

# **Nastran-GiD interface**

## **Tutorials**

# Nastran tutorials

1 Static Analysis of a Beam .....	4
2 Static Analysis of a Gear .....	13
3 Direct Frequency Response Analysis .....	19
4 Modal Analysis of a Beam .....	27

## Nastran tutorials

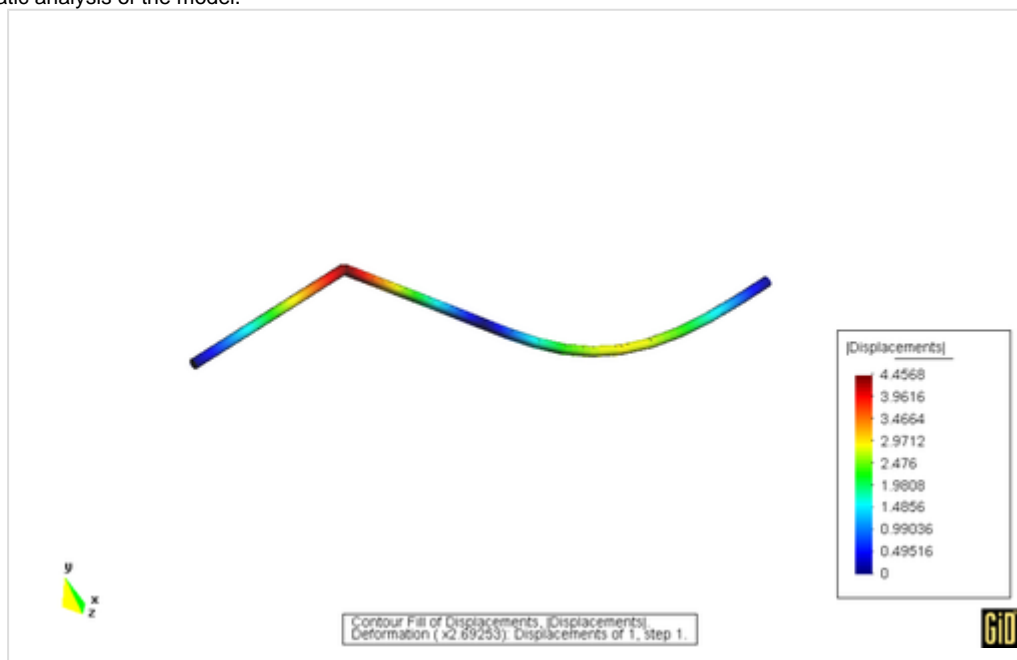


## Static Analysis of a Beam

This example describes a static linear elastic analysis of a beam with double T section under several static loads.

The following list is a summary of all steps of tutorial:

- Create geometry of a beam with double T section.
- Create a finite element model of this beam.
- Run static analysis of the model.

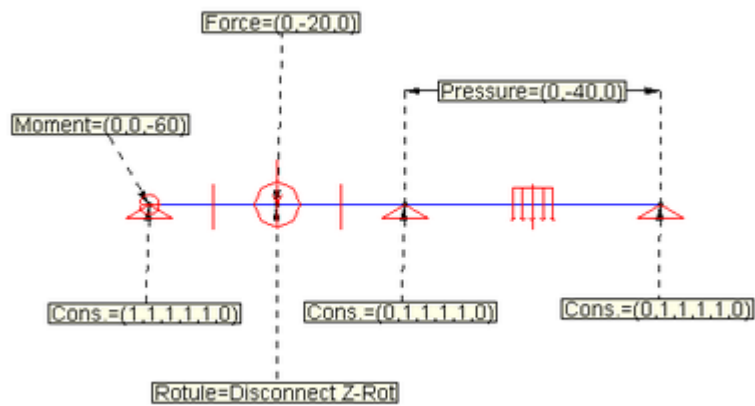


### Description of Model:

Determine displacements and reactions in pinned points of a beam with double T section under several static loads.

### Properties:

Length	100 inch	
Height (Section)	2 inch	
Width (Section)	1 inch	
Thickness (Section)	0.1 inch	
$I_1$	0.229 inch <sup>4</sup>	
$I_2$	0.017 inch <sup>4</sup>	



For basic reference, this example can be found in the distribution, directory Examples/Nastran and with the name **staticbeam.gid**

Procedure:

Start:



- 1- Open a new project, right click on this icon in taskbar
- 2- Load problem type Nastran. Follow the menu sequence below:  
Data -> Problem type nastran  
Appears a splash image and the name of the master window changes to NASTRAN Interface.
- 3- Change the view plane to XY:  
View -> Rotate -> Plane XY (original)

Create geometry:

1. Create line.

Geometry Create Straight line

Insert coordinates of points to the command line to define the beam (line). Command line is in the bottom of GiD master window.

- 1<sup>st</sup> point -> 0,0
- 2<sup>nd</sup> point -> 30,0
- 3<sup>rd</sup> point -> 60,0
- 4<sup>th</sup> point -> 120,0

(It is only necessary to introduce points using two coordinates, the third coordinate Z is assumed to be 0).

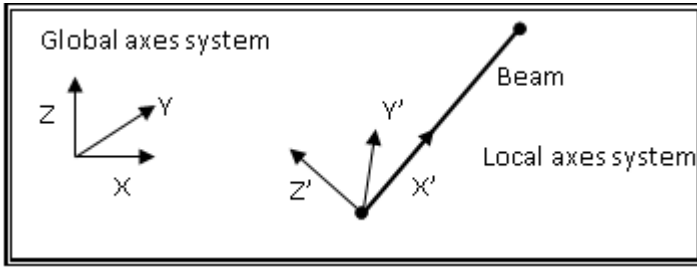
Press escape or middle button of mouse.

## Define 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' axe 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.

1. **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.
2. **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.
3. **User defined.** User can create 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.

In this example we will assign automatic local axes:

Data -> Properties -> Local Axes

Set statement Local Axes to Automatic (An automatic local will defined for all lines selected).

Select all lines of the geometry.

## Assign property and material:

1. Define a new material.

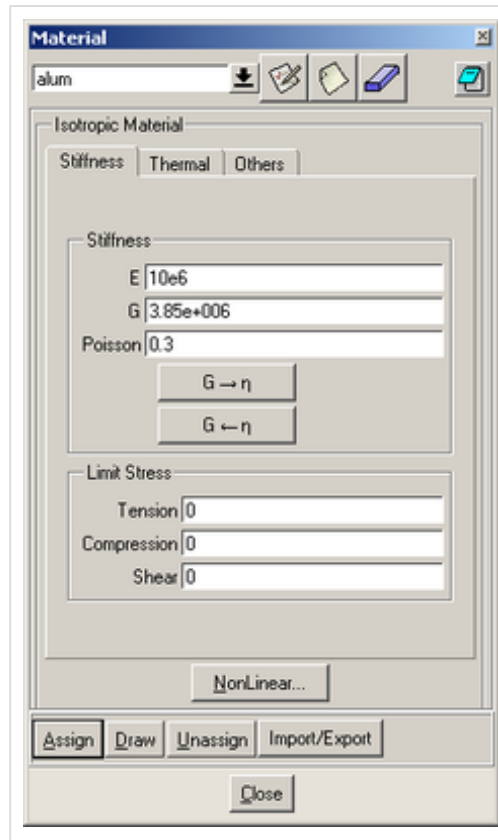
Data Materials



Click on the following icon to create a new material:

Enter name for the new material **alum**.

Fill all statements like in the picture:



And set density, in Others page, to 0.101



Click on the following icon to save the new material:

2- Define and assign properties for the beam.

Data Properties Property



You can go directly to the property window by clicking on this icon: without closing the material window and going back to the menu.

Select option *property* in the menu.

Select property *beam* from the top combo box.



Click on the following icon to create the new property:

Enter a name for the new property '*cantilever*'.

Fill all statements as in the following picture:

Property

cantilever

PROPERTY BEAM

Area: 0.38

Moments of Inertia: I

I1: 0.229

I2: 0.017

I12: 0.0

Torsional Constant: 0.0

Y Shear Area: 0.0

Z Shear Area: 0.0

Nonstructural mass/length: 0.0

Y Neutral Axis Offset: 0.0

Z Neutral Axis Offset: 0.0

Stress Recovery 2\_to\_4\_Blank=Square

Values: Y Z

Composition Material: alum

Create Material

Assign Draw Unassign Exchange

Close

You have to select the previously created material in Composition Material.



Click on the following icon to save the new property:

Now click the **Assign** button, and select all lines of the geometry.

To see if the property is well assigned click on **Draw** button, select *This property* option.

The NASTRAN Interface program window should look like this:



### Assign Constraints:

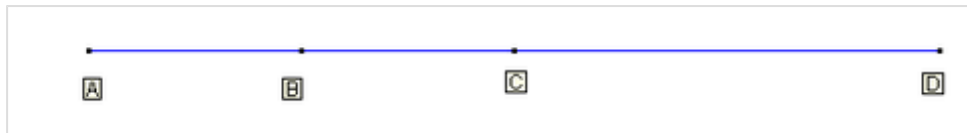
1- Assign prescribed displacements and rotation.

Data-> Boundary Conditions -> Constraints

Click on the following icon to set the condition over points:







Assign the following prescribed movements:

	X-Displ	Y-Displ	Z-Displ	X-Rot	Y-Rot	Z-Rot
Point A	1	1	1	1	1	0
Point C	0	1	1	1	1	0
Point D	0	1	1	1	1	0

### Connections

1. Disconnect degree of freedom from point B.

Data Boundary Conditions Connections  
Checks disconnect Z-Rotation statement.

Click on **Assign** button and select point B.

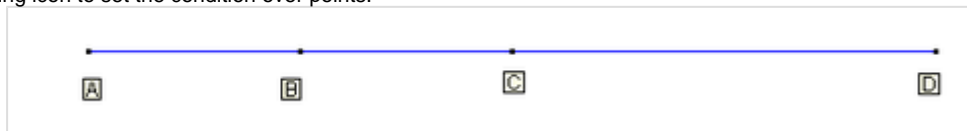
With this condition it is possible to disconnect some degrees of freedom of the union points of some beams. In this example this condition creates a 2-D ball-joint in XY-plane.

### Assign Static Loads

1. Assign static punctual loads.

Data Loads Static Loads

Click on the following icon to set the condition over points:



Point A

Select from the top combo box static load *Moment*.

(Mx-Force, My-Force, Mz-Force) = (0,0, -60)

Click on **Assign** button and select point A.

Point B

Select the static load *Point-Force-Load* from the top combo box.

(X-Force, Y-Force, Z-Force) = (0, -20, 0)

Click on **Assign** button and select point B.

- 2- Assign static distributed loads.

Data Loads Static Loads

Click on the following icon to set the condition over lines:



Line from C to D

Select from the top combo box static load *Line-Pressure-Load*.

Coord. System = BASIC

(X-Pressure, Y-Pressure, Z-Pressure)= (0, -40, 0)

Click on **Assign** button and select the line C to D.

### Mesh the Geometry:

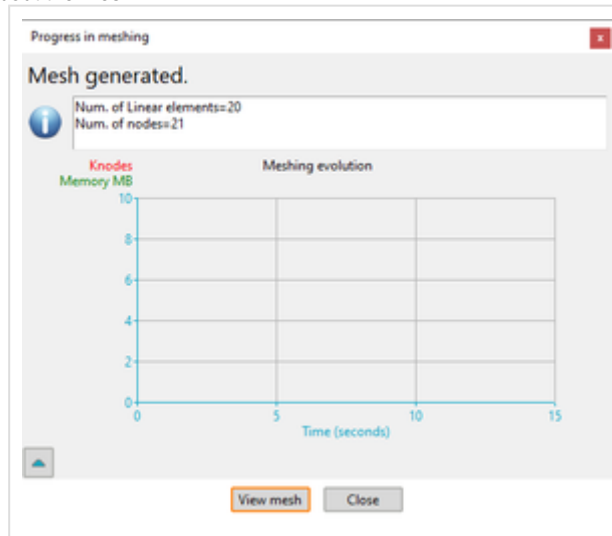
1-Create a mesh.

Meshing Generate

Now you are asked about the size of elements to be generated, change to value 6.

Click on **OK** button

Appears a window with information about the mesh:



Num. of linear elements = 20

Num. of nodes = 21

### Perform the Analysis:

1. Design Executive Control Section.

Data Problem Data Executive Control

Select type of NASTRAN will be use in the analysis.

Check STATICS and leave all the other statements uncheck.

Leave the rest of statements with the default values.

Click on **Accept Data** button.

2 - Design Case Control Section.

Data Problem Data Case Control

2.1. - Input data

Leave all statements with default values.

2.2. - Output data

Set Title to "**Beam\_example**"

Select which format file you want to use:

-Small: Every file of a Bulk Data statement will use the 8-characters definition.

- Large: Every file of a Bulk Data statement will use the 16-characters definition.

Leave Subtitle, Label ... and Post process with default values.

Check Displacements and constraint forces and uncheck the rest of output requests.

In the Output Design section leave the default values.

*Note: If you want to post process the results of MI/NASTRAN analysis with NASTRAN Interface, you have to set the output device to PUNCH.*

Click on **Accept Data** button.

### Obtain Input File for NASTRAN Code:

### Option 1:

Calculate Calculate

### Option 2:

Files Export Calculation File

Select a folder and a file name to write the file. It is very important to write the correct extension of the NASTRAN input file (i.e. \*.nas, \*.dat, \*.nid ...).

Post process:



Click on the following icon to enter in the post process:

### Import punch file:

1. Import the punch file (\*.pch) obtained from NASTRAN.

Files Import Import PUNCH

When the file import process is finished, close the file-find window.

### Import FEMAP ASCII neutral file:

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

Files Import FEMAP file

When the file import process is finished, close the file-find window.

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

Setup Default Analysis Options

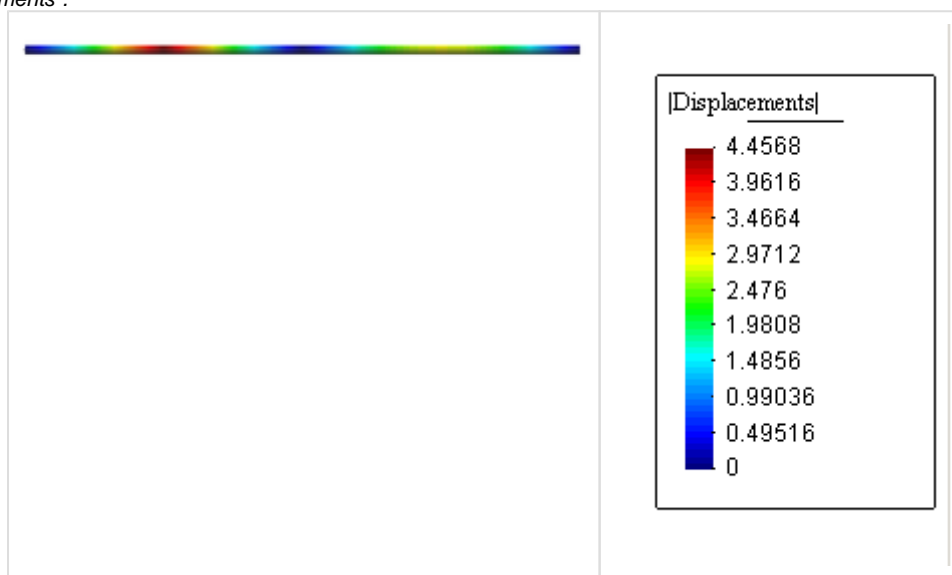
Selects RSLTFILETYPE and set to FEMAP ASCII in Output Control Directives

### Visualization of results:

1. Contour Fill.

View results Contour Fill

Select "Displacements".

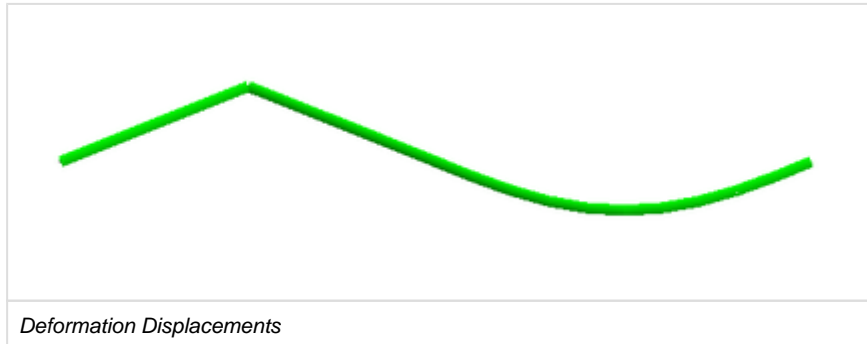


**Contour Fill**

Displacements

## 1. Deformation

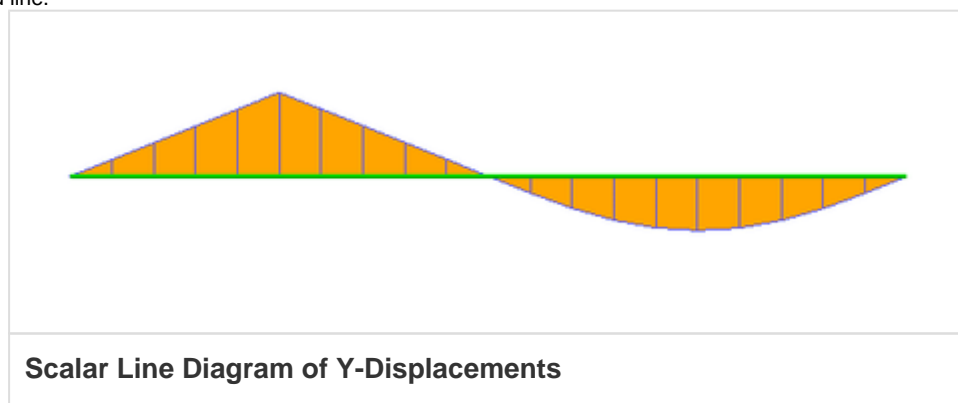
View results Deformation  
Select "*Displacements*".



## 1. Line Diagram

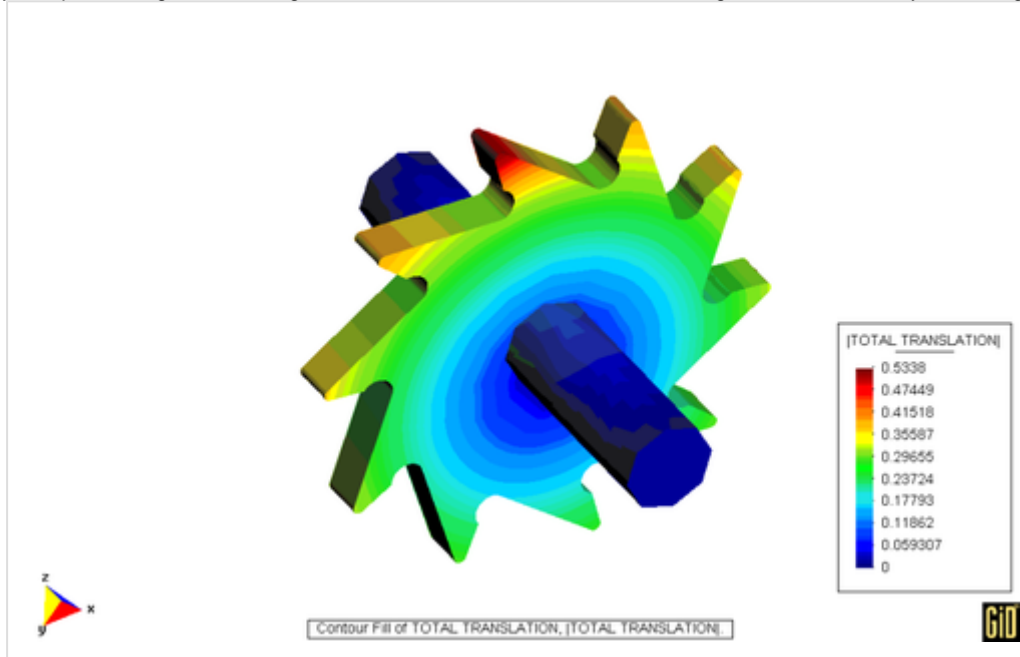
View results Line Diagram  
Select *Scalar* and option Y-Displacements

Now change the setting for line diagram.  
Options Line Diagrams Show elevations  
Select option Filled line.



## Static Analysis of a Gear

Create geometry to represent a gear with octagonal axis. Create a finite elements model of gear. Run static analysis of the gear.

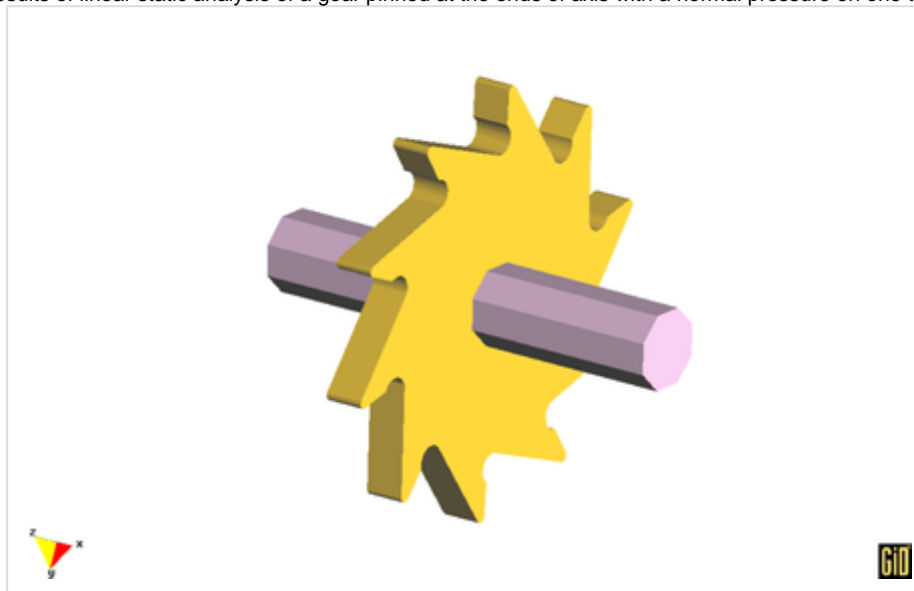


Purpose:

Create geometry to represent a gear with octagonal axis. Create a finite elements model of gear. Run static analysis of the gear.

Description of the Model:

Determine different results of linear static analysis of a gear pinned at the ends of axis with a normal pressure on one tooth.



Procedure:

Start:

1. Open GiD file called pieza.gid. This file is localized in the Examples folder in the GiD installation directory.

To get this file, use the Internet Retrieve option of GiD and download Nastran Examples.  
 Internet Retrieve option is available under:  
 Data -> Problem Type -> Internet Retrieve  
 For more information see GiD manuals.

### Comment:

If you want to create the gear's CAD model manually without downloading it, see the Post-process Tutorial of GiD, which can be obtained from the GiD web page in Support section.

### Preferences:

With the target to obtain a good mesh some changes in meshing preferences of GiD have to be made.

1. Open Preferences window:

Utilities -> Preferences

1. Select meshing tab and set window entries to the following values:

Surface mesher: Rsurf  
 Mesh until end  
 Automatic correct sizes  
 Unstructured size transition: 0.6  
 Smoothing: High Angle  
 No mesh frozen layers  
 Allow automatic structured

These settings will prevent the creation of tetrahedron elements with seek angle bigger than 80 degrees. Please note, that gear's CAD model is designed like unique volume to allow the creation of a correct finite elements model.

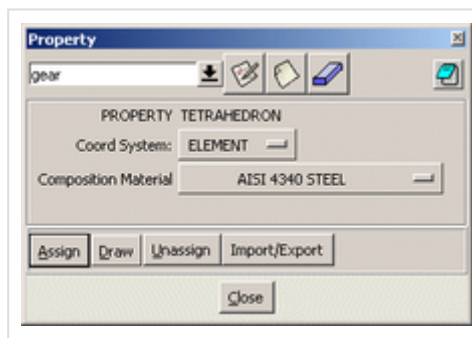
### Assign Property:

- 1- Define and assign the property for the surface.  
 Data -> Properties -> Property  
 Select from the top combo boxes the property *Tetrahedron*.



Click on the following icon to create a new property:

Enter name for the new property **gear**.  
 Fill all statements like in the picture:

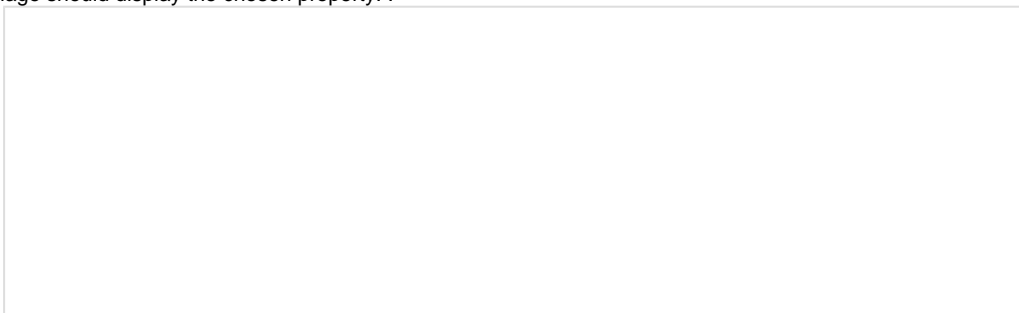


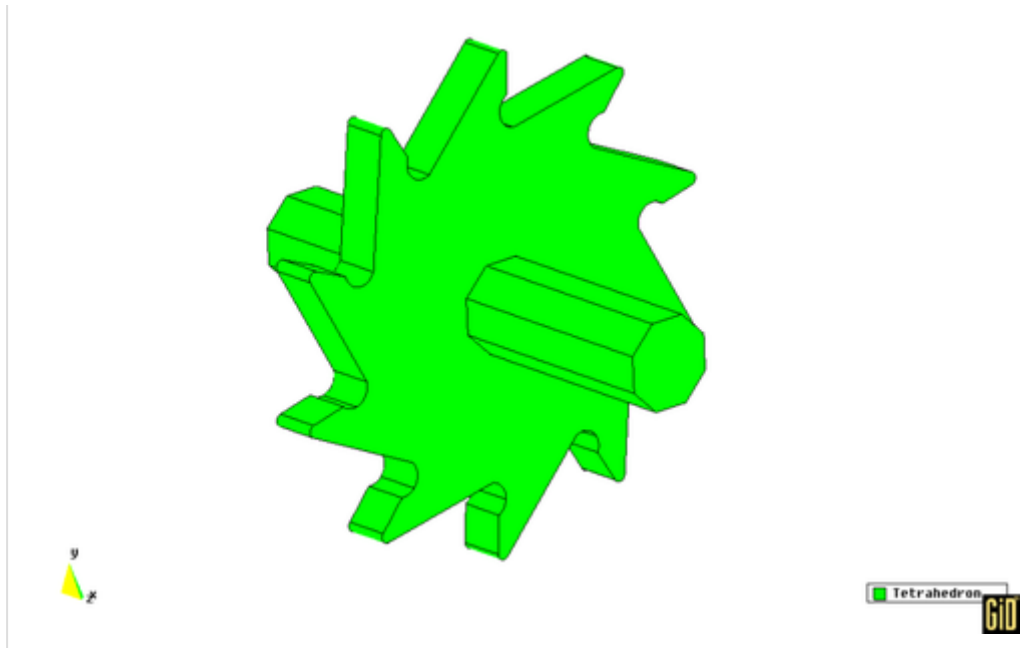
Make sure, that a one of the default materials is defined in Composition Material.



Click on the following icon to save the new property:

Now click on **Assign** button, and select the unique volume of the geometry.  
 To make a proof if the property is well assigned click on **Draw** button, select *This property* option.  
 The following image should display the chosen property. :





### Assign Constraints:

1- Assign prescribed displacements and rotation.

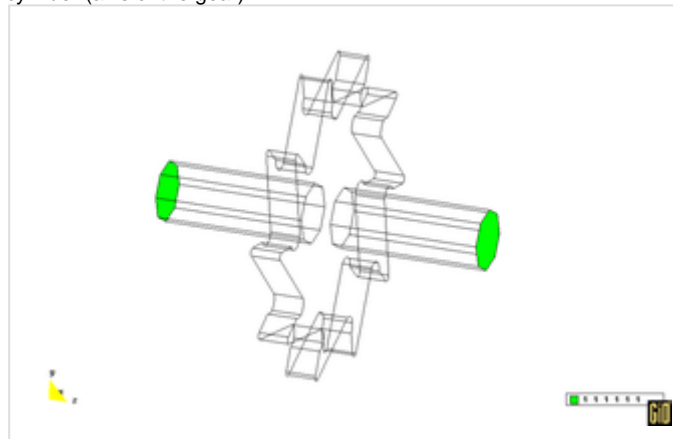
Data-> Boundary Conditions -> Constraints

Click on the following icon to set the condition over surfaces:



The model has the two lids of octagonal axis pinned, which means that all degrees of freedom have to be prescribed. To do this it is necessary to check all statements.

2- Select the two lids of octagonal cylinder (axis of the gear).



### Assign Static Loads

Assign Normal distributed pressure over one tooth of the gear.

Data -> Loads -> Static Loads

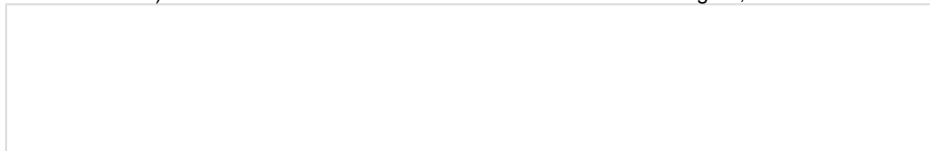
Click on the following icon to set the condition over surfaces:

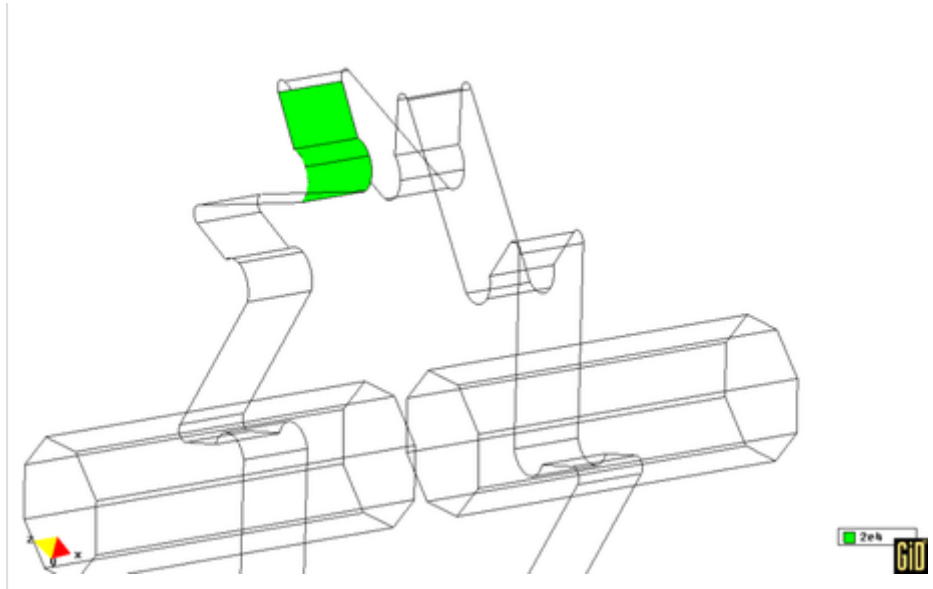


From the top combo box static load select *Normal-Surface-Load*.

Set the value of the load to **2e4** (positive sign means that

load is oriented inward to the surface) and select the three interior surfaces of one tooth of the gear, as stated in the following image:





### Perform the Analysis:

Design Executive Control Section.

Data -> Problem Data -> Executive Control

Select the NASTRAN Type that will be used for the analysis.

Check STATICS and leave all the other statements unchecked or with default values.

Click on **Accept Data** button.

2 - Design Case Control Section.

Data -> Problem Data -> Case Control

2.1. - Input data

In this example we don't consider the acceleration due to the gravity. Leave all statements with default values.

2.2. - Output data

Set Title to "Gear\_example"

Select which format file you want to use:

-Small: Every file of a Bulk Data statement will use the 8-characters definition.

- Large: Every file of a Bulk Data statement will use the 16-characters definition.  
Check Displacements, constraints forces, elements forces and elements stresses, uncheck the rest of output requests.  
In Output Design section leave the default values.  
Note: The output device has to be set to PUNCH for post-processing with MI/NASTRAN Interface.  
Click on **Accept Data** button.

### Obtain Input File for NASTRAN Code:

#### Option 1:

Calculate -> Calculate

#### Option 2:

File -> Import/Export -> Write Calculation File

In the Save File window, select the filename and location. Please note, it is very important to use the appropriate NASTRAN file extension.

Post process:



Click on the following icon to enter in the post process:

#### Import punch file:

Import the punch file (\*.pch) obtained from NASTRAN.

File -> Import -> Import PUNCH



Select the \*.pch file located in NASTRAN results folder.  
After the importing process is finished, close the window.

#### Import FEMAP ASCII neutral file:

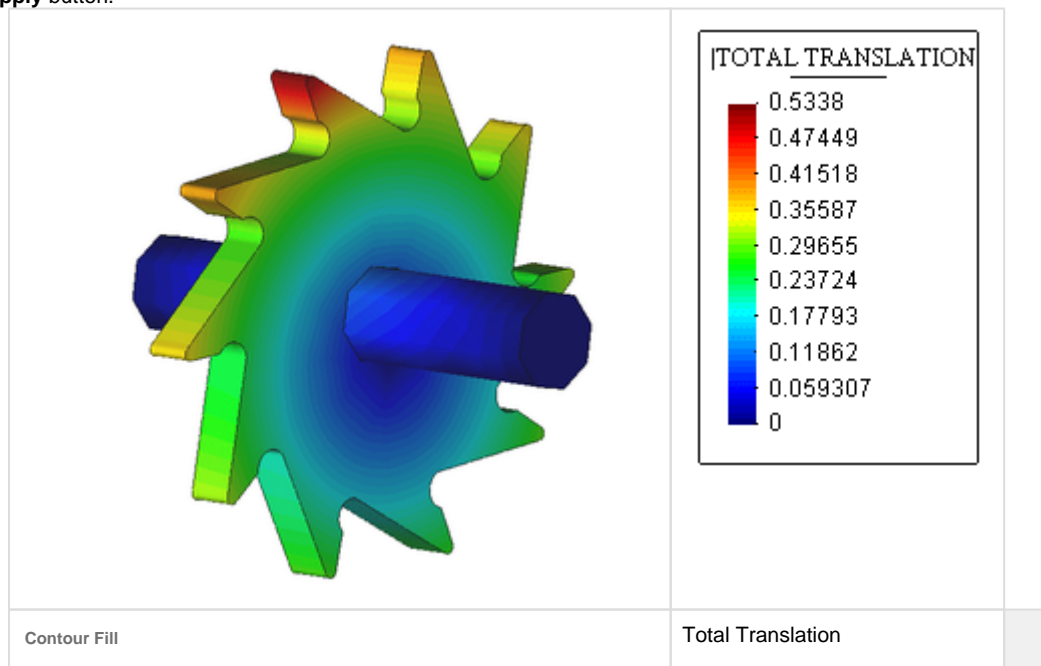
Import the FEMAP ASCII file (\*.neu) obtained from NE/NASTRAN.  
File -> Import -> FEMAP file  
Select the \*.neu file located in NASTRAN results folder.  
After the importing process is finished close the window.  
Note: To obtain a FEMAP ASCII file in NE/NASTRAN go to NASTRAN editor:  
Setup -> Default Analysis Options  
Selects RSLTFILETYPE and set to FEMAP ASCII in Output Control Directives.

#### Results visualization:

Open View results window.  
Windows -> View Results

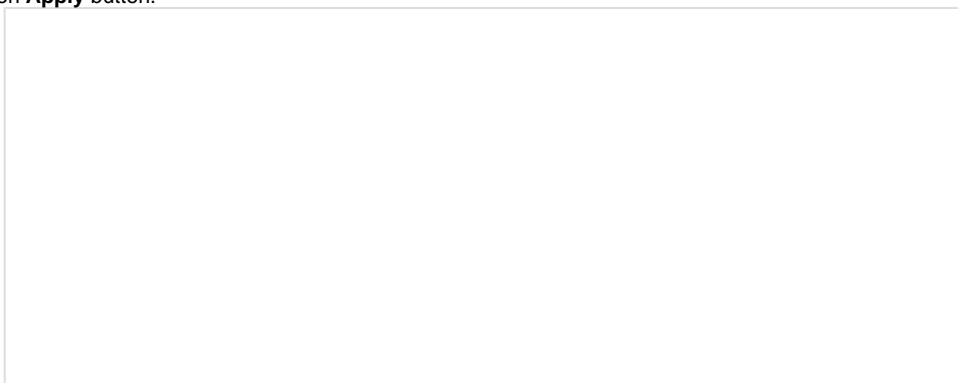
#### Contour Fill

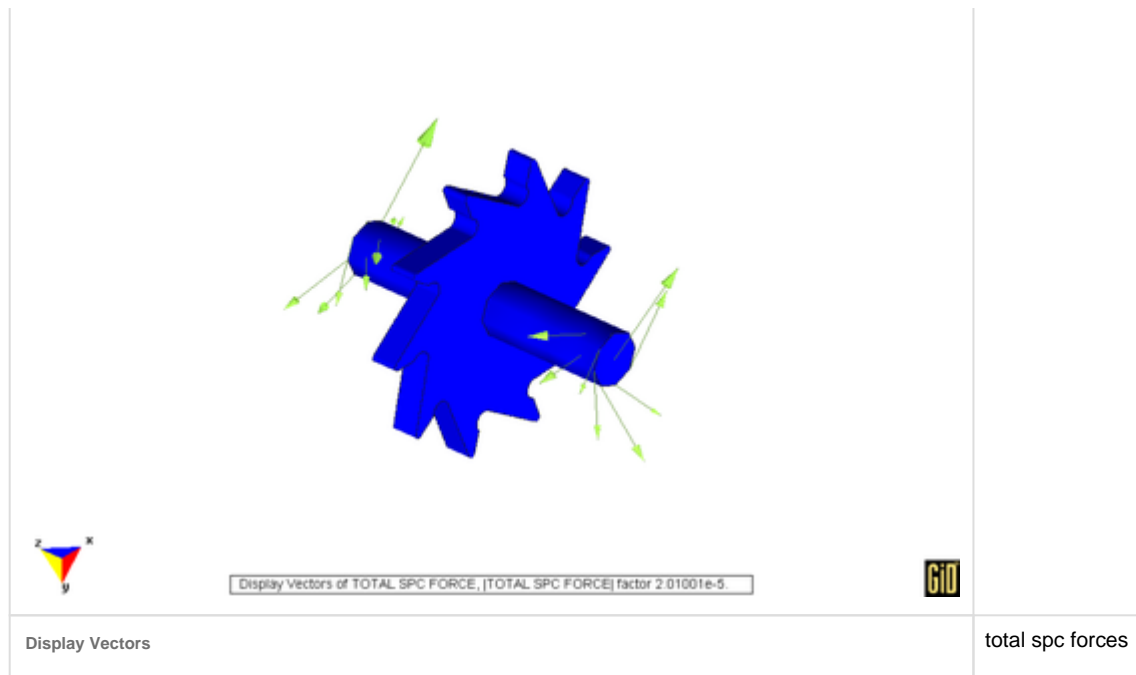
In View results window:  
Set View to *Contour Fill*.  
Set Results to *total translation*  
Set Component to *total translation modulus*.  
Click on **Apply** button.



#### Display Vectors

In View results window:  
Set View to *Display Vectors*.  
Set Results to *total spc forces*  
Set Component to *total spc forces modulus*.  
Click on **Apply** button.

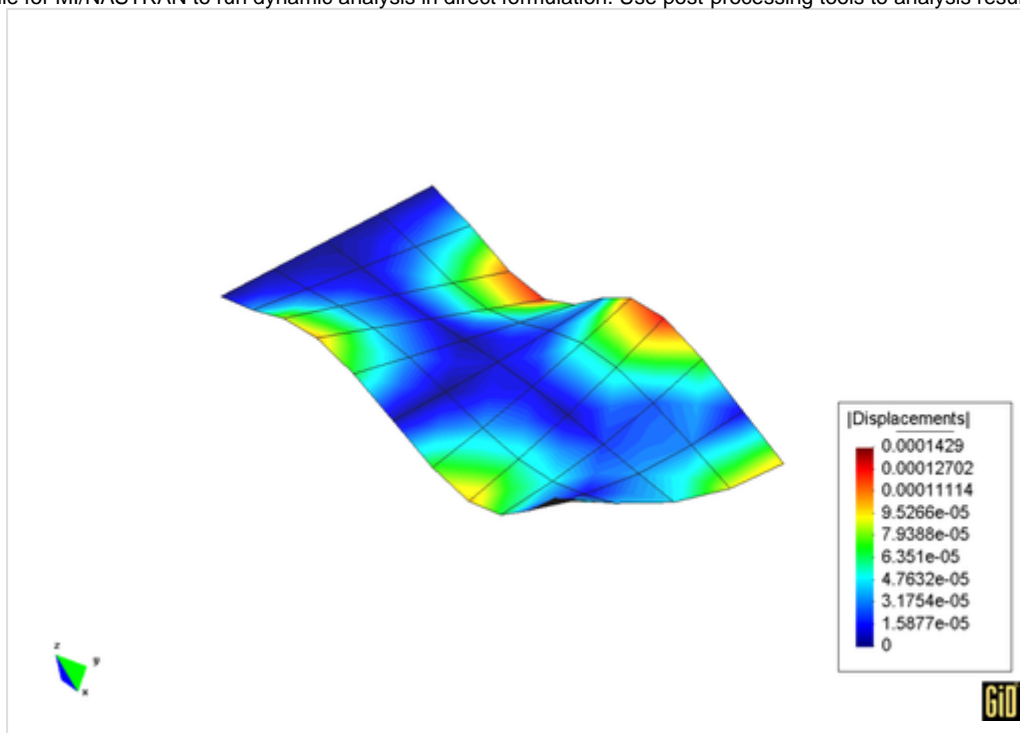




Of course there are more results and more possible visualizations, user is invited to explore all possibilities of GiD.

## Direct Frequency Response Analysis

Create geometry to represent a flat rectangular plate. Create a finite elements model of plate. Use a dynamic frequency dependent load  
Write an input file for MI/NASTRAN to run dynamic analysis in direct formulation. Use post-processing tools to analysis results.



### Purpose:

Create geometry to represent a flat rectangular plate.  
Create a finite elements model of plate.  
Use a dynamic frequency dependent load  
Write an input file for MI/NASTRAN to run dynamic analysis in direct formulation.  
Use post-processing tools to analysis results.

### Description of Model:

Determine the frequency response of a 5x2 inch flat plate under frequency-varying excitation by using the direct formulation.  
A unit load at a corner of the tip will excite the plate model.  
A frequency step of 20Hz will be used for this analysis within the range of 20 and 100Hz.  
Structural dumping will be set to 0,06.

### Properties:

Length	5 inch
Weight	2 inch
Thickness	0.1 inch
Mass Density	0.282 lbs/inch <sup>3</sup>
Young's Modulus	30.0e+6 lbs/inch <sup>2</sup>
Poisson's Ratio	0.3

### Procedure:

### Start:



- 1- Open a new project, right click on this icon in taskbar
- 2- Load problem type Nastran. Follow the menu sequence below:  
Data -> Problem type -> nastran  
Appears a splash image and the name of the master window changes to NASTRAN Interface.
- 3- Change the view plane to XY:  
View -> Rotate -> Plane XY (original)

#### Create geometry:

1. Create contour lines of the plate.

Geometry -> Create -> Line

Now insert in command line the coordinates of points to define the plate. Follow the order is very important.

First point -> 0,0

Second point -> 5,0

Third point -> 5,2

Fourth point -> 0,2

Fifth point -> 0,0

(Only it is necessary introduce points using two coordinates, the third coordinate Z is assume to 0 ).

Now you are asked if you want to join the first point and

the fifth point , select **Join**

Press escape or middle button of mouse.


1. Create surface.

Geometry -> Create -> NURBS Surface -> By contour



Or a more quick option is click on the following icon:

Situated in right vertical tool bar.

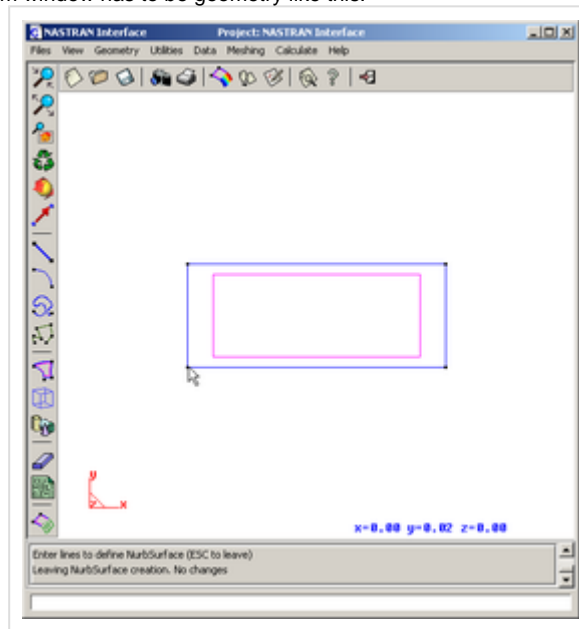
Now the pointer of mouse change to this form 

It means that now is possible select the four lines created before to create the surface.

Select the four lines in the screen.

Press escape or middle button of mouse.

Now in the NASTRAN Interface program window has to be geometry like this:



#### Assign property and material:

1. Define a new material.

Data -> Materials



Click on the following icon to create a new material:

Enter name for the new material **mat\_1**

Fill all statements like in the picture:

Material

mat 1

For default values write "DEFAULT" to the desired statement

Young (Ex): 30.0e6

Shear Modul: DEFAULT

Poisson (NUXY): 0.3

Mass Density: 0.282

Expansion-Coeff,a: 0.0

Temp Ref: 0.0

Limit Stress Tension: 0.0

Limit Stress Compression: 0.0

Limit Stress Shear: 0.0

Assign Draw Unassign Import/Export

Close



Click on the following icon to save the new material:

2-Define and assign the property for the surface.

Data -> Properties -> Property

If you don't close material window is possible get the property window clicking on the following icon



and select option "property" in the menu.

Select from the top combo box the property *membrane*.



Click on the following icon to create a new property:

Enter name for the new property **property\_1**.

Fill all statements like in the picture:

Property

property 1

PROPERTY MEMBRANE

Thicknesses,Tagv or T1: 0.1

Nonstructural mass/area: 0.0

Composition Material: mat 1

Assign Draw Unassign Import/Export

Close

Look that in Composition Material there is selected the material created before.

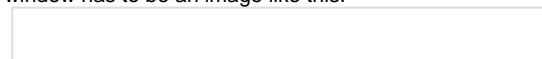


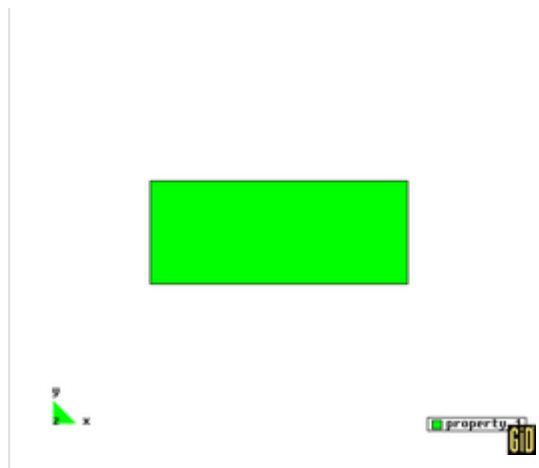
Click on the following icon to save the new property:

Now click on **Assign** button, choose Surfaces and select the surface of the geometry.

To make a proof if the property is well assigned click on **Draw** button, select *This property* option.

Now in the NASTRAN Interface program window has to be an image like this:





### Assign Constraints:

1. Assign prescribed displacements and rotation.

Data-> Boundary Conditions -> Constraints

Click on the following icon to set the condition over lines:



Uncheck Z-Rotation and leave all other statements check.

Now click on **Assign** button and select the vertical line on the left side (line number 4).

*Note: It is possible label geometry entities using this option:*

-Press right button mouse to get the contextual menu.

-Select option *Label* and choose *All*.

### Assign Dynamic Load

1. Define the table of interpolation values associate to dynamic load.

Data -> Loads -> Tables

Click on the following icon to create a new table:



Enter name for the new table **freq\_var**.

Data Entry

- Check Single Value

- X= 0.0 and Y= 1.0

- Click on **Add** button

- X=1000 and Y= 1.0

- Click on **Add** button

- Click on the following icon to save the new table:



- Click on **Close** button.

## 1. Assign dynamic load.

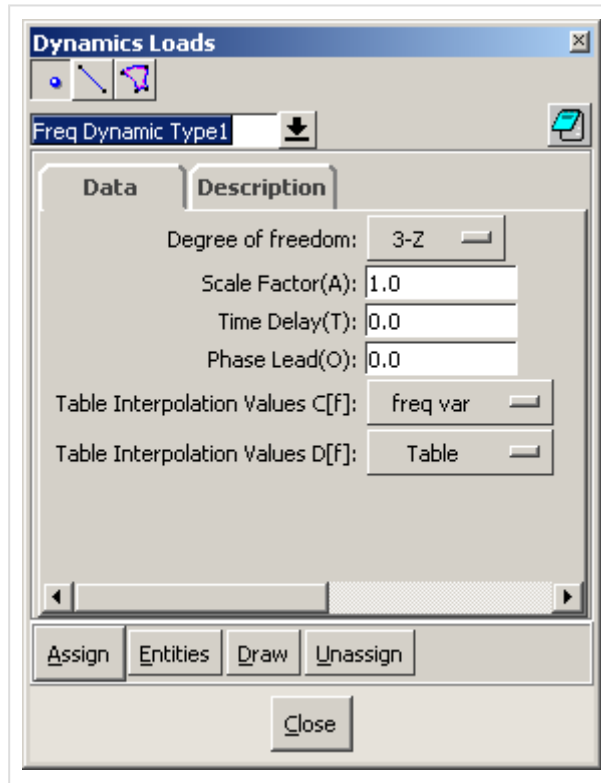
Data -&gt; Loads -&gt; Dynamics Loads



Click on the following icon to set the condition over points:

Select from the top combo box dynamic load *Freq Dynamic Type1*.

Fill all statements like in the picture:



Look that for Table Interpolation Values D[f] has value *Table*. *Table* is the default value for this statement and is equal to a blank table, it means that D[f] is equal to zero function.

Now click on Assign button and select the right bottom point (point number 2).

Click **Close** button.

**Mesh the Geometry:**

1-Create a structured mesh.

Meshing -&gt; Structured -&gt; Surfaces

Select the surface of the geometry.

Press escape or middle button mouse.

Now you are asked for number of cells to assign to lines, set to 10.

Click **OK** button.

Select lines 1 and 3, the large ones.

Press escape or middle button mouse.

Now you are asked again for number of cells to assign to lines, set to 4.

Click **OK** button.

Select lines 2 and 4, the short ones.

Press escape or middle button mouse.

Click **Cancel** button.

## 1. Generate the structured mesh.

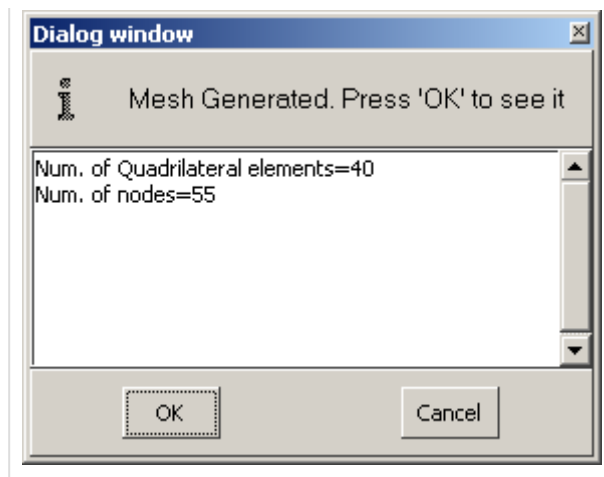
Meshing -&gt; Generate

Now you are asked about the size of elements to be generate, leave default value (0.54)

Click **OK** button

Appears a window with information about the mesh:





Num. of Quadrilateral elements = 40

Num. of nodes = 55

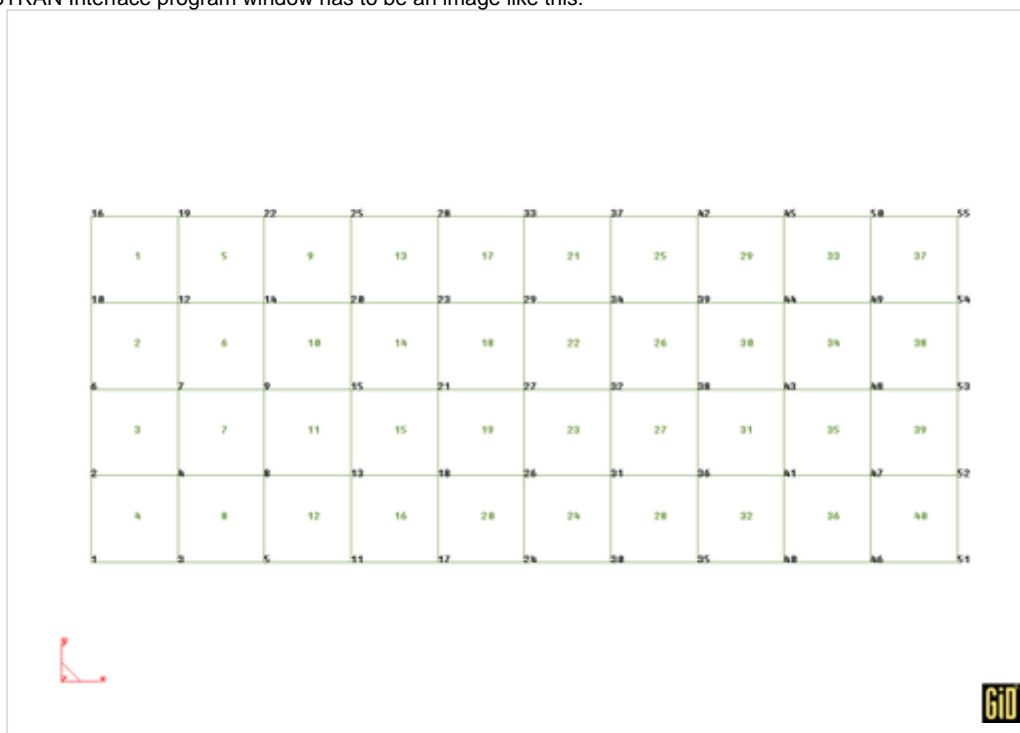
Click **OK** button.

*Note: It is possible label mesh elements and nodes using this option:*

*-Press right button mouse to get the contextual menu.*

*-Select option Label and choose All.*

Now in the NASTRAN Interface program window has to be an image like this:



### Perform the Analysis:

#### 1. Design Executive Control Section.

Data -> Problem Data -> Executive Control

Select type of NASTRAN will be use in the analysis.

Check STATICS and DIRECT FREQUENCY RESPONSE and leave all the other statements unchecked.

Leave the rest of statements with the default values.

Click **Accept data** button.

#### 2 - Design Case Control Section.

Data -> Problem Data -> Case Control

2.1.- Input data

Leave all statements with default values.

2.2.- Output data



Set Title to **"Direct\_frequency\_Response"**

Leave Subtitle, Label ... and Post process with default values.

Check Displacements and Velocity, and uncheck the rest of output requests.

In Output Design section leave the default values.

*Note: If the user wants to post processing the results of the analysis with MI/NASTRAN Interface is mandatory that the output device has to set to PUNCH.*

Click **Accept data** button

1. Design Dynamic Analysis.

Data -> Problem Data -> Dynamics

Set these values to the different statements:

Solution Method = Direct

Domain of Solution = Frequency

Overall Structural Damping Coeff. = 0.06

Frequency Step

Initial step = 20

Frequency Increment = 20

Number of frequency increments = 49

Mass formulation = Coupled

Click **Accept data** button

1. Set PARAM values.

Data -> Problem Data -> PARAM

In MI/NASTRAN tab set WTMASS to 0.00259

Leave the rest of statements with default values.

Click **Accept data** button.

#### Obtain Input File for NASTRAN Code:

File -> Import/Export -> Write Calculation File

Appears a window to select in which folder and the name for save the file. It is very important write extension of the file in the name.

Post process:



Click on the following icon to enter in the post process:

#### Import punch file:

1. Import the punch file (\*\*\*.pch) obtained from NASTRAN.

File -> Import -> Import PUNCH

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

After importing process is finished close the window.

#### Import FEMAP ASCII neutral file:

1. 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

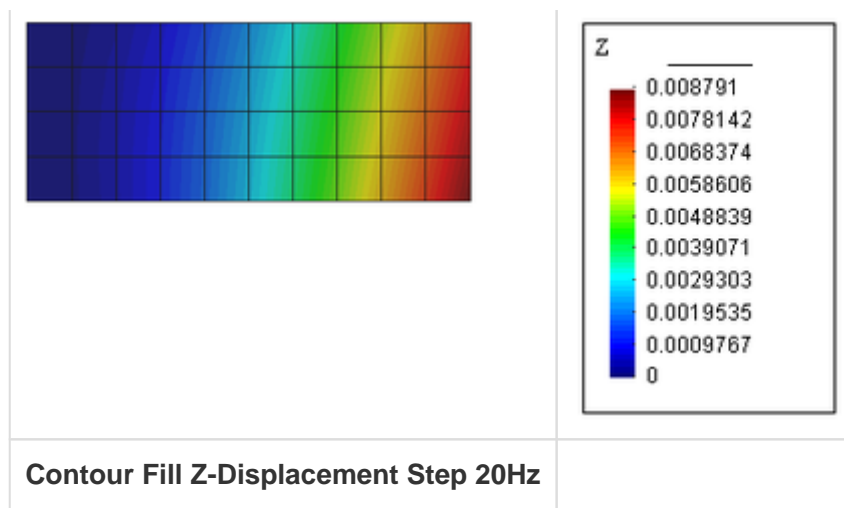
Selects RSLTFILETYPE and set to FEMAP ASCII in Output Control Directives

#### Visualization of results:

1. Contour Fill.

View results -> Contour Fill

Select one of the results displacements or velocity.



To change step of analysis use this menu sequence:  
View results -> Default Analysis/Step -> 1  
Select that step you want to see.

1. Deformation.

View results -> Deformation  
Select one of the results displacements or velocity.

Use trackball rotation to rotate the plate and see better the deformation.



#### Numerical results:



To see numerical results in nodes, click on this icon  
and select in which nodes you want to see the current numerical results display in the screen.  
For example:  
Node 51 step 200Hz  
Displacements

X= 1.28075e-19	Y= -4.89892e-20	Z= -0.00547175	<b>Dis.</b>	=0.0547175
----------------	-----------------	----------------	-------------	------------

Velocity

X= -1.23013e-19	Y= -5.22769e-19	Z= 0.467011	<b>Vel.</b>	= 0.467011
-----------------	-----------------	-------------	-------------	------------

#### Create an Animation:

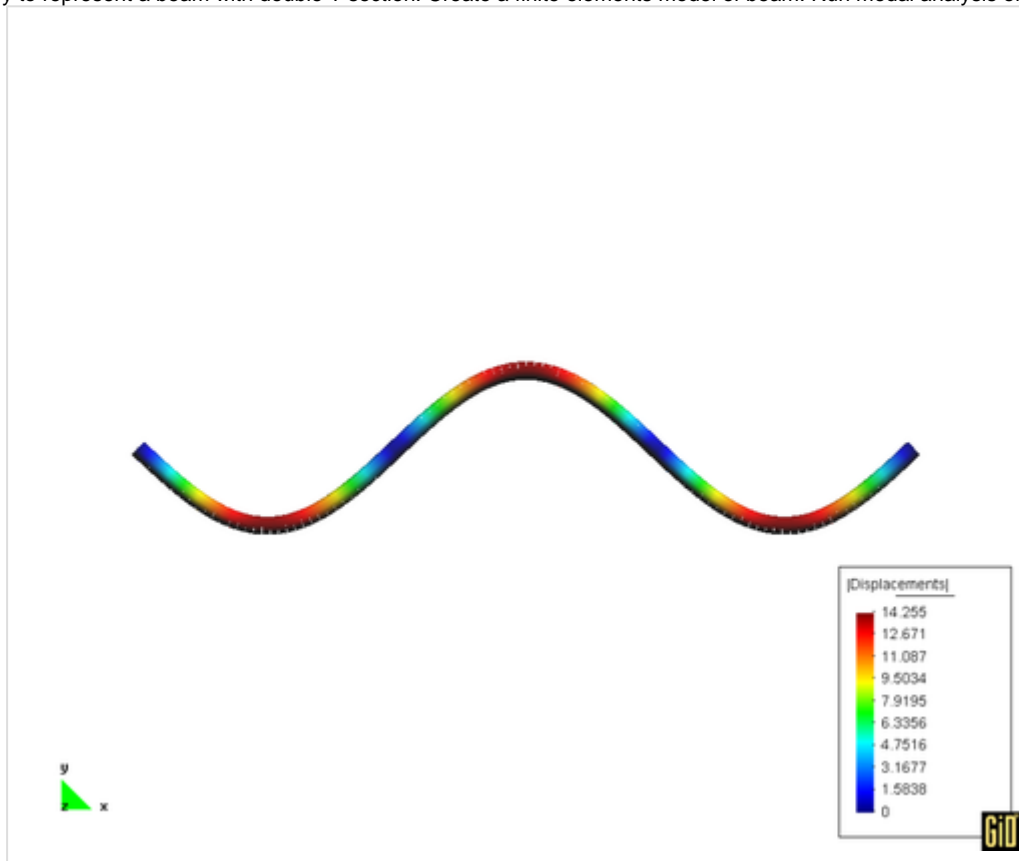
1. Select which kind of results will be displayed
2. Open Animation Window.

Windows -> Animate

1. Click **Play** button.

## Modal Analysis of a Beam

Create geometry to represent a beam with double T section. Create a finite elements model of beam. Run modal analysis of the beam.



### Purpose:

Create geometry to represent a beam with double T section.  
Create a finite elements model of beam.  
Run modal analysis of the beam.

### Description of Model:

Determine the first three modes of a beam with double T section, using Givens method for eigenvalues extraction.

### Properties:

Length	100 inch
Weight (Section)	2 inch
Width (Section)	1 inch
Thickness (Section)	0.1 inch
$I_1$	0.229 inch <sup>4</sup>
$I_2$	0.017 inch <sup>4</sup>

### Procedure:

### Start:



- 1- Open a new project, right click on this icon in taskbar
- 2- Load problem type Nastran. Follow the menu sequence below:  
Data -> Problem type -> nastran  
Appears a splash image and the name of the master window changes to NASTRAN Interface.
- 3- Change the view plane to XY:  
View -> Rotate -> Plane XY (original)

#### Create geometry:

1. a. Create line.

Geometry -> Create -> Line

Now insert in command line the coordinates of points to define the plate. Follow the order is very important.

First point -> 0,0

Second point -> 100,0

(Only it is necessary introduce points using two coordinates, the third coordinate Z is assume to 0 ).

Press escape or middle button of mouse.

#### Assign property and material:

1. a. Define a new material.

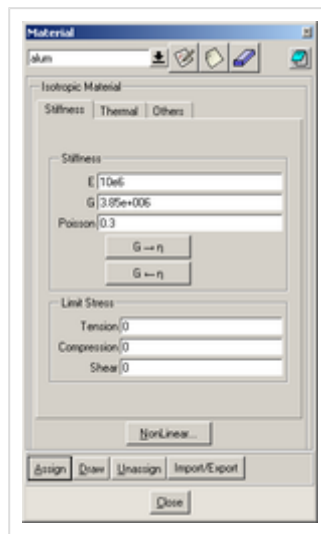
Data -> Materials



Click on the following icon to create a new material:

Enter name for the new material **alum**.

Fill all statements like in the picture:



And set density, in Others page, to 0.101



Click on the following icon to save the new material:

2-Define and assign the property for the surface.

Data -> Properties -> Property



If you don't close material window is possible get the property window clicking on the following icon and select option "property" in the menu.

Select from the top combo boxes the property "beam".

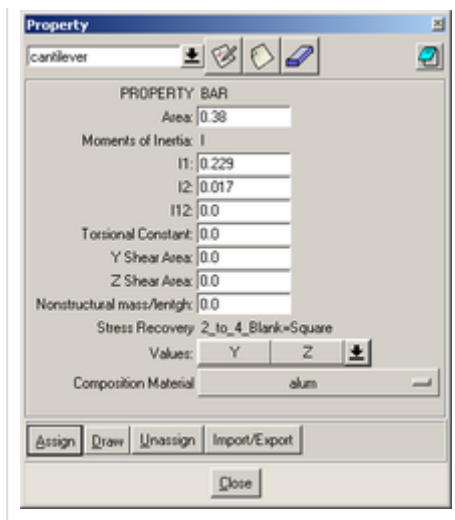


Click on the following icon to create a new property:

Enter name for the new property **cantilever**.

Fill all statements like in the picture:





You have to select the previously created material in Composition Material.



Click on the following icon to save the new property:

Now click on Assign button, select the line.

To make a proof if the property is well assigned click on **Draw** button, select *This property* option.

Now in the NASTRAN Interface program window has to be an image like this:

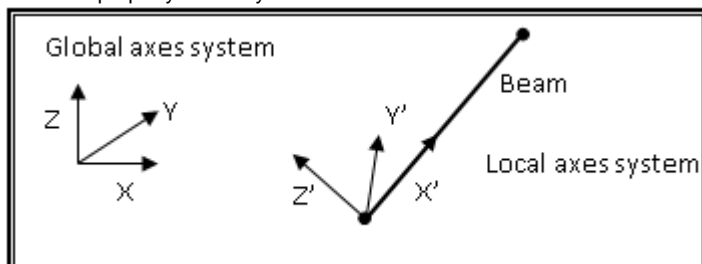


### Define Local Axes

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' axe 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.

1. **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.
2. **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.
3. **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.

Assign Local Axes.

Data -> Properties -> Local Axes

Set statement Local Axes to Automatic (An automatic local will defined for all lines selected).

Select the line of the geometry.

### Mesh the Geometry:

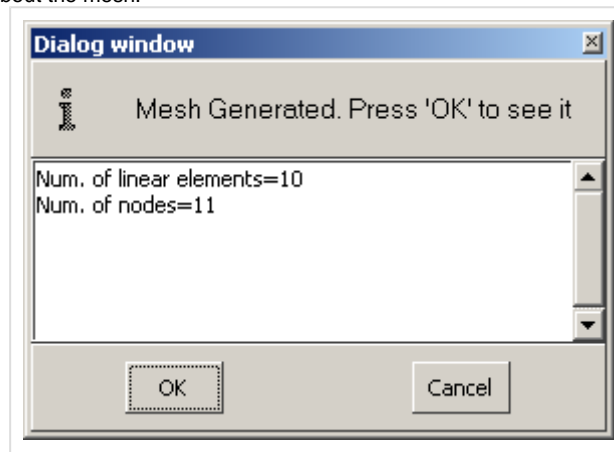
1-Create a mesh.

Meshing -> Generate

Now you are asked about the size of elements to be generate, leave default value (10)

Click on OK button

Appears a window with information about the mesh:



Num. of linear elements = 10

Num. of nodes = 11

Click on **OK** button.

*Note: It is possible label mesh elements and nodes using this option:*


-Press right button mouse to get the contextual menu.

-Select option "Label" and choose "All".

### Assign Constraints:

1- Assign prescribed displacements and rotation.

Data-> Boundary Conditions -> Constraints

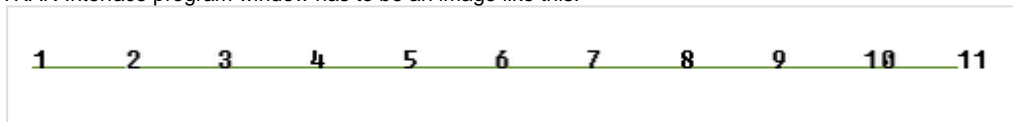
Click on the following icon to set the condition over points: 

Show the labels of nodes using this option:

-Press right button mouse to get the contextual menu.

-Select option "Label" and choose "All in" and choose points.

Now in the NASTRAN Interface program window has to be an image like this:



Node 1

Check the following items:

X-Displacement

Y-Displacement

Z-Displacement

X-Rotation

Y-Rotation

Uncheck Z-Rotation.

**Node 11**

Check the following items:

Y-Displacement

Z-Displacement

X-Rotation

Y-Rotation

Uncheck X-Displacement and Z-Rotation.

Rest of nodes (except node 1 and node 11)

Check the following items:

Z-Displacement

X-Rotation

Y-Rotation

Uncheck

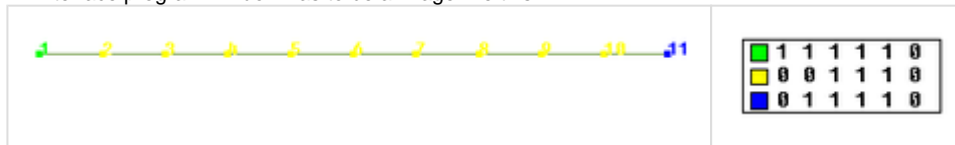
X-Displacement

Y-Displacement

Z-Rotation.

To make a proof if the constraints are well assigned click on **Draw** button, select "Colors" option.

Now in the NASTRAN Interface program window has to be a image like this:

**Perform the Analysis:**

## 1. Design Executive Control Section.

Data -> Problem Data -> Executive Control

Select type of NASTRAN will be use in the analysis.

Check MODES and leave all the other statements uncheck.

Leave the rest of statements with the default values.

Click on **Accept data** button.

## 2 - Design Case Control Section.

Data -> Problem Data -> Case Control

## 2.1.- Input data

Leave all statements with default values.

## 2.2.- Output data

Set Title to **"Modes\_Analysis"**

Leave Subtitle, Label ... and Post process with default values.

Check Displacements uncheck the rest of output requests.

In Output Design section leave the default values.

*Note: If the user wants to post processing the results of the analysis with MI/NASTRAN Interface is mandatory that the output device has to set to PUNCH.*

Click on **Accept data** button.

## 3- Design Modes Extraction.

Data -> Problem Data -> Dynamics

In modes analysis tab:

Set the following values to the different statements:

-Method of eigenvalues extraction = GIV.

-First Frequency = 0.0

-Last Frequency = 350.0

-Desired number = 3

-Check Mass orthogonality test option.

-Leave the default values for the rest of statements.

In Dynamic Design tab:  
Set Mass formulation = Coupled  
Click on **Accept data** button

1. Set PARAM values.

Data -> Problem Data -> PARAM  
In MI/NASTRAN tab set WTMASS to 0.00259  
Leave the rest of statements with default values.  
Click on **Accept data** button.

#### Obtain Input File for NASTRAN Code:

File -> Import/Export -> Write Calculation File  
Appears a window to select in which folder and the name for save the file. It is very important write extension of the file in

Post process:



Click on the following icon to enter in the post process:

#### Import punch file:

1- Import the punch file (\*.pch) obtained from NASTRAN.  
File -> Import -> Import PUNCH  
Appears a window to select the punch file located in NASTRAN results folder.  
After importing process is finished close the window.

#### Import FEMAP ASCII neutral file:

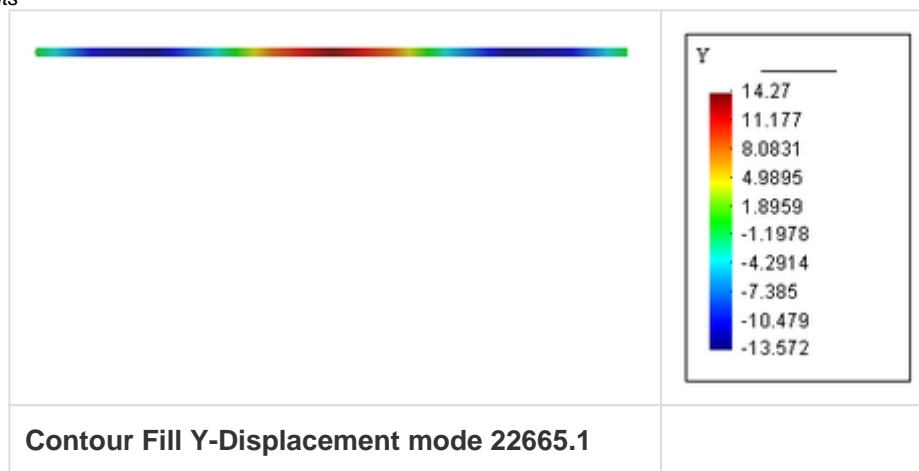
1. Import the FEMAP ASCII file (\*.neu) obtained from NE/NASTRAN.

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

#### Visualization of results:

1. Contour Fill.

View results -> Contour Fill  
Select "*Displacements*"





To change step of analysis use this menu sequence:

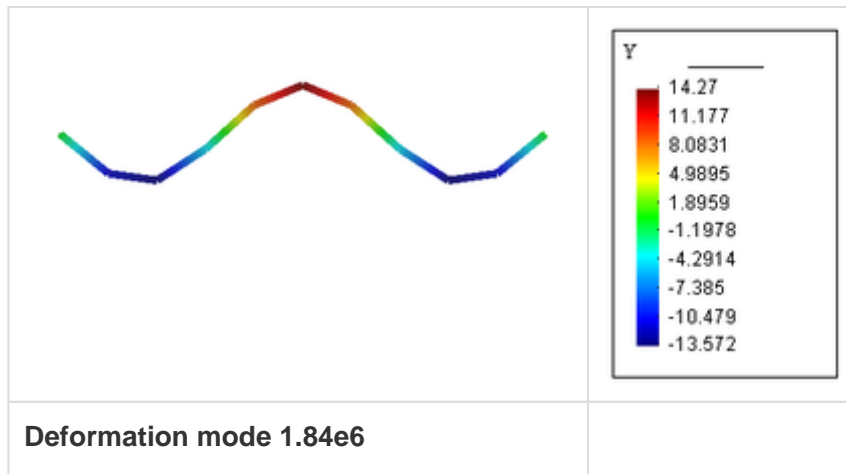
View results -> Default Analysis/Step -> 1

Select that step you want to see.

1. Deformation.

View results -> Deformation

Select displacements.



#### Numerical results:



To see numerical results in nodes, click on the following icon

Select in which nodes you want to see the current numerical results display in the screen.

#### Create an Animation:

1. Select which kind of results will be displayed
2. Open Animation Window.

Windows -> Animate

1. Click on Play button.

#### Modes of beam

It is necessary edit the results file **\*\*\*.F06** located in results folder of NASTRAN.

Here are the results with (MI/NASTRAN):

Modes no.	EIGENVALUE	CYCLIC FREQUENCY
1	2.266509E+04	<b>2.396066E+01</b>
2	3.627142E+05	<b>9.585227E+01</b>
3	1.837810E+06	<b>2.157598E+02</b>