volumetric Heat	35
7.6 Heat Boundaries	36
8 Advanced conditions	37
8.1 Concentrated Mass Element	38
8.2 Output sets	39
9 Material	40
10 Property	41
11 Tables	46
12 Executive Control	48
13 Case Control	49
13.1 Input data	50
13.2 Output Data	51
13.3 Advanced Data	52
14 Dynamics	53
14.1 Modes Analysis	54
14.2 Dynamic Design	56
15 Buckling	57
16 Nonlinear Analysis	58
17 Parameters	59
18 Verify Properties	60
19 Obtain input file for NASTRAN	61
20 Post processing	62
21 Importing NASTRAN Models	63

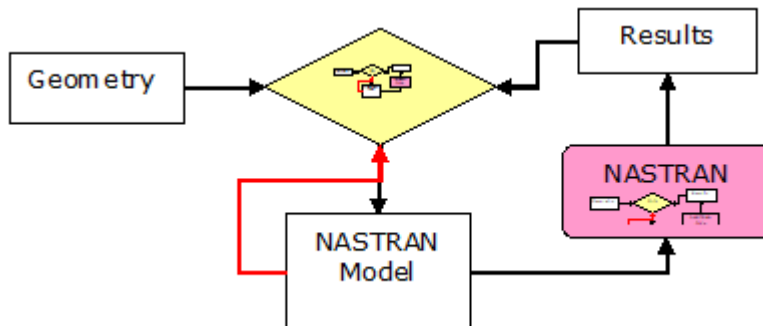
Nastran reference manual



GiD-NASTRAN Interface Features

The GiD-NASTRAN Interface is designed to be a bridge between NASTRAN codes and GiD pre/postprocessor. NASTRAN uses the power tools of GiD mesh and the easy way to assign conditions directly over geometry. With these special capabilities, *NASTRAN* generates input files easy and fast. Working with beams is very intuitive and automatized, and the conditions can be applied directly to the line before meshing and creating linear elements.

For NASTRAN old users, this way of defining loads and constraints may be new, but these improvements will make the work much easier.



About this Manual

In this manual, different kinds of fonts are used to help the users follow all the possibilities offered by GiD-NASTRAN Interface:

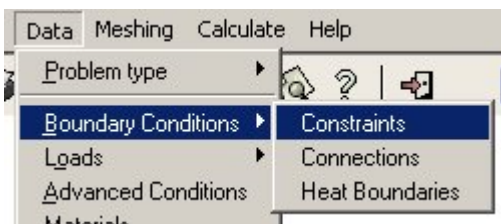
- **font** is used for the options found in the menus and windows.

In the following, a menu sequence will be indicated by a graphical arrow

i.e.

Data Boundary conditions Constraints

corresponds to the menu sequence shown in *Figure 2*.



Introduction

The analysis of a problem by means of GiD-NASTRAN Interface consists of the following basic steps:

- Pre-processing
 1. Creation (or importation) of the geometry to be analyzed
 2. Assignment of the material properties, boundary conditions and definition of general data
- Mesh generation
- Calculation
- Post-processing

Pre-processing

The first step for any analysis is the setting up of the problem. This includes the creation or importation from a CAD file of the problem's geometry assignment of boundary and initial conditions, and the generation of the mesh. This will all be made in the pre and post processor GiD.

Pre-processing part of the analysis consists of the following steps:

- Conditions assignment
- Materials definition
- Boundaries definition
- General problem data insertion
- Solver data definition
- Units definition

- Mesh size assignment
- Mesh Generation

The GiD-NASTRAN Interface icon bar has been designed to guide the user, during the preprocessing part of the analysis.

To load GiD-NASTRAN Interface in GiD to follow the menu sequence bellow:

Data Problemtyp

A splash image will appear the first time that GiD-NASTRAN interface is loaded.

Item help

To obtain specific item help, click the question mark icon in the toolkit and then click on the item you need help on.



Load Cases

One load case is a group of one or more loads assigned to entities. When a new model is defined in GiD-NASTRAN Interface a default load case, called Load case 1, is defined. It is possible to rename this loadcase or define new ones by using the loadcase button in the toolbar or in the load cases window.

When a load is assigned to one or several entities, this load is inserted inside the loadcase that is currently active. To make a loadcase active, select it in the loadcase menu in the toolbar or in the loadcases window. To show which load case is currently active, the loadcase button changes to reflect the load case number.

If combined load cases are not defined, GiD-NASTRAN Interface calculates just one analysis that is equivalent to that of all the loads belonging to one load case.

Combined load cases

Data Load cases

Combined load cases are defined in the load cases window

Options are:

- **Prestress model:** Some NASTRAN codes offer the possibility to simulate models affected by a pre-stiffened or pre-stressed load case. The basic idea is to run a first combined loadcase that will contain all the simple loadcases used to calculate pre-stress in the model and another combined loadcase with the post-stress load.
- **One result for every loadcase:** One combined loadcase is created for every simple loadcase. It is possible to enter amplification factors for every loadcase. This option is equal to run so many models as loadcases defined.
 - **Use combined loadcases:** User can define as many loadcases as needed. It is possible to enter amplification factors for every combination of simple loadcases in combined loadcases.

When using combined loadcases, one default combined loadcase is already existent. To create more, press right mouse button over the existent combined load case name. A menu appears that offers several options.

One factor must be entered for every simple loadcase. This factor will multiply the load in order to create the combined loadcase. In the postprocessing part, after the analysis, there will be one different result for every combined loadcase. Enter value 0.0 in order to deactivate that loadcase.

It is also possible to choose the same constraints for combined loadcases. Sometimes this option can be very useful because when there are only changes in loads distribution and the constraints are always the same.

Constraints

This section describes all options that the user can find in the menu Constraints

Constraints (Points, Lines, Surfaces)

MENU SEQUENCE:

Data Boundary Conditions Constraints

Allows assigning prescribed displacements and rotations around global axes.

These conditions can be assigned over points, lines and surfaces, and affect nodes.

X-Constraint->Prescribed displacement along X-global axis

X-Rotation->Prescribed rotation around X-global axis

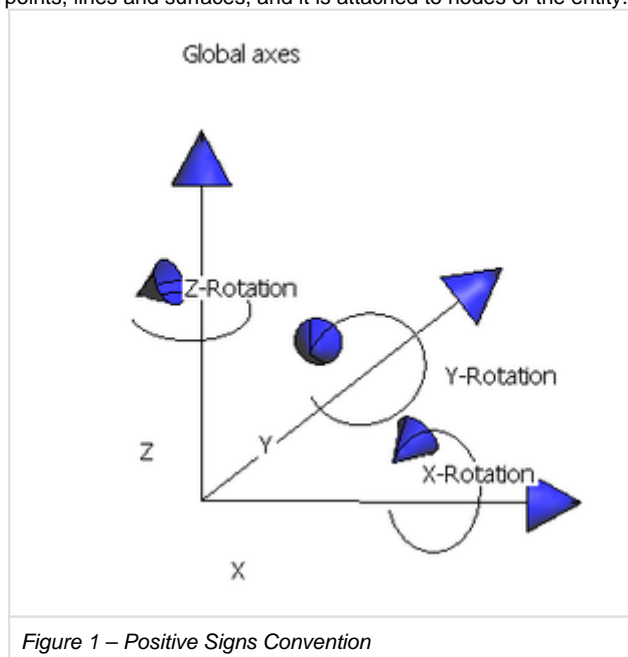
The prescribed displacements and rotations always make references to global axes.

Enforced Displacements (Points, Lines, Surfaces)

MENU SEQUENCE:

Data Boundary Conditions Constraints

Assigns enforced displacements or rotations. The signs convention for components of a vector follows the right hand rule (see figure below). This condition can be assigned over points, lines and surfaces, and it is attached to nodes of the entity.



Enforced displacements and rotations always make references to global axes.

Static Loads

This section describes all options that the user can find in the menu Static Loads.

Point Force Load

MENU SEQUENCE:

Data Loads Static loads

This load is applied to one point of a structure. The sign of components is equal to the one defined for the enforced displacements. Figure 1

Moment

MENU SEQUENCE:

Data Loads Static loads

This moment is applied to one point of a structure. A sign of components is equal to the one defined for the enforced displacements (right hand rule). Figure 1

Line-Pressure-Load

MENU SEQUENCE:

Data Loads Static loads

Defines uniformly distributed applied loads to line entities of a geometry (bars, beams and curved beams). It is possible to select the coordinate system for data entries in this load.

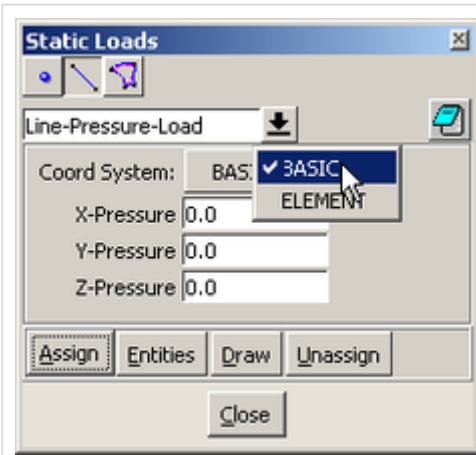


Figure 2 – Coordinate System

If the option "BASIC" is selected, all data entries make reference to global (basic) axes. If the option "ELEMENT" is selected, all data entries make reference to local (element) axes. Axes are defined by the user in Local Axes condition (for more information about Local Axes, see Local Axes Condition). Remember that it is necessary to define local axes for every line of structure.

The following figures are examples of differences between "BASIC" and "ELEMENT" coordinate system.

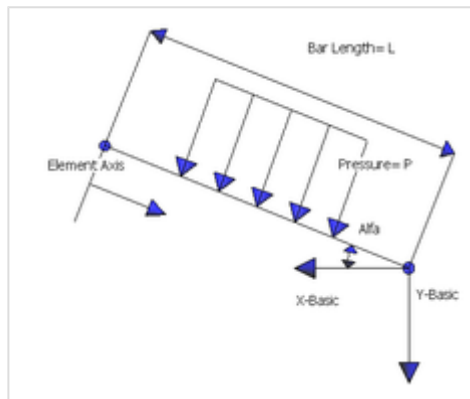


Figure 3 – Pressure Load Y-Element

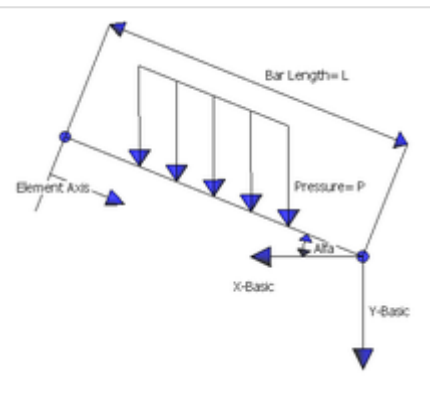


Figure 4 – Pressure Load Y-Basic

In the Figure 3 (Pressure Load Y-Element), the user selects Element coordinate system. The pressure load is in a direction of Y-Element axis. In the Figure 4 (Pressure Load y-Basic) the user selects Basic coordinate system.

In these two options the total load applies to the beam equals $P \cdot L$ in the Y-Basic direction.

Comment for NASTRAN advanced users: It is not necessary to mesh to assign this condition. It can be assigned directly over lines of a structure.

Line-Projected-Pressure

MENU SEQUENCE:

Data Loads Static loads

This condition is very similar to the previous one. The coordinate system works the same way, based on the Line-Pressure-Load.

The only difference is that the total load applied to the beam equals $P*L*\cos(\alpha)$ in the Y-Basic direction, where alpha is the angle of the X-Element axis with respect to the X-Basic axis. (See the two previous figures).

Comment for NASTRAN advanced users: It is not necessary to mesh to assign this condition. It can be assigned directly over lines of a structure.

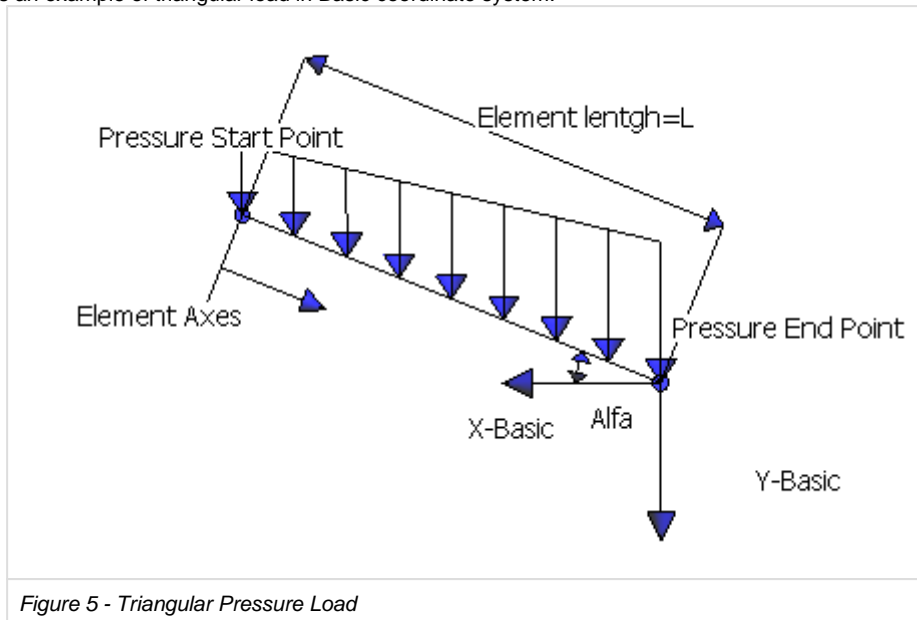
Line-Triangular-Pressure

MENU SEQUENCE:

Data Loads Static loads

Defines a triangular distributed load to linear elements of a structure. The coordinate system works the same way: Line-Pressure-Load, but in this condition, element axes make reference to element axes and not to the local axes of the line (normally, it is common that element axes are equal to line axes). It is very important to use this condition correctly. Remember that it is only applicable to mesh linear elements. In this condition it is necessary to introduce a pressure to the start point and a pressure to the end point. To know which ends of the element is the start point, the user has to pay attention to the origin of the X-Element axis.

The following figure is an example of triangular load in Basic coordinate system.



Suggestion: If the user wants to apply this kind of load directly over a line, one option may be mesh with only one liner element all line.

Surface-Pressure-Load

MENU SEQUENCE:

Data Loads Static loads

Defines a pressure load on a surface.

Surface Pressure Load always makes references to global axes.

Normal-Surface-Load

MENU SEQUENCE:

Data Loads Static loads

Defines pressure load acting normal to a surface.

A negative sign of the pressure is inward to the surface.

Local Axes

MENU SEQUENCE:

Data Properties Local Axes

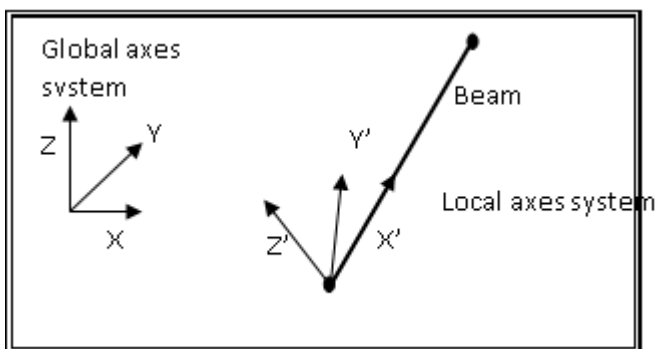


Figure 6 – Local Axes

The model has been created related to a global axes system XYZ that is unique for the entire problem. But every beam must have its own local axes system X'Y'Z' in order to:

1. Refer section properties like *Inertia modulus* or *thickness* and *height* to this system.
2. Some of the loads (that have the prefix *Local*) are related also to this system.
3. Strength results over the beam are referred to this local axes system.

The main property of this system is that the local X' axis must have the same direction than the beam.



The ways for defining local axes systems are:

1- Default. The program assigns a different local axes system to every beam with the following criteria:

- X' axe has the direction of the beam.
- If X' axe has the same direction than global Z axe, Y' axe has the same direction than global X. If not, Y' axe is calculated so as to be horizontal (orthogonal to X' and Z).
- Z' axe is the cross product of X' axe and Y' axe. It will try to point to the same sense than global Z (dot product of Z and Z' axes will be positive or zero).

Note: The intuitive idea is that vertical beams have the Y' axe in the direction of global X. All the other beams have the Y' axe horizontal and with the Z' axe pointing up.

2- Automatic. Similar to the previous one but the local axes system is assigned automatically to the beam by GiD. The final orientation can be checked with the *Draw Local Axes* option in the GiD Conditions window.

3- Automatic alt. Similar to the previous one but an alternative proposal of local axes is given. Typically, User should assign Automatic local axes and check them, after assigning, with the *Draw local axes* option. If a different local axes system is desired, normally rotated 90 degrees from the first one, then it is only necessary to assign again the same condition to the entities with the **Automatic alt** option selected.

4- User defined. User can created different named local axes systems with the GiD command:

Data->Local axes->Define

and with the different methods that can be chosen there. The names of the defined local axes will be added to the menu where Local axes are chosen.

Connections

MENU SEQUENCE

Data Boundary Conditions Connections

With this condition it is possible to disconnect some degrees of freedom of points in a structure. Degrees of freedom make reference to local axes of a line.

If the user wants to disconnect some degrees of freedom between two beams, this condition has to be assigned to the union point.

Note, that the conditions are only applicable to beams and not to shells.

Dynamic loads

This section describes all options of the Dynamic Loads menu.

There are two ways to enter the conditions for dynamic analysis:

- First, when the conditions are dependent on the frequency. In this case the user has to use frequency dependent dynamic loads. Only loads with its' names including 'Freq' can be used in this case.
 - Second, when the conditions are time dependent. In this case the user has to use time dependent dynamic loads. Only loads with its' names including 'Time' can be used in this case.
- It is important not to confuse these two forms of loads introduction with an analysis formulation (modal or direct).

Freq Dynamic Type 1

MENU SEQUENCE:

Data Loads Dynamics Loads

Defines a frequency dependent dynamic load of the form:

$$p(f) = A(C(f) + iD(f))e^{i(\theta - 2\pi f_t)}$$

This type is used in transient response problems, where:

f= independent variable, it is a frequency.

i= imaginary unit.

A= Scale factor

 F_t = Frequency delay θ = Phase lead

C(f) and D(f) are functions defined by tables of interpolation. Tables have to be created before assigning this condition and associated to the condition using statements "Table Interpolation Values C[f]" or "Table Interpolation Values D[f]".

The user has to define these tables using the following menu sequence: Materials->Tables. More information about table creation can be found in the chapter Tables of this manual.

Finally, the user has to select in which degree of freedom he wants the load to act.

This condition can be assigned over points, lines and surfaces and has effect over nodes of the mesh.

Freq Dynamic Type 2

MENU SEQUENCE:

Data Loads Dynamics Loads

Defines a frequency dependent dynamic load of the form:

$$p(f) = AB(f)e^{i(\phi(f) + \theta - 2\pi f \tau)}$$

For use in frequency response problems, where:

f= independent variable, it is a frequency.

i= imaginary unit.

A= Scale factor

 F_t = Frequency delay θ = Phase leadB(f) and $\phi(f)$ are functions defined by tables of interpolation. Tables have to be created before assigning this condition and associated tothe condition using statements "Table Interpolation Values B[f]" or "Table Interpolation Values ϕ [f]".

The user has to define these tables using the following menu sequence: Materials->Tables. More information about table creation can be found in the chapter Tables of this manual

This condition can be assigned over points, lines and surfaces and has effect over nodes of the mesh.

Time Dynamic Type 1

MENU SEQUENCE:

Data Loads Dynamics Loads

Defines a time dependent dynamic load of the form:

$$p(t) = AF(t - \tau)$$

This condition is used in transient response problems, where:

t= independent variable, it is a time.

A= Scale factor

 τ = Time delay

$F(t - \tau)$ is a function defined by table of interpolation. Table has to be created before assigning this condition and associated to the condition using statements "Table Interpolation Values $F(t - \tau)$ ".

The user has to define these tables using the following menu sequence: Materials->Tables. More information about table creation can be found in the chapter Tables of this manual.

This condition can be assigned over points, lines and surfaces and affects nodes of the mesh.

Time Dynamic Type 2

MENU SEQUENCE:

Data Loads Dynamics Loads

Defines a time dependent dynamic load of the form:

$$p(t) = \begin{cases} 0 & , \tilde{t} < 0 \text{ or } \tilde{t} > T_2 - T_1 \\ A \tilde{t}^B e^{C \tilde{t}} \cos(2\pi f \tilde{t} + P) & , 0 \leq \tilde{t} \leq T_2 - T_1 \end{cases}$$

where $\tilde{t} = t - T_1 - \tau$

For use in transient response problems, where:

t= independent variable, it is a time.

A= Scale factor

 τ = time delay T_1 =Inferior time limit T_2 =Superior time limit

F= frequency in cycles per unit time

P= Phase angle (in degrees)

C= Exponential coefficient

B= Growth coefficient.

It is not necessary to define any interpolation table for this load.

This condition can be assigned over points, lines and surfaces, and affects nodes of the mesh.

Cosine/Sine Load

MENU SEQUENCE:

Data Loads Dynamics Loads

Defines a time dependent dynamic load of the form:

$$P(t) = \begin{cases} 0 & t' < 0 \text{ or } t' > T_2 - T_1 \\ A \cos(2\pi f t' + P) & 0 \leq t' \leq T_2 - T_1 \end{cases}$$

where $t' = t - T_1 - \tau$

This condition is used in transient response problems. The definitions of the parameters of this load are equal to Time Dynamic Type 2.

Pressure Freq Type 1 (lines and surfaces)

MENU SEQUENCE:

Data Loads Dynamics Loads

Defines a frequency dependent dynamic pressure for frequency response problems use.

This load is very similar to "Freq Dynamic Type 1" with some changes in parameters:

A=scale factor was changed to a vector that the user has to introduce using statements: X-Pressure, Y-Pressure, and Z-Pressure.

F_t= Frequency delay is equal to zero.

θ = Phase lead is equal to zero.

C(f) and D(f) are functions defined by tables of interpolation. Tables have to be created before assigning this condition and associated to the condition using statements "Table Interpolation Values C[f]" or "Table Interpolation Values D[f]".

In conclusion, the final aspect of the load is as follows:

$$p(f) = \vec{A}(C(f) + iD(f))$$

where $\vec{A} = (X - \text{Pressure}, Y - \text{Pressure}, Z - \text{Pressure})$

If lines are used, it is possible to select in which coordinate system (BASIC, ELEMENT) and which type of load (NORMAL, PROJECTED) will be applied to line of geometry. Descriptions of these statements are explained with more details in static loads that can be applied to lines. See these chapters for more information.

This load can be assigned over lines and affects elements of the mesh.

Remarks

This load is only available for MSC/NASTRAN and NE/NASTRAN codes.

Pressure Freq Type 2 (lines and surfaces)

MENU SEQUENCE:

Data Loads Dynamics Loads

Defines a frequency dependent dynamic pressure for use in frequency response problems.

This load is very similar to "Freq Dynamic Type2" with some changes in parameters:

A=scale factor was changed to a vector that the user has to introduce using statements: X-Pressure, Y-Pressure, and Z-Pressure.

F_t= Frequency delay is equal to zero.

θ = Phase lead is equal to zero.

B(f) and \varnothing (f) are functions defined by tables of interpolation. Tables have to be created before assigning this condition and associated to the condition using statements "Table Interpolation Values B[f]" or "Table Interpolation Values h[f]".

In conclusion, the final aspect of the load is like this:

$$p(f) = \vec{A} \left(B(f) e^{i(\varnothing(f))} \right)$$

where $\vec{A} = (X - \text{Pressure}, Y - \text{Pressure}, Z - \text{Pressure})$

If lines are used, it is possible to select in which coordinate system (BASIC, ELEMENT) and which type of load (NORMAL, PROJECTED) will be applied to line of geometry. Descriptions of these statements are explained with more details in static loads that can be applied to lines. See these chapters for more information.

This load can be assigned over lines and affects elements of the mesh.

Remarks

This load is only available for MSC/NASTRAN and NE/NASTRAN codes.

Pressure Time Type 1 (lines and surfaces)

MENU SEQUENCE:

Data Loads Dynamics Loads

Defines a time dependent dynamic pressure for use in transient response problems.

This load is very similar to "Time Dynamic Type 1" with some changes in parameters:

A=scale factor was changed to a vector that the user has to introduce using statements: X-Pressure, Y-Pressure, and Z-Pressure.

τ = Time delay is equal to zero.

$F(t-\tau)$ is a functions defined by table of interpolation. Table has to be created before assigning this condition and associated to the

condition using statements "Table Interpolation Values $F(t-\tau)$ ".

The user has to define these tables using the following menu sequence: Materials->Tables. More information about table creation can be found in chapter Tables of this manual.

In conclusion, the final aspect of the load is like this:

$$p(t) = \vec{A}(F(t))$$

where $\vec{A} = (X - \text{Pressure}, Y - \text{Pressure}, Z - \text{Pressure})$

If lines are used, it is possible to select in which coordinate system (BASIC, ELEMENT) and which type of load (NORMAL, PROJECTED) will be applied to a line of geometry. Descriptions of these statements are explained with more details in static loads that can be applied to lines. See these chapters for more information.

This load can be assigned over lines and affects elements of the mesh.

Remarks

This load is only available for MSC/NASTRAN and NE/NASTRAN codes.

Pressure Time Type 2 (lines and surfaces)

MENU SEQUENCE:

Data Loads Dynamics Loads

Defines a time dependent dynamic pressure for use in transient response problems.

This load is very similar to "Time Dynamic Type 2" with some changes in parameters:

A=scale factor was changed to a vector that has to be introduced using statements: X-Pressure, Y-Pressure, and Z-Pressure.

τ = Time delay

T_1 =Inferior time limit

T_2 =Superior time limit

F= frequency in cycles per unit time

P= Phase angle (in degrees)

C= Exponential coefficient

B= Growth coefficient.

In this load is not necessary to define any interpolation table.

In conclusion, the final aspect of the load is as follows:

$$p(t) = \begin{cases} 0 & , \tilde{t} < 0 \text{ or } \tilde{t} > T_2 - T_1 \\ \bar{A} \tilde{t}^B e^{C\tilde{t}} \cos(2\pi f\tilde{t} + P) & , 0 \leq \tilde{t} \leq T_2 - T_1 \end{cases}$$

where $\tilde{t} = t - T_1 - \tau$ and $\bar{A} = (X\text{-Pr essure}, Y\text{-Pr essure}, Z\text{-Pr essure})$

If lines are used, it is possible to select in which coordinate system (BASIC, ELEMENT) and which type of load (NORMAL, PROJECTED) will be applied to line of geometry. Descriptions of these statements are explained with more details in static loads that can be applied to lines. See these chapters for more information.

This load can be assigned over lines and affects elements of the mesh.

Remarks

This load is only available for MSC/NASTRAN and NE/NASTRAN codes.

Initial Conditions (Points, lines and surfaces)

MENU SEQUENCE:

Data Loads Dynamics Loads

Defines values for the initial conditions of coordinates used in transient analysis. Both displacement and velocity values may be specified at independent (global) coordinates of the structural model.

Initial conditions may be used only in direct formulation.

This condition can be assigned over points, lines and surfaces and affects nodes.

Thermal Loads

This section describes the available thermal loads in heat transfer analysis.

Initial Temperature

MENU SEQUENCE:

Data Loads Thermal Loads

This condition assigns the initial temperature of the grid nodes. This condition can be assigned to points, lines and surfaces. It will have effect over grid nodes of the mesh included in the geometry entity with the condition assigned.

Heat Flux

MENU SEQUENCE:

Data Loads Thermal Loads

This condition assigns a prescribed heat flow. This condition can be assigned to points, lines and surfaces. To apply this condition to lines and surfaces has to be combined with "heat boundary" condition to define the geometrical information of the condition. Heat flux is analogous to distributed pressure in Static Analysis.

Convection Boundary

MENU SEQUENCE:

Data Loads Thermal Loads

This condition defines a convection boundary. It can be assigned to points lines and surfaces. To apply this condition is mandatory to assign a heat boundary condition to define geometric properties of the boundary. By pressing de “Temp...” is possible to define a temperature-dependent convection coefficient. This option can be useful for nonlinear heat transfer analysis.

Radiation Boundary

MENU SEQUENCE:

Data Loads Thermal Loads

This condition defines a radiation boundary. It can be assigned to points lines and surfaces. To apply this condition is mandatory to assign a heat boundary condition to define geometric properties of the boundary. By pressing de “Temp...” is possible to define a temperature-dependent radiation view factor. This option can be useful for nonlinear heat transfer analysis.

Volumetric Heat

MENU SEQUENCE:

Data Loads Thermal Loads

This condition defines a volumetric heat distribution. The volumetric heat condition plays the same role of gravity in Static Analysis.

It can be assigned to lines, surfaces and volumes. To apply this condition is mandatory to assign a heat boundary condition to define geometric properties of the elements, i.e. volume of linear elements equal to length x section area.

Heat Boundaries

MENU SEQUENCE:

Data Boundary Heat Boundaries

It defines a boundary surface element for heat transfer analysis. It is mandatory to assign this condition to any geometrical entity that will be a heat boundary.

Heat Boundary condition gives geometrical information of the heat boundary like the orientation of the boundary and the area factor of application.

It is possible to define five types of heat boundary:

1. **Prescribed Temperature:** This type of heat boundary doesn't need to be defined by the use of "heat boundary" condition. It is only necessary to assign "Fixed Temperature" condition to the desired model boundary.
2. **Heat flow across boundary due to radiation:** To define this type of heat boundary is necessary to assign the "heat boundary" condition (assigning this condition we are defining a general heat boundary) and to assign "radiation boundary" condition to define a radiation boundary.
3. **Heat flow across boundary due to convection:** This type of boundary is very similar to the previous one. It is also necessary to assign the "heat boundary" condition and "convection boundary" to define a convection boundary.
4. **Prescribed rate of heat flow across boundary:** This type of boundary is useful when an imposed heat flow is boundary condition of the problem. This boundary works in the same way like the two previous ones.
5. **Insulated** (no heat flow across boundary): For this type of boundary any condition has to be assigned. By default all elements are insulated.

Advanced conditions

This section describes special conditions that can be used in dynamic and static analysis.

Concentrated Mass Element

MENU SEQUENCE:

Data Advanced Conditions

Defines a concentrated mass at a node of the structural model.

There are two different ways the condition information can be introduced:

-CONM1: Properties of the concentrated mass have to be introduced using a symmetric mass matrix. This format gives total control to the user to perform the properties of the element.

-CONM2: For a less general means of defining concentrated mass at nodes. See the following statements descriptions:

-Mass Value: Real

-Offset distances: (X1, X2, X3) Offset distances for the mass in the global coordinate system (Real).

I_{ij} : Mass moments of inertia measured at the mass center of gravity in global coordinate system.

Remarks

The form of the inertia matrix about its center of gravity (c.g.) is taken as:

$$\begin{pmatrix} M & 0 & 0 & \vdots & 0 & MX_3 & MX_2 \\ & M & 0 & \vdots & -MX_3 & 0 & MX_1 \\ & & M & \vdots & MX_2 & -MX_1 & 0 \\ \dots & \dots & \dots & \dots & \dots & \dots & \dots \\ & & & \vdots & \bar{I}_{11} & -\bar{I}_{21} & -\bar{I}_{31} \\ & \text{Symetric} & & \vdots & & \bar{I}_{22} & \bar{I}_{32} \\ & & & \vdots & & & \bar{I}_{33} \end{pmatrix}$$

where $\bar{I}_{11} = I_{11} + M(X_2^2 + X_3^2)$

and $\bar{I}_{21} = I_{21} + M(X_1X_2)$

and $\bar{I}_{22}, \bar{I}_{31}, \bar{I}_{32}$ and \bar{I}_{33} are similarly defined

Output sets

MENU SEQUENCE:

Data Advanced Conditions

Allows create sets of points, lines or surfaces. These sets are used to configure output file of NASTRAN. NASTRAN's output file only contains output requests for elements or nodes of these sets. To use this condition, the user has to select which kind of request (see table below) should be associated with the set and after click over geometry elements included in the set.

In case that no output set is defined, the output requests will be printed for all elements and nodes.

This condition is used when only part of the model is of interest and not all results in every element or node have to be printed. The necessary time for running the model can be significantly reduced using this condition.

Table of outputs

Abbreviat ion	Definition	Entities	Mesh
Displace ment	Requests the displacements for a selected set of PHYSICAL points	Points	Nodes
SpcForces	Requests the single-point forces of constraint at a set of points	Points	Nodes
Accelerat ion	Requests the accelerations for a selected set of PHYSICAL points	Points	Nodes
Velocity	Requests the velocities for a selected set of PHYSICAL points	Points	Nodes
Load	Selects a set of applied loads for output	Points	Nodes
ESE	Requests structural element strain energies	Lines and Surfaces	1-D and 2-D elements.
Forces	Requests the forces in a set of structural elements	Lines and Surfaces	1-D and 2-D elements.
Strain	Requests the strains/curvatures in a set of structural elements (applicable to TRIA1, TRIA2, QUAD1, and QUAD2 only).	Lines and Surfaces	1-D and 2-D elements.
Stress	Requests the stresses in a set of structural elements	Lines and Surfaces	1-D and 2-D elements.

Material

MENU SEQUENCE:

Data Materials

Defines new materials. It is possible to define a material with the following properties:

-Stiffness Page: In this page it is possible to define mechanical properties of a material.

- E, Young Modulus
- G, Shear Modulus
- nu, Poisson coefficient.

- Limit Stress: Allowable stresses in tension, compression, and shear. Required if composite element failure index is desired.

-Thermal Page: In this page it is possible to define thermal properties of a material.

-Others: Defines different properties not defined in previous pages.

A mouse-over help is available for all labels of all entries.

Every field has a "Temp..." button associated. By the means of this button it is possible to define a temperature dependent property. When "Temp.." button is pressed the Tables window will appear. Please see Tables section of this manual to see how this option works.

Stress-dependent material properties

Defines stress-dependent material properties for nonlinear analysis. Click the *Nonlinear...* button to open the properties window.

Nonlinear elastic materials are defined by stress-strain curve. This curve can be defined by clicking at the *Function...* button. Stress-strain curve should be defined in the first and third quadrants to accommodate different uniaxial tensions and compression properties. All other properties do not apply to this type of materials.

It is possible to define elasto-plastic materials in two different ways:

-Use the linear constants coupled with the plasticity modulus H. This is the work hardening slope, and it is related to the tangential modulus, E_T (the slope of stress vs. plastic strain) by the following:

$$H = E_T / (1 - E_T/E)$$

-Define a curve in stress-strain plane. The curve must be defined in the first quadrant. The first point must be at origin

($X1 = 0, Y1 = 0$) and the second point ($X2, Y2$) must be at the initial yield point ($Y1$ or $2c$). The slope of the line joining the origin to the yield stress must be equal to the value of E .

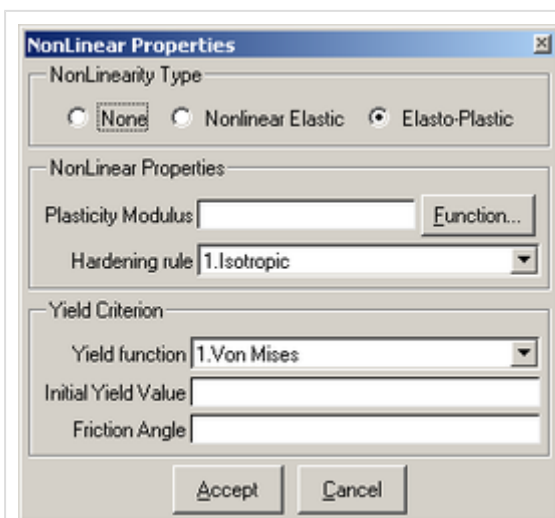


Figure 9 Nonlinear Properties Window

Property

MENU SEQUENCE:

Data Properties Property

A beam or shell entity can only have one property defined. It is possible to choose between the following types of properties:

-**Linear elements:** List of properties that can be assigned to lines.

-**Bar:** useful for bars and beams definition with the same beginning and ending section.

Definitions of the different statements:

-**Area:** Area of a section.

-**Moments of inertia:** Area moments of inertia (Real, $I_{112} > I_{212}$)

-**Torsional constant** (Real)

-**Y-Shear Area (K1), Z-Shear Area (K2):** The quantities K1 and K2 are expressed by relative amounts (0.0 to 1.0) of the total cross-sectional area. The quantities contribute to the transverse shear stiffness (KAG) in the direction of the two principal axes. These quantities are ignored if I12 is not zero. The following are the default values for K1 and K2:

$$K1 = (12 \cdot E \cdot I1) / (L \cdot L \cdot L)$$

$$K2 = (12 \cdot E \cdot I2) / (L \cdot L \cdot L)$$

-**Nonstructural mass/ length:** It is possible to add mass to structural mass (Real).

-**Stress recovery:** Stress recovery coefficients (Real).

-**Material composition:** It is possible to choose a material composition from a materials list defined in the materials section.

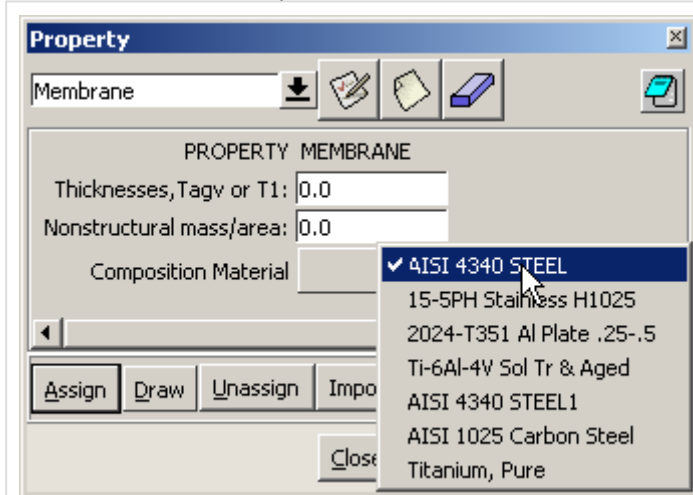



Figure 7 – Composition Material Combo Box

- Section Button**  This functionality allows to the user defines the shape of different sections of bars and beams. At the moment it is possible work with rectangular section, semi-circular section, trapezoidal section and double T section.

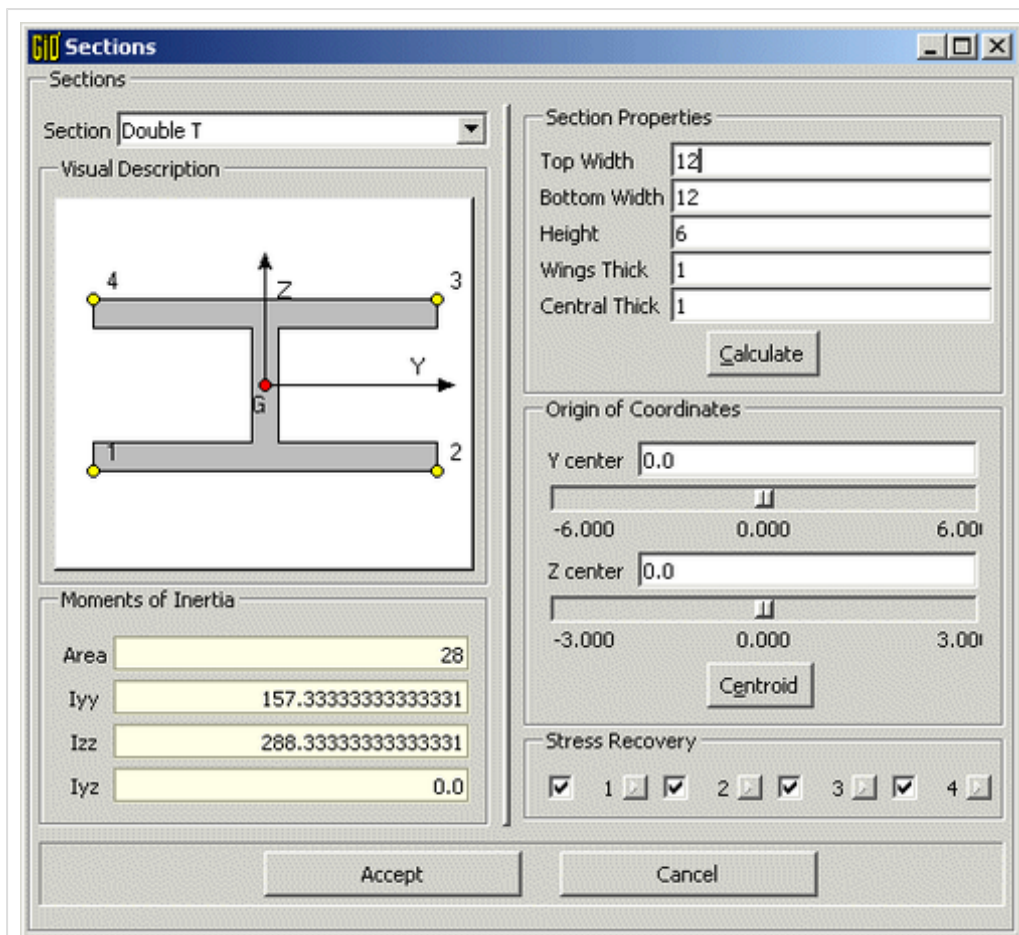


Figure 8 - Section window

-Beam: In this version it is the same like a bar.

-Curved Beam: The only difference between bars and beams is Bend Radius statement.

-Tube: Defines the properties of a thin-walled cylindrical tube element.

-Viscous Damper: Defines a one-dimensional viscous damper element of the structural model. Used only for direct formulation of dynamic analyses. Parameters of this property are viscous coefficients for extension and rotation (Real).

-Spring: spring element is a spring, which connects either translational (Axial) or rotational (Torsional) degrees of freedom. You cannot specify both stiffness and damping values for the same elements.

-DOF Spring: Unlike the Spring Element, which acts along the line between the elemental endpoints, the DOF Spring connects two nodal degrees of freedom - independent from their orientation relative to each other. Degrees of freedom via the combo boxes should be chosen at the end of the dialog box. Similar to the spring however, you can specify both stiffness and damping.

Note for NASTRAN advanced users:

The following table describes relations between viscous damper, springs, DOF springs and NASTRAN elements.

• Spring	Torsional	Stiffness	CROD
Spring	Axial	Stiffness	CROD
DOF Spring		Damper	CDAMP2
DOF Spring		Stiffness	CELAS2
DOF Spring		Stiffness/Damper	CELAS2
Viscous Damper	Axial/Torsional	Damper	CVISC

Springs are modeled using CROD elements, where $1.e-6$ multiplies stiffness factor in axial springs and $2.5e-6$ in torsional springs. Springs are composed of a material with very high Young Modulus.

-Surface elements: List of properties that can be assigned to surfaces.

-Shear Panel: Defines a shear panel element (SHEAR) of a structural model. A shear panel is a two-dimensional structural element that resists an action of tangential forces applied to its edges, but does not resist an action of normal forces. The structural and nonstructural mass of the shear panel is lumped at the connected nodes. (For more information see NASTRAN user manual).

- **Laminate:** Defines a laminate panel. This window is divided in two sections Laminate and Materials.

Materials section has to be filled first.

-Materials section: Defines the material property for an orthotropic material temperature independent.

E1: Modulus of elasticity in longitudinal direction, also defined as the fiber direction or 1-direction.

E2: Modulus of elasticity in lateral direction, also defined as the matrix direction or 2-direction.

NU12: Poisson's ratio ($e2/e1$ for uniaxial loading in 1-direction). Note that $u21 = e2/e1$ for uniaxial loading in 2-direction is related to $u12$, E1, and E2 by the relation $u12 E2 = u21 E1$.

G12: In-plane shear modulus.

G13: Transverse shear modulus for shear in 1-Z plane.

G23: Transverse shear modulus for shear in 2-Z plane.

-Laminate section: Defines the properties of an n-ply composite material laminate.

Distance Bottom: Distance from the reference plane to the bottom surface.

-Allowable stress: Allowable shear stress of the bonding material. Required if failure index is desired.

-Failure theory: The following theories are allowed.

HILL or the Hill theory

HOFF for the Hoffman theory

TSAI for the Tsai-Wu theory

STRESS for the maximum stress theory

STRAIN for the maximum strain theory

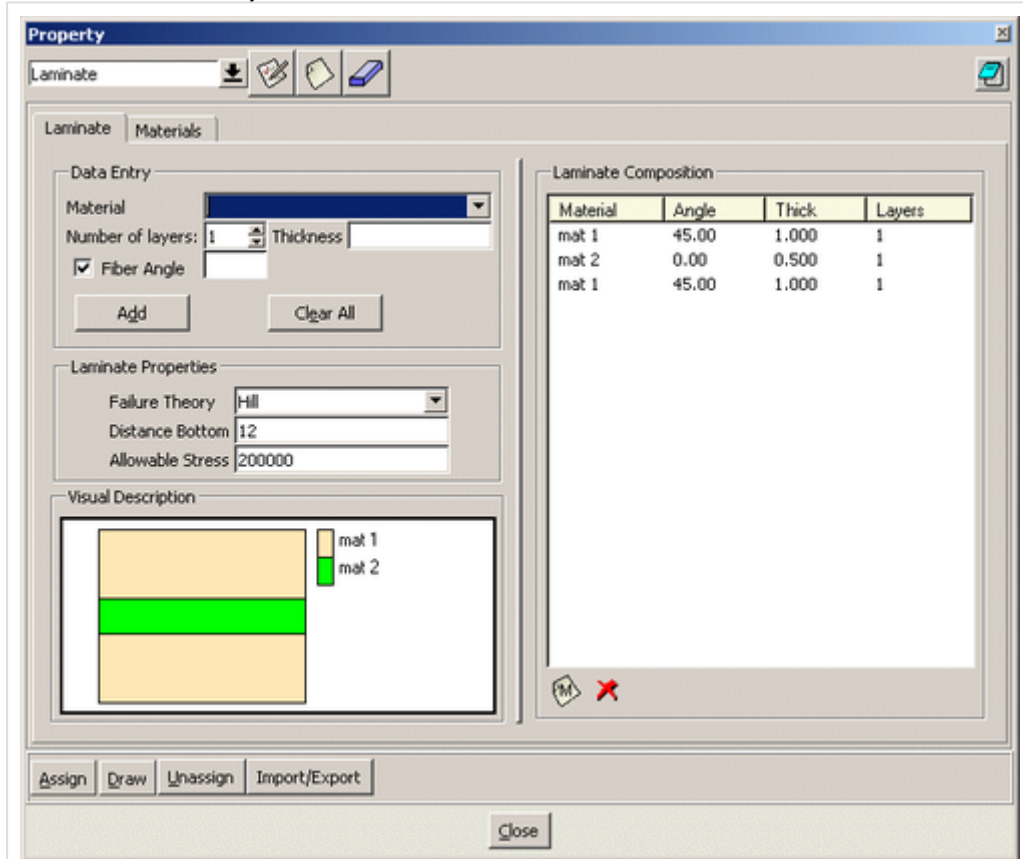


Figure 9 - Laminate window

-The rest of properties are the same with different degrees of control over the definition parameters of surface. With these properties it is possible to define a plate element of a structural model. This is an iso-parametric membrane-bending element, with variable element thickness, layered composite material, and thermal analysis capabilities.

A property that gives more control over definition parameters of surface is Plate property. Some statements were removed in the rest of the properties, which means that the program uses default values for these statements. Following is a list of default values for statements:

-Stress recovery: Fiber distances for stress computation. The positive direction is determined by the right hand rule and the order in which the nodes are listed on the connection entry. (Real or blank; defaults are $-T/2$ for Z1 and $+T/2$ for Z2.) (T=Thickness)

Bend Stiffness: Bending stiffness parameter (Real or blank, default = 1.0)

Tshear/MenThickness: Transverse shear thickness divided by membrane thickness (Real or blank; default = 0.833333)

-Volume elements: List of properties that can be assigned to volumes.

-Tetrahedron: Defines the connections of a four-sided solid element with four or ten nodes. It is possible to select between Element coordinate system and Basic coordinate system.

Basic coordinate system is global coordinate system of geometry (screen axes).

Definition of element coordinate system is the following:

The element coordinate system is derived from the three vectors R, S, and T that join the midpoints of opposite edges. R vector joins midpoints of edges G1-G2 and G3-G4S vector joins midpoints of edges G1-G3 and G2-G4T vector joins midpoints of edges G1-G4 and G2-G3The origin of the coordinate system is located at G1. The element coordinate system is chosen as close as possible to the R, S, and T vectors and points in the same general direction. (Mathematically speaking, the coordinate system is computed in such a way, that if the R, S, and T vectors are described in the element coordinate system, a 3-by-3 positive definite symmetric matrix would be produced).

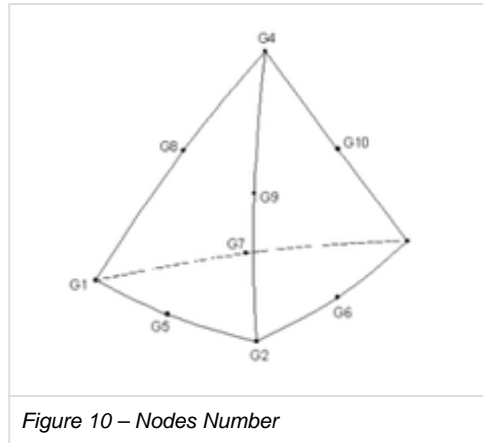


Figure 10 – Nodes Number

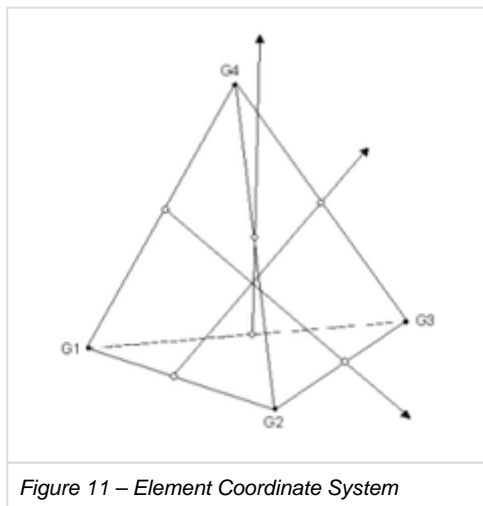


Figure 11 – Element Coordinate System

-Hexahedron: Defines the connections of a six-sided solid element with eight or twenty nodes. It is possible to select between Element coordinate system and Basic coordinate system.

Basic coordinate system is global coordinate system of geometry (screen axes).

Definition of element coordinate system is the following:

The element coordinate system for the CHEXA element is defined in terms of the three vectors R, S, and T that join the centroids of opposite faces. R vector joins the centroids of faces G4-G1-G5-G8 and G3-G2-G6-G7S vector joins the centroids of faces G1-G2-G6-G5 and G4-G3-G7-G8T vector joins the centroids of faces G1-G2-G3-G4 and G5-G6-G7-G8The origin of the coordinate system is located at the intersection of these vectors. The X, Y, and Z axes of the element coordinate system are chosen as close as possible to the R, S, and T vectors and point in the same general direction. (Mathematically speaking, the coordinate system is computed in such a way, that if the R, S, and T vectors were described in the element coordinate system a 3-by-3 positive definite symmetric matrix would be produced.)

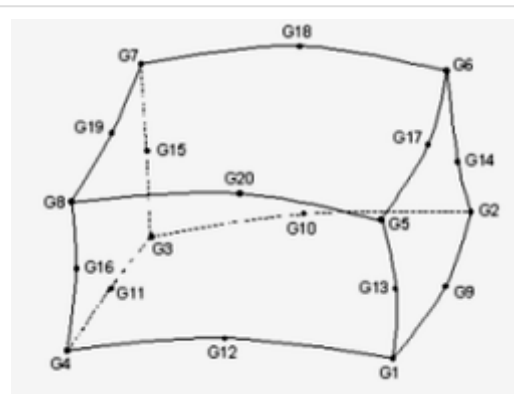


Figure 12 – Nodes Numbers

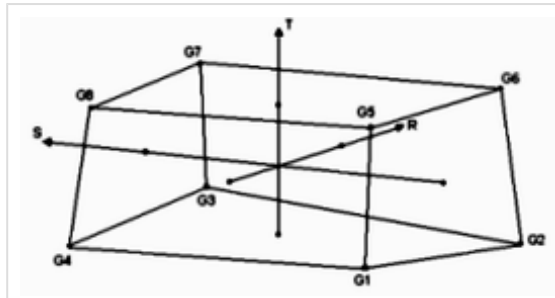


Figure 13 - Element Coordinate System

Tables

MENU SEQUENCE:

Data Loads Tables

This option is used to define a table of interpolated values. A table of interpolated values is used in the following cases:

- **Dynamic Loads:** A dynamic load is defined through a table of interpolated values. The meaning of the independent variable (X) depends on the definition domain of the load: time or frequency.
- **Thermal variation material properties:** It is possible to define temperature dependent material properties. The dependency between temperature and the property, like Young Modulus, is defined through a table of interpolated values.

It is very important to select which kind of table will be created by the use of the "Value Type" combo-box placed in top left corner of the window.

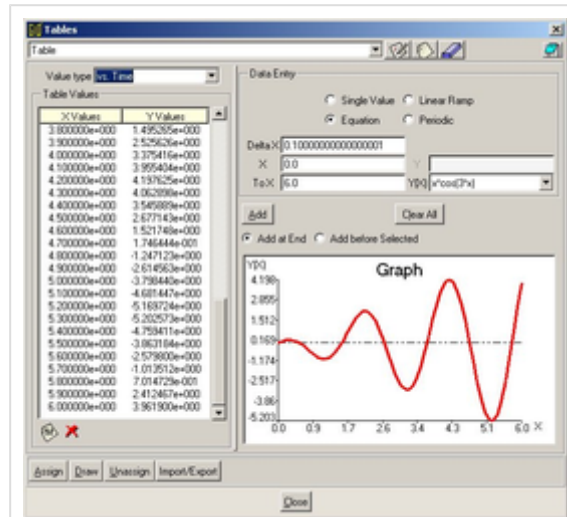


Figure 14 – Tables Window

-Data Entry Section: In this section it is possible to select which kind of entry user wants to perform. There are four different types.

- **Single value:** Allows to introduce a couple of values (X,Y).
- **Linear Ramp:** Permits to define a straight line. User has to introduce: Start point (X), value of Y coordinate at start point (Y), End point (To X), value of Y coordinate at end point (To Y) and increment for coordinate X (Delta X).
- **Equation:** It is very similar to Linear Ramp, but in this case, the user can use any kind of function to define the values of Y coordinate. To see which functions are allowed and their grammatical look at the end of this chapter.
- **Periodic:** Permits to define a periodic function, it means: a function with the same value for Y coordinate after one period.

-Add Button: After introducing the desired values to different files it is necessary to click on the add button to add the new values to the interpolation table. There are two possible ways to do this:

-Add at End: The new values are included at the end of the table.

-Add before Selected: The new values are included at the selected point of the table. For use this option it is necessary first to select a value of the table and then click on the Add Button.



With this button it is possible to delete a set of selected entries of the table.



With this button it is possible to get a selected entry (couple of values (X,Y)) from the table. It is possible to do the same with a double click of the mouse over the desired entry.

List of math function

abs(arg)

Returns the absolute value of *arg*.

acos(arg)

Returns the arc cosine of *arg*, in the range $[0, \pi]$ radians. *Arg* should be in the range $[-1, 1]$.

asin(arg)

Returns the arc sine of *arg*, in the range $[-\pi/2, \pi/2]$ radians. *Arg* should be in the range $[-1, 1]$.

atan(arg)

Returns the arc tangent of *arg*, in the range $[-\pi/2, \pi/2]$ radians.

atan2(x, y)

Returns the arc tangent of y/x , in the range $[-\pi, \pi]$ radians. *x* and *y* cannot both be 0.

ceil(arg)

Returns the smallest integer value not less than *arg*.

cos(arg)

Returns the cosine of *arg*, measured in radians.

cosh(arg)

Returns the hyperbolic cosine of *arg*.

exp(arg)

Returns the exponential of *arg*, defined as e^{**arg} .

floor(*arg*)

Returns the largest integral value not greater than *arg*.

fmod(*x*, *y*)

Returns the floating-point remainder of the division of *x* by *y*.

log(*arg*)

Returns the natural logarithm of *arg*. *Arg* must be a positive value.

log10(*arg*)

Returns the base 10 logarithm of *arg*. *Arg* must be a positive value.

pow(*x*, *y*)

Computes the value of *x* raised to the power *y*. If *x* is negative, *y* must be an integer value.

sin(*arg*)

Returns the sine of *arg*, measured in radians.

sinh(*arg*)

Returns the hyperbolic sine of *arg*.

sqrt(*arg*)

Returns the square root of *arg*. *Arg* must be non-negative.

tan(*arg*)

Returns the tangent of *arg*, measured in radians.

tanh(*arg*)

Returns the hyperbolic tangent of *arg*.

Executive Control

MENU SEQUENCE:

Data Problem Executive Control

This dialog box is used to define the Executive Control commands for your NASTRAN model.

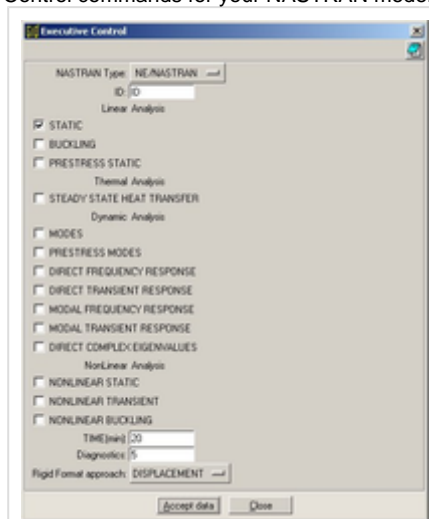


Figure 15 – Executive Control Window

-The NASTRAN type defines the type of NASTRAN code used in the analysis. At the moment, it is possible to use MI, MSC and NE NASTRAN.

-The ID is written as a title to the ID command.

-The Type of analysis defines which type of analysis will be run with NASTRAN. It is possible to select more than one analysis type at the same time, but only without any contradiction between them.

-The Time (min) option sets the maximum allowable CPU time for this analysis. If this number is very low, the analysis can prematurely terminate.

-The Diagnostics requests additional information to be generated or requests executive operations to be performed. (See NASTRAN manual for details.)

-Rigid Format approach combo box allows the user to select the Rigid Format approach or a user provided Direct Matrix Abstraction Program (DMAP). There are three possible formats:

- Displacement: Indicates one of the Displacement Approach rigid formats.
- Heat: Indicates one of the Heat Transfer Approach rigid formats. Not available in this version.
- Aero: Indicates one of the Aero-elastic Approach rigid formats. Not available in this version.

Case Control

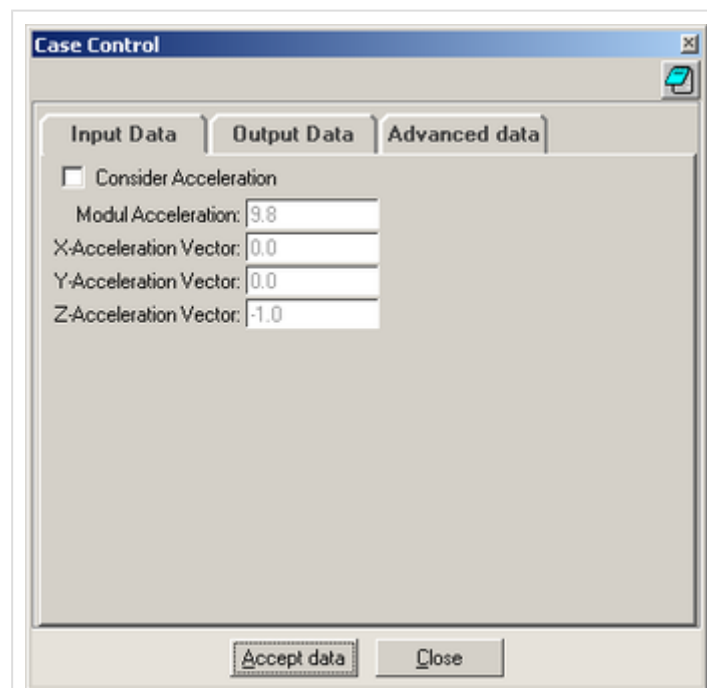
MENU SEQUENCE:

Data Problem Case Control

The case control section is used to select loads and constraints for the analysis. It can also affect the output of the analysis (i.e. determining which data is written to files for recovery).

This window has two different parts: Input Data and Output Data.

Input data



The image shows a 'Case Control' dialog box with three tabs: 'Input Data', 'Output Data', and 'Advanced data'. The 'Input Data' tab is selected. It contains a checkbox labeled 'Consider Acceleration' which is currently unchecked. Below this checkbox are four input fields: 'Modul Acceleration' with the value '9.8', 'X:Acceleration Vector' with the value '0.0', 'Y:Acceleration Vector' with the value '0.0', and 'Z:Acceleration Vector' with the value '-1.0'. At the bottom of the dialog are two buttons: 'Accept data' and 'Close'.

Figure 16 – Input data Section

-Consider Acceleration: To consider the gravity effect (weight) of a structure, this option has to be checked.

Output Data

This section controls what output will be calculated and written to the output files. Undesired output types can be turned off.

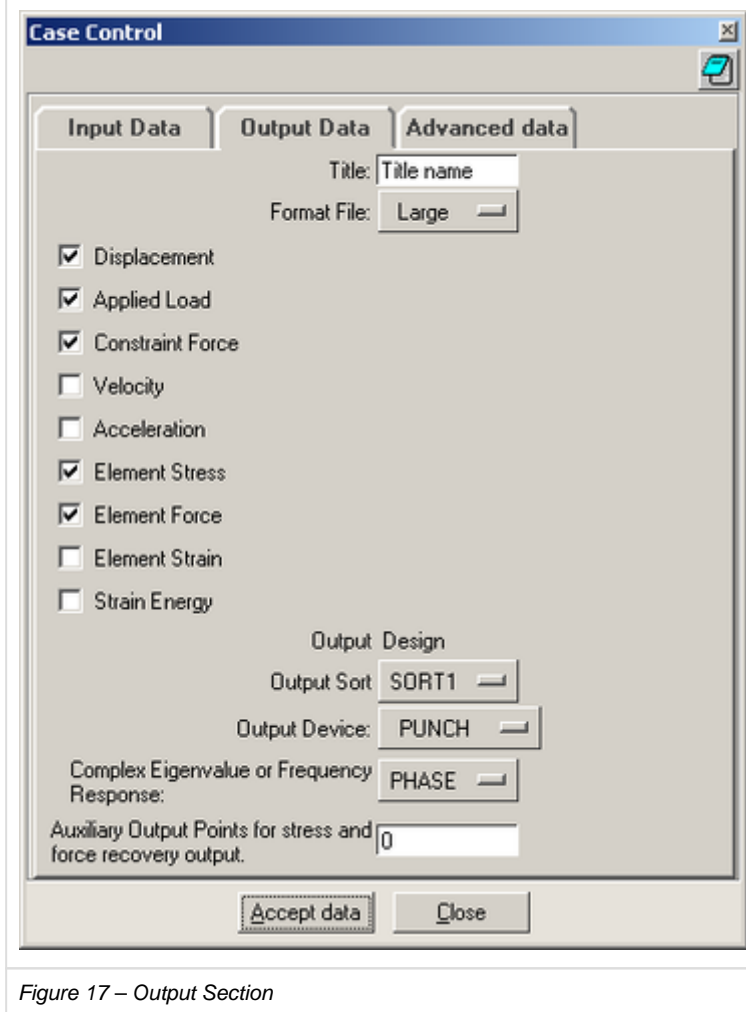


Figure 17 – Output Section

-Title: Alphanumeric strings introduced in these statements have to start with a letter. These inputs appear at the start of every page of NASTRAN's output file.

-Format File: These options determine the format that will be used for writing Bulk Data commands. By default, NASTRAN Interface uses Small field format (8 character fields). For an extra precision for some of your models, choose the large field formats (16 character fields). The large field formats produces larger files that are harder to read.

-Displacement, applied loads... These statements are used to make output requests for stresses, forces ... as well as the calculated response of degrees of freedom used in the model.

-Output Design: Sets which kind of output file presentation is desired

-Output Sort: Sets which sort is used for presentation.

SORT1: Output will be presented as a tabular listing of nodes for each load, frequency, eigenvalue or time depending on the rigid format. SORT1 is not available in transient problems (where the default is SORT2).

SORT2: Output will be presented as a tabular listing of load, frequency, or time for each node. SORT2 is available only in static analysis, transient, and frequency response problems.

-Output Device: Sets the presentation output device.

PRINT: A printer

NO PRINT: An output file. The output file will not be sent to the output device.

PUNCH: A punch file. (.pch). This output file can be used in post process of GiD. (More information can be found in the Post Process chapter).

-Complex eigenvalue or frequency response: Sets which kind of representation is desired for complex numbers.

PHASE: Requests magnitude and phase ($0.0^\circ \leq \text{phase} < 360.0^\circ$) on complex eigenvalue or frequency response problems.

REAL o IMG: Requests real or imaginary output on complex eigenvalue or frequency response problems.

-Auxiliary Output Points: Defines a series of points along the axis of a bar element (CBAR entry) for stress and force recovery output. A maximum of six internal points can be specified.

Remarks

1. All outputs have the same presentation.

2. An output request for ALL in transient and frequency response problems generally produces large amounts of printout. An alternative to this can be to define a set of parts of the model that are of interest, using output sets condition (For more information see the Advanced Conditions chapter).

Advanced Data

The screenshot shows the 'Case Control' dialog box with the 'Advanced Data' tab selected. The dialog has three tabs: 'Input Data', 'Output Data', and 'Advanced Data'. The 'Advanced Data' tab contains the following fields and controls:

- Subtitle:** A text box containing 'Subtitle name'.
- Label:** A text box containing 'Label name'.
- Lines per printed page:** A text box containing '50'.
- Maximum number of output lines:** A text box containing '20000'.
- ECHO:** A dropdown menu currently set to 'NONE'.

At the bottom of the dialog, there are two buttons: 'Accept data' and 'Close'.

Figure 18 – Advanced Data

-Lines per printed page: Sets the number of data lines per printed page.

-Maximum number of output lines: Sets the maximum number of output lines (default 20.000).

-ECHO: Requests echo of Bulk Data Section.

OptionsMeaning

SORTSorted echo will be printed

UNSORTUnsorted echo will be printed

BOTHBoth sorted and unsorted echoes will be printed

NONENo echo will be printed

PUNCHThe sorted Bulk Data Section will be punched onto statements

Dynamics

MENU SEQUENCE:

Data Problem Data Dynamics

The Dynamics section is used to set different parameters of dynamic analysis problems and for eigenvalues extraction (modes analysis). This window consists of two parts: Modes Analysis and Dynamic Design.

Modes Analysis

Defines data needed to perform real eigenvalue analysis.

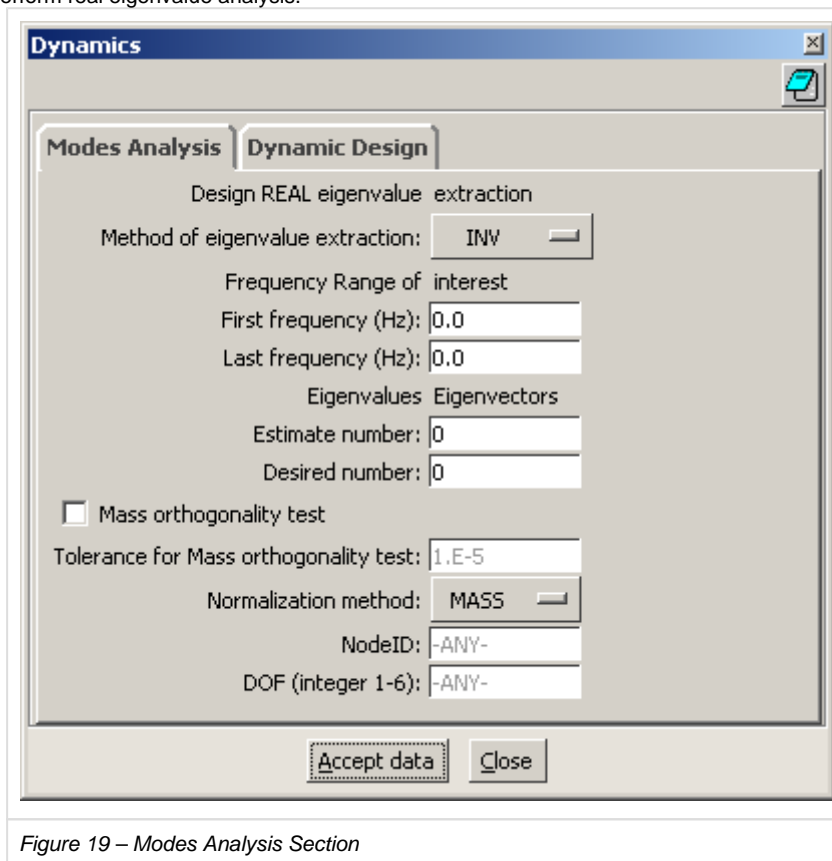


Figure 19 – Modes Analysis Section

-Method of eigenvalue extraction: Sets which method will be used to find eigenvalues. See the following table:

INV	Inverse power method, symmetric matrix operations.
DET	Determinant method, symmetric matrix operations.
GIV	Givens method of tridiagonalization.
MGIV	Modified Givens method (see Remark 2).
FEER	Tridiagonal reduction method, symmetric matrix operations.
FEER-Q	See Remark 1.
FEER-X	See Remark 1.
UINV	Inverse power method, un-symmetric matrix operations.
UDET	Determinant method, un-symmetric matrix operations.

Remarks

1. The rigid body frequencies are set to zero, unless FEER-X is requested. If FEER-Q is requested, certain key areas in FEER computations are done in quad precision (Real*16) for 32-bit word machines and in double precision for 60- and 64-bit word machines. The FEER-Q request would yield much better rigid body eigenvalues, but it may take two to three times longer to compute than FEER or FEER-X.
2. Givens method requires the mass matrix not to be singular. The MGIV method allows a singular mass matrix. The bigger dynamic matrix will require more CPU time and more memory.

-Frequency range of interest: (Required for METHOD = DET, INV, UDET, or UINV) (Real ≤ 0.0 ; $F1 \leq F2$). If METHOD = GIV, frequency range acts over desired eigenvectors. The frequency range is ignored if ND > 0. In this case the ND number of first eigenvectors associated to positive roots is found. (Real, $F1 \leq F2$).

If METHOD = FEER, F1 is the center of range of interest (Default is $F1 = 0.0$) (Real > 0.0), and F2 is an acceptable relative error of tolerance expressed as percentage on frequency-squared (Default percentage is $0.1/n$, where n is the order of the stiffness matrix) (Real > 0.0).

-Estimate number of roots in range (Required for METHOD = DET, INV, UDET, or UINV. Ignored for METHOD = FEER) (Integer > 0).

-Desired number

The value of this field always has to be a positive integer.

This field has the following functions:

- For methods DET, INV, UDET, or UNIV: represents desired number of roots. If left blank, the default of '3 * Estimated number' is taken.
- For method GIV: represents desired number of eigenvectors.
- For method FEER: represents desired number of roots and desired number of eigenvectors. If left blank, an automatic numbers are calculated to extract at least one accurate mode.

-Mass orthogonality test: Method for normalizing eigenvectors.

The following are the available methods:

MA SS	Normalizes eigenvectors to unit value of the generalized mass.
MAX	Normalizes eigenvectors to unit value of the largest component in the analysis set.
POI NT	Normalizes eigenvectors to unit value of the component defined in fields NodeID and DOF. If the defined component is set to zero, the method is changed to MAX by default.

-NodeID: Node or scalar point identification number (Required if and only if Mass orthogonality test is set to POINT). The value of this field always has to be a positive integer.

Dynamic Design

Provides the solution type and control information for Dynamic Analyses. Each dynamic load used in a Dynamic Analysis must have the appropriate Solution Method activated.

Dynamics

Modes Analysis **Dynamic Design**

Solution Method: Direct

Domain of Solution: Transient

Equivalent Viscous Damping

Overall Structural Damping Coeff(G): 0.0

Equivalent Viscous Damping Conversion

Frequency for System Damping(W3-Hz): 0.0

Frequency for Element Damping(W4-Hz): 0.0

Transient Time Step

Number of time steps: 2

Time increment: 0.0

Skip factor for output: 0

Frequency Step

Initial frequency: 0.0

Frequency increment: 0.0

Number of frequency increments: 0

Mass formulation: Default

Response Based on Modes

Number of Modes: 0

Lowest Freq(Hz): 0.0

Highest Freq(Hz): 0.0

☐ Select Range of frequencies for Output

From(Hz): 0.0

Output Number of frequency increments: 0

Accept data Close

Figure 20 – Dynamics Section

- Solution Method and Domain of Solution: chooses the type of Dynamics Solution to be performed. The four following options are available: (1) Direct Transient, (2) Modal Transient, (3) Direct Frequency, and (4) Modal Frequency. The inappropriate boxes for each Solution Method will be inactivated automatically.

-Equivalent Viscous Damping: These statements provide damping information of a structure. The Overall Structural Damping Coefficient is an input for all four-solution methods.

-Equivalent Viscous Damping Conversion: Information for both System Damping and Element Damping is provided in this box. These values are only inputs for Direct and Modal Transient Analysis.

-Transient Time Step Interval: For transient analyses, these options control the number of steps, steps sizes, and the output interval.

-Frequency Step: The Solutions Frequencies are defined in this section. These statements define analyzed frequencies for both Direct and Modal Frequency Analysis.

-Response Based On Modes: For modal solution methods, these options allow choosing the number and/or range of modes to include in the frequency response or transient formulation.

-Select Range of Frequencies for Output: Allows to set which frequencies will be printed in the output file. In some cases it is interesting to run analysis for more frequencies that can be studied later.

Buckling

MENU SEQUENCE:

Data Analysis Design Bucking

The buckling section is designed to set the features of a buckling analysis.

Buckling analysis can be performed in two different ways:

- Linear buckling analysis: This type of analysis will compute desired modes of buckling of the model.

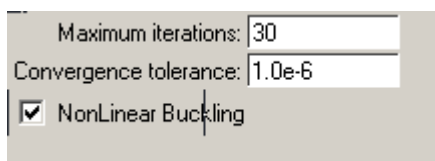
To perform this analysis, it is necessary to set values for the modes extraction process:

-Eigenvalue range of interest: Selects range of interest for buckling modes.

-Number of desired roots: Number of desired modes.

-Strum sequence: If this option is selected, the analysis will take a longer time, but the numerical solution will be more accurate.

- Nonlinear buckling analysis: Nonlinear buckling analysis is similar to the linear one except that a nonlinear solution is used to form the tangent stiffness matrix. To perform this analysis it is necessary to check *nonlinear buckling option* in the bottom of the window and to define all necessary parameters in the *nonlinear section*.



Maximum iterations: 30

Convergence tolerance: 1.0e-6

☒ NonLinear Buckling

Nonlinear Analysis

MENU SEQUENCE:

Data Analysis Design Nonlinear

The Nonlinear Analysis section is used to set different parameters of nonlinear analysis problems. This section has to be defined when a nonlinear analysis of any type (static nonlinear, dynamic nonlinear, buckling nonlinear...) is performed.

It is highly recommended to take a look to NASTRAN manual to see how to set the values of every statement, especially to NLPARM and NLTSEPT NASTRAN statements.

The following is a brief description of section fields:

-Analysis Type: Determines the type of solution that will be performed for the particular load set. Available options are Static, Buckling, and Transient. Only appropriate control information in the remainder of the window will be available based upon the type of solution you choose.

-Iterations: These values provide the time and iteration control information for the nonlinear analysis steps. They control the *Number of Increments* and the *Time Increment* to be used, as well as the *Maximum Iterations* for each step. No time increment is used for static analysis.

-Stiffness Updates: Specifies the number of iterations to be performed before the stiffness matrix is updated, as well as the update *Method*.

-Skip factor: Allows you to request or eliminate output at every n^{th} step (dynamic).

-Convergence Tolerances: The type of *Convergence Tolerances* (*Load*, *Displacement*, and/or *Work*) as well as the tolerance values themselves are defined in these boxes.

Parameters

MENU SEQUENCE:

Data Problem Data Parameters

Specifies values for parameters used in DAMP sequences (including rigid formats). Parameters are organized in different sections to do easier their localization.

If the user doesn't have an experience with NASTRAN format, it is recommended to keep the default values. For more information see the NASTRAN manual.

Verify Properties

MENU SEQUENCE:

Data Verify Properties

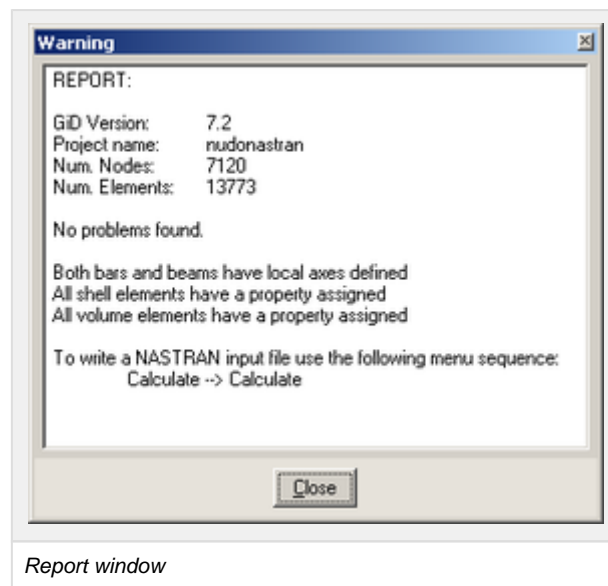
Checks properties of the model for errors.

Checking features:

- Finds elements with any property assigned.
- Checks compatibility of the assigned properties. For example linear mesh elements with beam property assigned has to have local axes defined.
- Creates a report list of elements with errors.

Note:

Depending on size of the model, checking the entire model might take few minutes.



Obtain input file for NASTRAN

MENU SEQUENCE:

Files Import/Export Calculation file

After the mesh is generated and all problem data assigned, follow the above menu sequence to obtain the NASTRAN input file.

Another option is use calculate option in menu calculate:

Calculate Calculate

Post processing

To see results of NASTRAN in the GiD post processing, the interface offers to different options

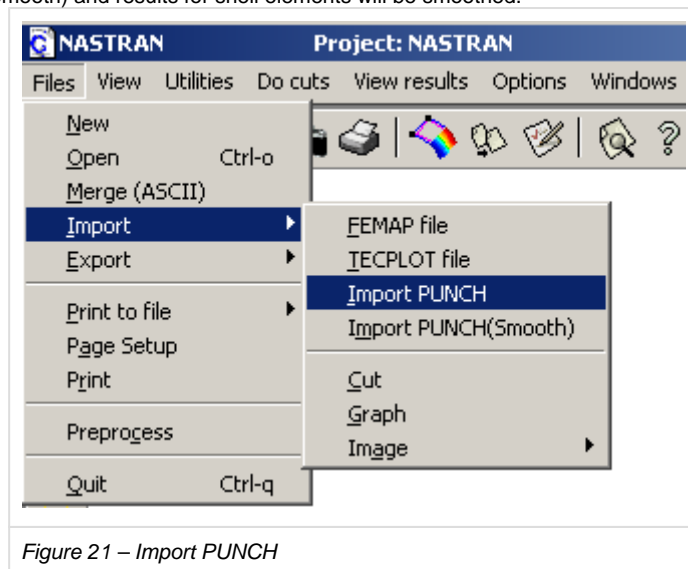
a) An importing tool for punch files (.pch) of NASTRAN.

For information about obtaining a punch file (.pch), see Output Data menu of Case Control (output device statement).

To import a punch file of NASTRAN, do this:

1. Go to the post process: Click this icon  of the toolbar
2. Click: Files->Import ->Import PUNCH or

Files->Import ->Import PUNCH(Smooth) and results for shell elements will be smoothed.



1. Search your punch file of your model.

Note: The Import PUNCH only appears in the Import menu if NASTRAN problem type is loaded.
For more information about post process of GiD see GiD manual.

b) Import FEMAP ASCII neutral file:

- Import the FEMAP ASCII file (*.neu) obtained from NE/NASTRAN.

File Import FEMAP file

Appears a window to select the *.neu file located in NASTRAN results folder.

After importing process is finished close the window.

Note: To obtain a FEMAP ASCII file in NE/NASTRAN go to NASTRAN editor:

Setup -> Default Analysis Options

Selects RSLTFILETYPE and set to FEMAP ASCII in Output Control Directives

Importing NASTRAN Models

With GiD-NASTRAN Interface is also possible to import a NASTRAN Model. This feature allows to the user to import all NASTRAN model information. Materials, conditions and problem general data will be imported and loaded in the interface from the NASTRAN model.

After the importing process is possible assign new conditions, unassigned existing ones and create a new NASTRAN model for new simulations.

This feature in the way of total integration of GiD and NASTRAN codes is available from version 2.3beta.

To import a NASTRAN Model following the steps below:

1- Load GiD-NASTRAN Interface in GiD:

Data Problem Type Nastran

2- Import NASTRAN Model:

Files Import NASTRAN mesh...