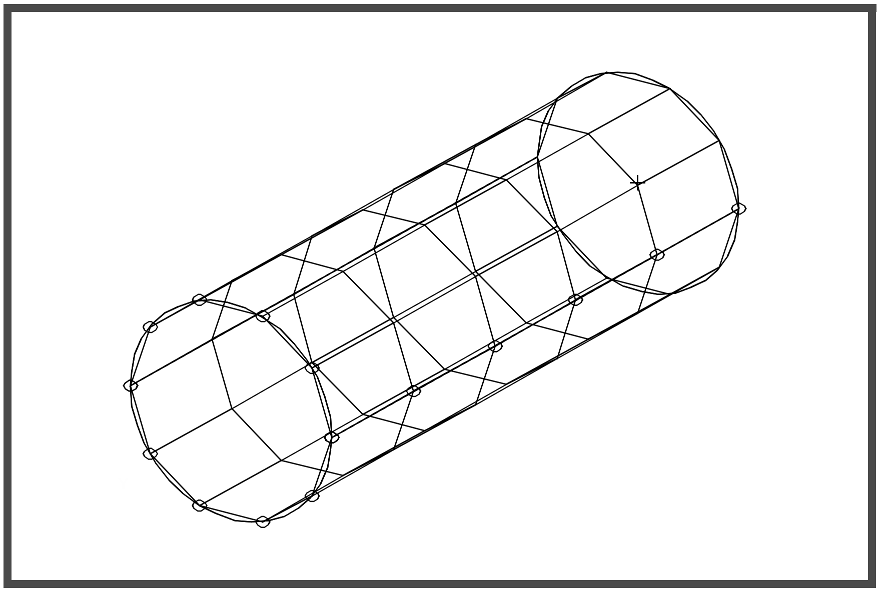
WORKSHOP 4

Rigid Element Analysis with RBAR



Objectives:

- Idealize the tube with QUAD4 elements.
- Use RBAR elements to model a “rigid” end.
- Produce a Nastran input file that represents the cylinder.
- Submit the file for analysis in MSC.Natran.
- Find the displacement vectors.

Model Description:

The goal of this example is to maintain a circular cross section at the rigid end of the tube, (using RBAR elements), while applying a torque of 6000 in·lbs about the z-axis of the tube.

Below is a Finite Element representation of the tube. One end of the tube is considered rigid, and the other end is fixed in all translational and rotational degrees of freedom. Table 4.1 contains all the necessary parameters to construct the input file.

Figure 4.1 - Grid Coordinates and Element Connectivities

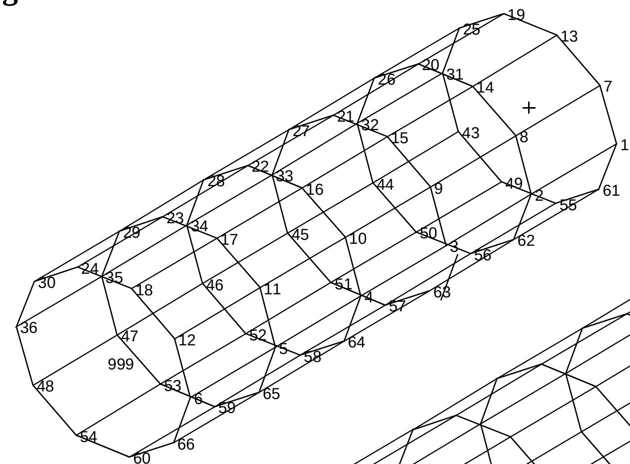


Figure 4.2 - Loads and Boundary Conditions

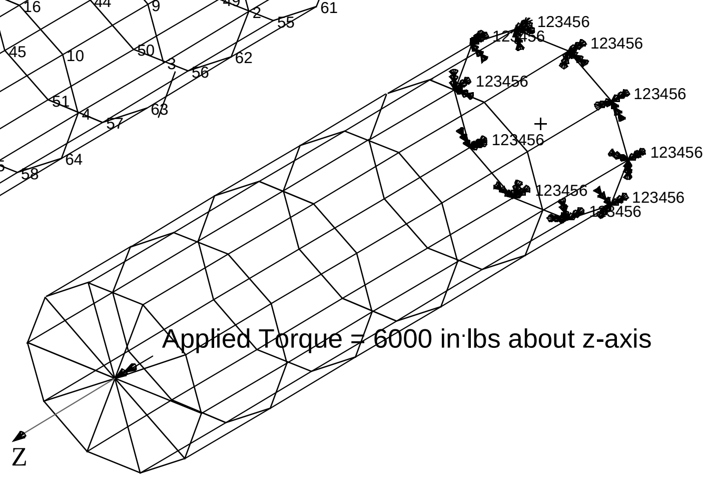


Table 4.1 - Model Properties

Radius:	**15 in**
Thickness:	**0.125 in**
Length:	**90 in**
Elastic Modulus:	**10E+06 lb/in^2**
Poisson Ratio:	**0.3**
Density:	**0.101 lbs/in^3**
Moment:	**6000 in-lbs**

Suggested Exercise Steps:

- Generate a finite element representation of the cylinder structure. (i.e., The nodes (GRID) and element connectivities (CQUAD4) should be defined manually.)
- Define material (MAT1) and element (PSHELL) properties.
- Create grid point 999 at the center of the rigid end. This point is to serve as the load application point, as well as the connection point for the rigid element.
- Idealize the rigid end with rigid elements (RBAR).
- Apply the fixed boundary constraints (SPC1).
- Apply a static concentrated moment at the center of the top enclosure, grid 999 (MOMENT).
- Prepare the model for linear static analysis (SOL 101).
- Generate an Input file and submit it to the MSC.Natran solver for linear static analysis.
- Review the results, specifically the displacements along the top edges.

```
ID SEMINAR,PROB4
```

```
CEND
```

```
BEGIN BULK
```

1	2	3	4	5	6	7	8	9	10

1	2	3	4	5	6	7	8	9	10

ENDDATA

Exercise Procedure:

1. Users who are not utilizing MSC.Patran for generating an input file should go to Step 15, otherwise, proceed to Step 2.

2. Create a new database called **prob4.db**.

File/New...

New Database Name	**prob4**
OK	

In the New Model Preference form set the following:

Tolerance	◆ **Default**
Analysis code:	**MSC/NASTRAN**
OK	

In the next few steps you will create the necessary geometry for the cylinder model.

Whenever possible click ❑ **Auto Execute** (turn off).

3. First create a cylindrical coordinate frame.

◆ **Geometry**

Action:	**Create**
Object:	**Coord**
Method:	**3Point**
Coord ID List:	**1**
Type:	**Cylindrical**
Origin:	**[0,0,0]**
Point on Axis 3	**[0,0,1]**
Point on Plane 1-3:	**[1,0,0]**
Apply	

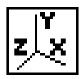
4. Now create a curve.

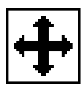
◆ **Geometry**

Action:	**Create**
Object:	**Curve**
Method:	**XYZ**
Vector Coordinate List:	**<0, 0, 90>**
Origin Coordinate List:	**[15, 0, 0]**

Apply

Change the view to isoview_1 by selecting this icon:

 **Iso 1 View**

 **Fit View**

5. Create a surface out of the curve you just made.

◆ **Geometry**

Action:	**Create**
Object:	**Surface**
Method:	**Revolve**
Axis:	**Coord 0.3**
Total Angle:	**180**
Curve List:	**Curve 1**

Apply

Repeat this step a second time to complete the cylinder. This time select the curve *opposite* **Curve 1** which is **Surface 1.2**.

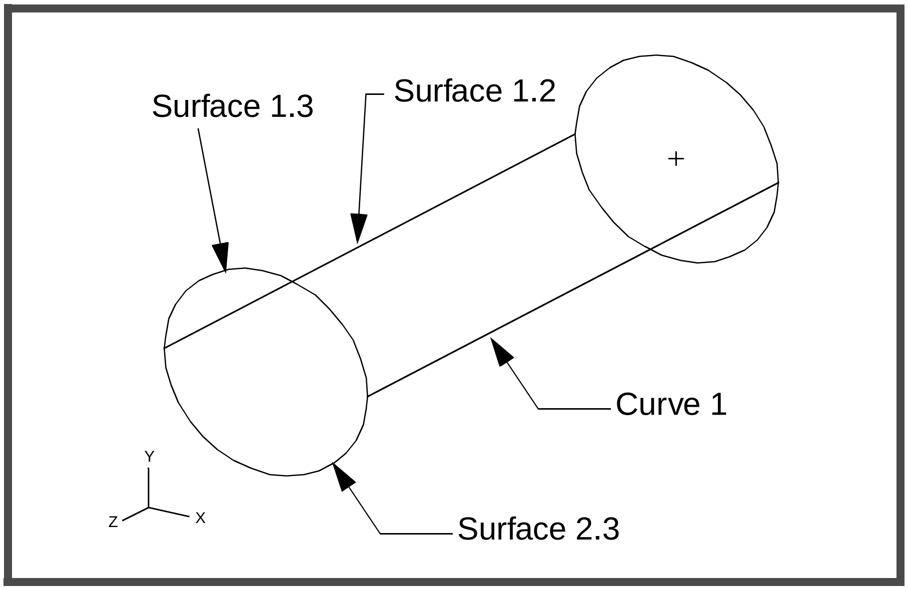
◆ **Geometry**

Action:	**Create**
Object:	**Surface**

Method:	**Revolve**
Axis:	**Coord 0.3**
Total Angle:	**180**
Curve List:	**Surface 1.2**

Apply

Figure 4.3 - Geometry of Rigid Element



6. Create the Finite Element Model and mesh the surface.

   First you will create 5 mesh seeds along the top and bottom edges (Surface 1.3 and 2.3), and 5 along the Curve 1.

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh Seed**
Type:	**Uniform**

◆ **Number of Elements**

Number =	**5**
Curve List:	**Curve 1, Surface 1.3, 2.3**

Apply

Mesh the surface.

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Surface**
Node Coordinate Frame...	
Analysis Coordinate Frame:	**Coord 1**
Refer. Coordinate Frame:	**Coord 1**
OK	
Surface List:	**Surface 1, 2**
Apply	

Equivalence the model to remove duplicate nodes at common geometry boundaries.

◆ **Finite Elements**

Action:	**Equivalence**
Object:	**All**
Method:	**Tolerance Cube**
Apply	

7. Now you will create the Material Properties for the plate.

◆ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name:	**alum**
Input Properties...	
Elastic Modulus =	**10.0E6**

Poisson Ratio = **0.3**

OK

Apply

8. Give the cylinder a thickness using **Properties**.

◆ **Properties**

Action: **Create**

Dimension: **2D**

Type: **Shell**

Property Set Name: **wall**

Input Properties...

Material Name: **m:alum**

Thickness: **0.125**

OK

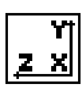
Select Members: **Surface 1, 2**

Add

Apply

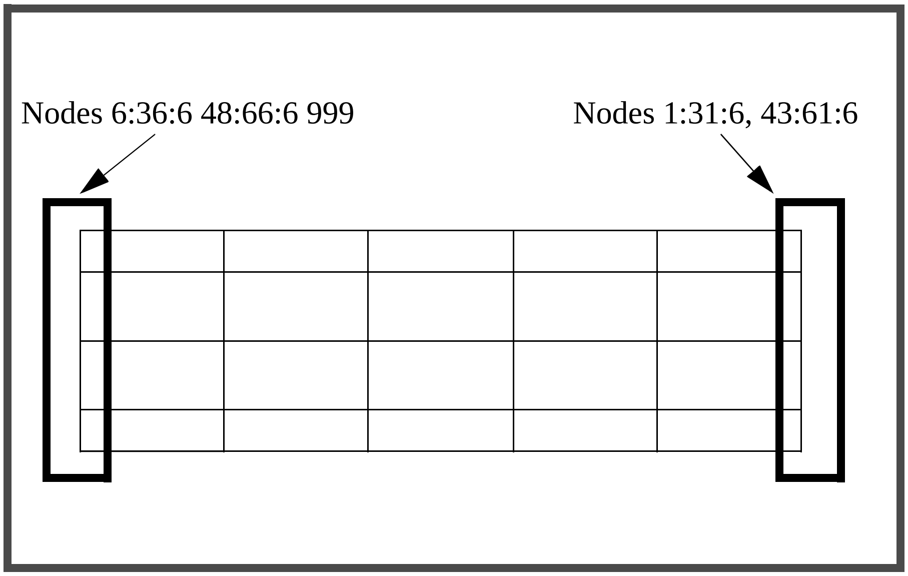
Next you will apply the Load and Boundary Conditions to the model.

9. Fix the base from moving in all Degrees of Freedom. To make the screen selection easier change the view to Right View by selecting this icon:



Right Side View

Figure 4.4 - Node Locations



◆ **Loads/BCs**

Action:	**Create**
Object:	**Displacement**
Type:	**Nodal**
New Set Name:	**fixed**

Input Data...

Translations <T1 T2 T3>	**<0, 0, 0>**
Rotations <R1 R2 R3>	**<0, 0, 0>**
Analysis Coordinate Frame:	**Coord 1**

OK

Select Application Region...

Geometry Filter:	◆ **FEM**
Select Nodes:	**Node 1:31:6, 43:61:6**

(see Fig 4.4)

Add

OK

Apply

10. Next you must create the loading point where the moment will be applied.

◆ **Finite Elements**

Action:	**Create**
Object:	**Node**
Method:	**Edit**
Node ID List:	**999**
Analysis Coordinate Frame:	**Coord 1**
Refer. Coordinate Frame:	**Coord 1**

❑ *Associate with Geometry (turn off)*

Node Location List:	**[0, 0, 90]**

Apply

11. Create the rigid element.

◆ **Finite Elements**

Action:	**Create**
Object:	**MPC**
Type:	**RBAR**

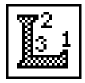
Define Terms...

◆ **Create Dependant**

Node List:	**Node 6**

You can type the node into the list directly or you can screen select it by changing back to **Isometric View1** and selecting the node on the *left edge* of the model.

Also, you may want to turn on node label as well.



Select DOFs by holding the Shift key down while clicking with the left mouse button.

DOFs: **UX**
UY
UZ

Apply

◆ **Create Independent**

Node List: **Node 999**

DOFs: **UX**
UY
UZ
RX
RY
RZ

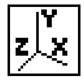
Apply

Cancel

Apply

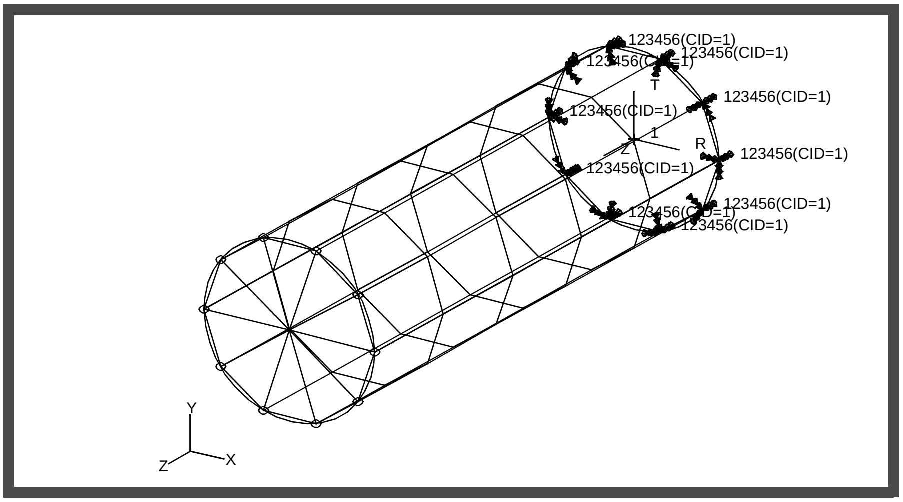
Repeat this procedure nine more times replacing the *Dependent Variable Node List* with **Nodes 12, 18, 24, 30, 36, 48, 54, 60, 66.** The DOFs and the independent node shall remain the same.

Change the view to isoview_1 by selecting this icon:



Iso 1 View

Figure 4.5 - Nodal Constraints on the Rigid Element



12. Now you will create the moment.

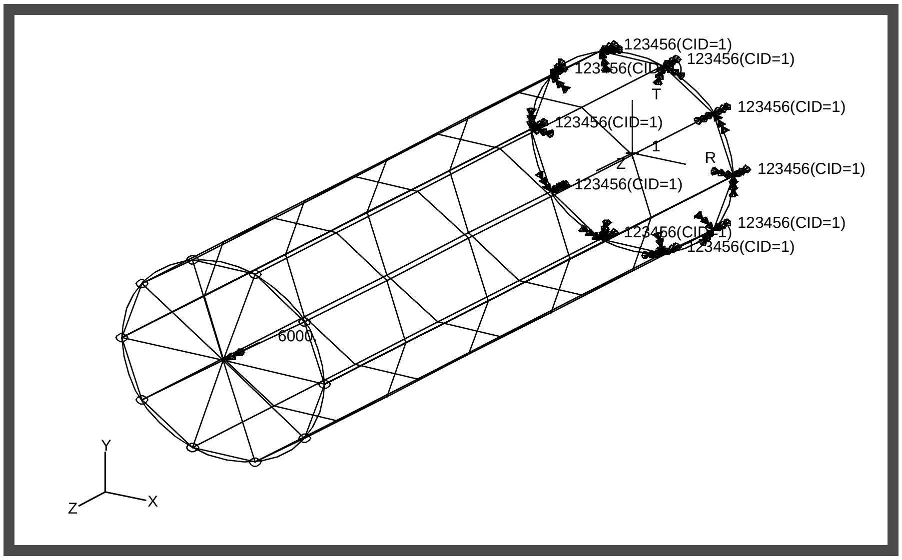
◆ **Loads/BCs**

Action:	**Create**
Object:	**Force**
Type:	**Nodal**
New Set Name:	**applied_moment**
Input Data...	
Force:	< >
Moment:	< , **, 6000**>
Analysis Coordinate Frame:	**Coord 0**
OK	
Select Application Region...	
Geometry Filter:	◆ **FEM**
Select Nodes:	**Node 999**
Add	

OK

Apply

Figure 4.6-Force and Displacement Constraints



13. Create a new group for output.

Group/Create...

New Group Name: **output_request**

Group Contents: **Add Entity Selection**

NOTE: The selection process for the nodes is the same as in *step 11*. Be certain to set the selection menu filter to nodes only.

Node

Also make sure you **exclude Node 999.**

Entity Selection: **Node 6:36:6, 48:66:6**

Apply

Cancel

14. Now you are ready to run the analysis.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Analysis Deck**
Job Name:	**prob4**
Translation Parameters...	
Data Output:	**XDB and Print**
OK	
Solution Type...	
Solution Type:	◆ **Linear Static**
OK	
Subcase Create...	
Subcase Name:	**output_1**
Available Load Cases:	**Default**
	(Highlight to Select)
Output Request...	
Form Type:	**Advanced**

Under Output Requests delete both “STRESS...” and “SPCFORCES...”, then highlight “DISPLACEMENT...”.

Select Group(s)/Set:	**output_request**
Modify	
OK	
Apply	
Cancel	
Subcase Select...	
Subcases For Solution Sequence:	**Output_1**

Subcases Selected: **Default**

(Click to deselect)

OK

Apply

An MSC.Natran input file called **prob4.bdf** will be generated. This process of translating your model into an input file is called the Forward Translation. The Forward Translation is complete when the Heartbeat turns green.

Generating an Input File for MSC.Natran Users:

MSC.Natran users can generate an input file using the data from Table 4.1. The result should be similar to the output below.

15. MSC.Natran Input File: **prob4.dat**

```
ID SEMINAR, PROB4
SOL 101
TIME 600
CEND
SET 1 = 6,12,18,24,30,36,48,54,60,66
SUBCASE 1
   SPC = 1
   LOAD = 1
   DISPLACEMENT = 1
BEGIN BULK
PARAM, POST, 0
PSHELL   1       1       .125    1               1
CQUAD4   1       1       1       2       8       7
=        *1      =       *1      *1      *1      *1
=3
CQUAD4   6       1       7       8       14      13
=        *1      =       *1      *1      *1      *1
=3
CQUAD4   11      1       13      14      20      19
=        *1      =       *1      *1      *1      *1
=3
CQUAD4   16      1       19      20      26      25
=        *1      =       *1      *1      *1      *1
=3
CQUAD4   21      1       25      26      32      31
=        *1      =       *1      *1      *1      *1
=3
CQUAD4   26      1       31      32      44      43
=        *1      =       *1      *1      *1      *1
=3
CQUAD4   31      1       43      44      50      49
=        *1      =       *1      *1      *1      *1
=3
CQUAD4   36      1       49      50      56      55
=        *1      =       *1      *1      *1      *1
=3
CQUAD4   41      1       55      56      62      61
=        *1      =       *1      *1      *1      *1
=3
CQUAD4   46      1       61      62      2       1
=        *1      =       *1      *1      *1      *1
=3
MAT1     1       10.+6           .3      .101
RBAR     51      6       999             123456  123
=        *1      *6      ==
=4
RBAR     57      48      999             123456  123
=        *1      *6      ==
```

```
=2
GRID      1       1       15.     0.      0.      1
=         *1      =       =       =       *18     =
=4
GRID      7       1       15.     36.     0.      1
=         *1      =       =       =       *18     =
=4
GRID      13      1       15.     72.     0.      1
=         *1      =       =       =       *18     =
=4
GRID      19      1       15.     108.    0.      1
=         *1      =       =       =       *18     =
=4
GRID      25      1       15.     144.    0.      1
=         *1      =       =       =       *18     =
=4
GRID      31      1       15.    -180.    0.      1
=         *1      =       =       =       *18     =
=4
GRID      43      1       15.    -144.    0.      1
=         *1      =       =       =       *18     =
=4
GRID      49      1       15.    -108.    0.      1
=         *1      =       =       =       *18     =
=4
GRID      55      1       15.    -72.     0.      1
=         *1      =       =       =       *18     =
=4
GRID      61      1       15.    -36.     0.      1
=         *1      =       =       =       *18     =
=4
GRID      999     1       0.      0.      90.     1
SPC1      1       123456  1       7       13      19      25      31      +       A
+       A 43      49      55      61
MOMENT    1       999     0       6000.   0.      0.      1.
CORD2C    1               0.      0.      0.      0.      0.      1.      +       B
+       B 1.      0.      0.
ENDDATA
```

Submitting the Input File for Analysis:

16. Submit the input file to MSC.Natran for analysis.

    16a. To submit the MSC.Patran **.bdf** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran prob4.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

    16b. To submit the MSC.Natran **.dat** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran prob4.dat scr=yes**. Monitor the run using the UNIX **ps** command.

17. When the run is completed, edit the **prob4.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing WARNING messages indicate modeling errors.

18. While still editing **prob4.f06**, search for the word:

D I S P L A C E M E N T (spaces are necessary)

T2 (GRID 6) = ____________________

T2 (GRID 30) = ____________________

T2 (GRID 60) = ____________________

Comparison of Results:

19. Compare the results obtained in the **.f06** file with the results on the following page:

D I S P L A C E M E N T   V E C T O R

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
6	G	2.334818E-18	8.929722E-04	-3.036160E-20	-1.846752E-05	-3.557538E-19	5.934382E-05
12	G	-7.439477E-20	8.929722E-04	4.799947E-19	-1.846752E-05	2.286989E-19	5.934382E-05
18	G	-2.455192E-18	8.929722E-04	9.146586E-19	-1.846752E-05	-3.967632E-20	5.934382E-05
24	G	-3.898189E-18	8.929722E-04	1.107603E-18	-1.846752E-05	-2.191029E-19	5.934382E-05
30	G	-3.852210E-18	8.929722E-04	9.851302E-19	-1.846752E-05	8.131516E-20	5.934382E-05
36	G	-2.334818E-18	8.929722E-04	5.940202E-19	-1.846752E-05	-3.388132E-19	5.934382E-05
48	G	7.439477E-20	8.929722E-04	8.366394E-20	-1.846752E-05	5.234664E-19	5.934382E-05
54	G	2.455192E-18	8.929722E-04	-3.509999E-19	-1.846752E-05	-7.157428E-20	5.934382E-05
60	G	3.898189E-18	8.929722E-04	-5.439445E-19	-1.846752E-05	-1.355253E-20	5.934382E-05
66	G	3.852210E-18	8.929722E-04	-4.214716E-19	-1.846752E-05	1.524659E-19	5.934382E-05

**20. MSC.Natran Users have finished this exercise.
MSC.Patran Users should proceed to the next step.**

21. Proceed with the Reverse Translation process, that is attaching the **prob4.xdb** results file into MSC.Patran. To do this, return to the Analysis form and proceed as follows:

◆ **Analysis**

Action:	**Attach XDB**
Object:	**Result Entities**
Method:	**Local**
Select Results File...	
Selected Results File:	**prob4.xdb**
OK	
Apply	

Set the **default_group** to current.

Group/Set Current...

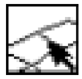
Set Current Group:	**default_group**
Cancel	

22. When the translation is complete bring up the **Results** form..

◆ **Results**

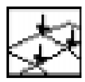
Action:	**Create**
Object:	**Deformation**
Select Result Case(s):	**Default, A1:Static Subcase**
Select Deformation Result:	**Displacements, Translational**
Show As:	**Resultant**

To change the target entities of the plot, click on the **Target Entities** icon.

 **Target Entities**

Target Entity:	**Groups**
Select Groups:	**default_group**

To change the display attributes of the plot, click on the **Display Attributes** icon.

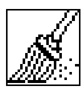
 **Display Attributes**

■ **Show Undeformed**

Line Style: - - - - - - - -

Apply

You may reset the graphics if you click on this icon:

 **Reset Graphics**

You can go back and select any *Results Case, Fringe Results or Deformation Results* you are interested in.

Quit MSC.Patran when you are finished with this exercise.