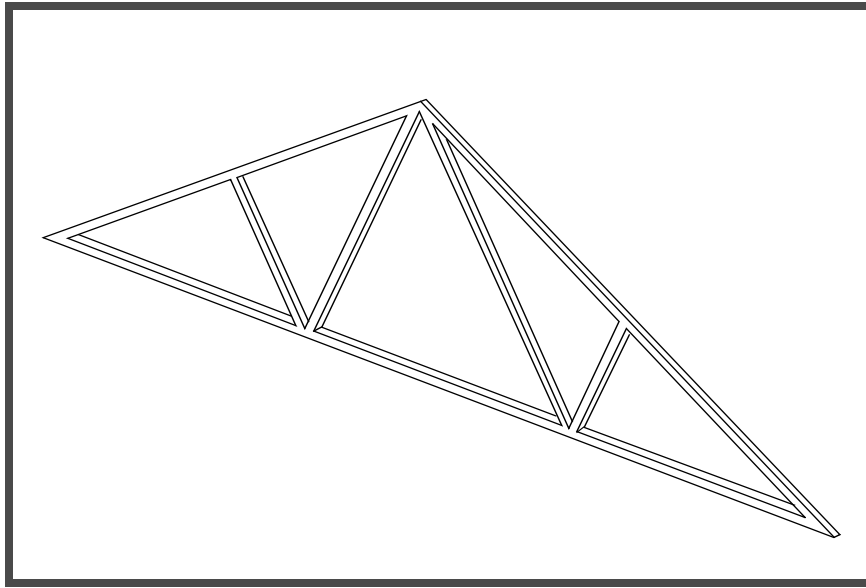

WORKSHOP 1

Linear Static Analysis of a Simply-Supported Truss



Objectives:

- Create a MSC.Nastran input file directly or by using MSC.Patran.
- Run the analysis using MSC.Nastran.
- Review results.

Model Description:

Below is a finite element representation of the truss structure shown on page 1-1. The nodal coordinates provided are defined in the Global cartesian coordinate system (MSC.Nastran Basic system).

The structure is comprised of truss segments connected by smooth pins such that each segment is either in tension or compression. The structure has a pinned support at Grid Point 1 and is supported by a roller at Grid Point 7. Point forces are applied at Grid Points 2, 4, and 6. In addition, out of plane translations and all rotations shall be constrained for all Grids.

HINT: DOF 3456 for Grid 1 through 7 can be constrained by using the permanent single point constraint option in the GRID entry.

Figure 1.1 - Grid Coordinates and Element Connectivities

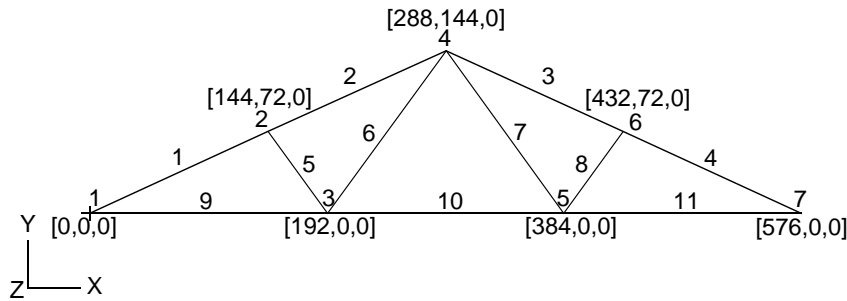


Figure 1.2 - Loads and Boundary Conditions

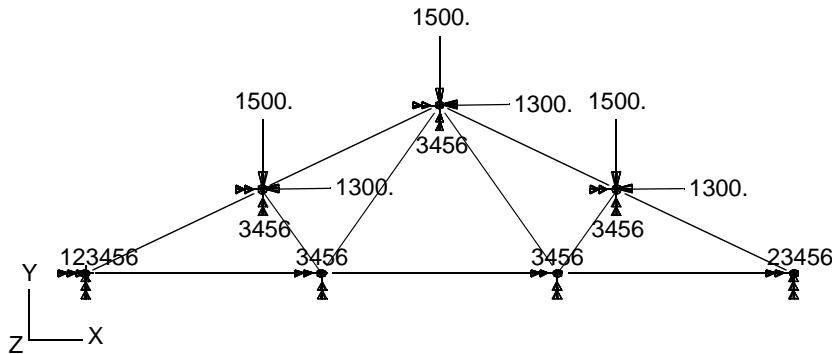


Table 1.1 - Model Properties

Cross-Sectional Area:	5.25 in²
Elastic Modulus:	1.76E+06 psi
Poisson Ratio:	0.3
Tension Stress Limit:	1900 psi
Compression Stress Limit:	1900 psi

Suggested Exercise Steps:

- Generate a finite element representation of the truss structure using (GRID) and (CROD) elements.
(**HINT:** Remember to use permanent constraints for DOF 3456.)
- Define material (MAT1) and element (PROD) properties.
- Apply simply-supported boundary constraints (SPC1) and point forces (FORCE).
- Use the load and boundary condition sets to define a load case (SUBCASE).
- Prepare the model for a linear static analysis (SOL 101).
- Submit it for a linear static analysis.
- Review results.

Exercise Procedure:

1. Users who are not utilizing MSC.Patran for generating an input file should go to Step 17, otherwise, proceed to Step 2.

2. Create a new database called **prob1.db**.

File/New...

New Database Name:

prob1

OK

In the New Model Preferences form set the following:

Tolerance:

◆ **Default**

Analysis Code:

MSC/NASTRAN

Analysis Type:

Structural

OK

Activate the entity labels by selecting the **Show Labels** button on the toolbar.



Show Labels

Change to a front view by selecting the **Front View** button on the toolbar.



Front View

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

3. Create the nodes by manually defining their respective coordinates:

◆ **Finite Elements**

Action:

Create

Object:

Node

Method:

Edit

Associate with Geometry (turn off)

Node Location List:

[0, 0, 0]

Apply

Repeat the previous operation to create the remaining nodes. Refer to the figure on page 1-3 for the nodal coordinates.

Node Location List:

[144, 72, 0]

Apply

Node Location List:

[192, 0, 0]

Apply

Node Location List:

[288, 144, 0]

Apply

Node Location List:

[384, 0, 0]

Apply

Node Location List:

[432, 72, 0]

Apply

Node Location List:

[576, 0, 0]

Apply

Next, manually define the truss segment connectivities with BAR2 elements using our newly created nodes. Again, refer to page 1-3 for connectivity information.

◆ **Finite Elements**

Action:

Create

Object:

Element

Method:

Edit

Shape:

Bar

Topology:

Bar2

Node 1 =
 Node 2 =

Repeat the previous operation until all the truss segments have been created.

Node 1 =
 Node 2 =

Hint: You can click in the box, then screen pick the node instead of typing the information in the box.

Node 1 =
 Node 2 =

Node 1 =
 Node 2 =

Node 1 =
 Node 2 =

Node 1 =
 Node 2 =

Node 1 =
 Node 2 =

Apply

Node 1 =

Node 5

Node 2 =

Node 6

Apply

Node 1 =

Node 1

Node 2 =

Node 3

Apply

Node 1 =

Node 3

Node 2 =

Node 5

Apply

Node 1 =

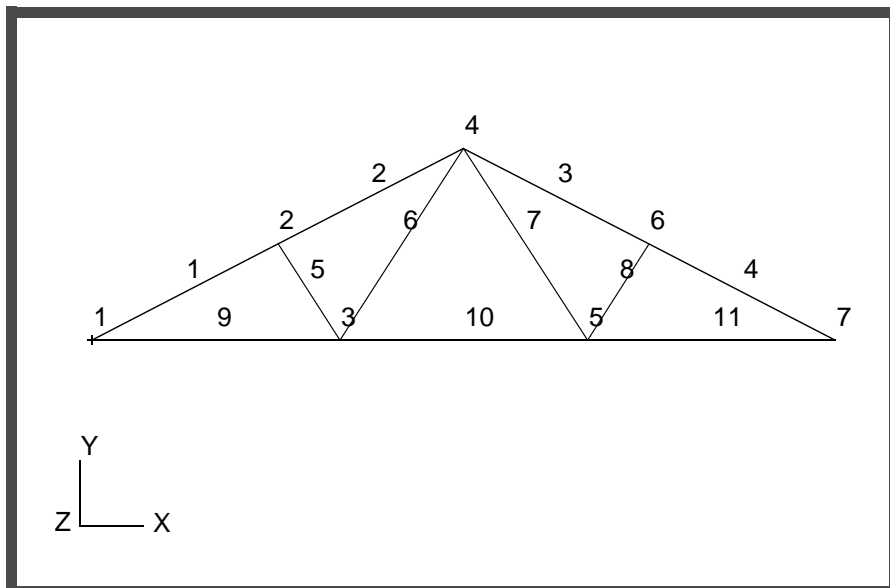
Node 5

Node 2 =

Node 7

Apply

Figure 1.3 - Nodal and Element Locations



4. Next, define a material using the specified modulus of elasticity and allowable stresses.

◆ **Materials**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Isotropic"/>
<i>Method:</i>	<input type="text" value="Manual Input"/>
<i>Material Name:</i>	<input type="text" value="mat_1"/>

<i>Constitutive Model:</i>	<input type="text" value="Linear Elastic"/>
<i>Elastic Modulus =</i>	<input type="text" value="1.76E6"/>
<i>Poisson Ratio=</i>	<input type="text" value="0.3"/>

<i>Constitutive Model:</i>	<input type="text" value="Failure"/>
<i>Tension Stress Limit =</i>	<input type="text" value="???"/> <i>(Enter material limit)</i>
<i>Compression Stress Limit =</i>	<input type="text" value="???"/> <i>(Enter material limit)</i>

5. Next, reference the material that was created in the previous step. Define the properties of the truss segments using the specified cross-sectional data.

◆ **Properties**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="1D"/>
<i>Type:</i>	<input type="text" value="Rod"/>

Property Set Name:

prop_1

Input Properties...

Material Name:

m:mat_1

(HINT: You can select the material *mat_1* from the available property sets.)

Area:

???

(Enter cross-sectional area)

OK

If you wish to use the mouse to select the members, you must first click on the Select Members input box. Then, select the **Beam Element** icon from the Select Menu in order to select entire model.



Beam Element

Select Members:

Elm 1:11

Add

Apply

- Shrink the elements by 10% for clarity; this allows us to easily assess the element connectivities. Use the **Display/Finite Elements...** option.

Display/Finite Elements...

FEM Shrink:

0.10

Apply

Create three nodal constraints and apply them to the analysis model. These boundary conditions represent the pinned support, roller support, and the permanent constraint.

- The left-hand support is defined as follows:

◆ **Loads/BCs**

Action:

Create

Object:

Displacement

Type:

Nodal

New Set Name:

Input Data...

Translations < T1 T2 T3 >

OK

Select Application Region...

Geometry Filter: **FEM**

Select Nodes:

Add

OK

Apply

6b. The right-hand support is located at the opposite end of the truss.

◆ Loads/BCs

Action:

Object:

Type:

New Set Name:

Input Data...

Translations < T1 T2 T3 >

OK

Select Application Region...

Geometry Filter: **FEM**

Select Nodes:

Add

OK

Apply

6c. The out of plane translations and all rotations can be constrained as follows:

◆ **Loads/BCs**

Action:

Create

Object:

Displacement

Type:

Nodal

New Set Name:

permanent_constraint

Input Data...

Translations < T1 T2 T3 >

< , , 0 >

Rotations < R1 R2 R3 >

< 0, 0, 0 >

OK

Select Application Region...

Geometry Filter:

◆ FEM

Select Nodes:

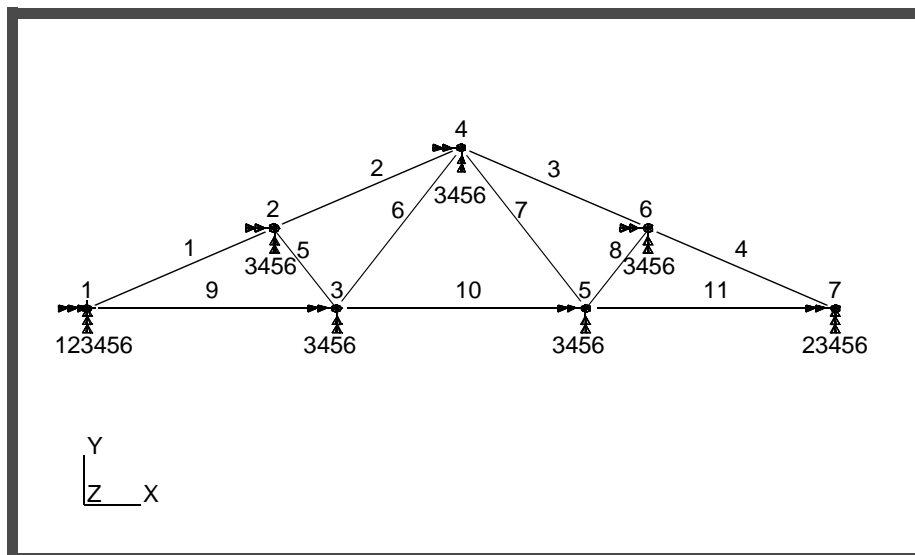
Node 1:7

Add

OK

Apply

Figure 1.4 - Displacement Constraints



7. Apply forces to the upper joints of the truss as shown on page 1-3. Vertical forces of 1500 lbs and horizontal forces of 1300 lbs should be applied at the proper nodes.

7a. First, define the vertical load.

◆ **Loads/BCs**

Action:

Create

Object:

Force

Type:

Nodal

New Set Name:

force_1

Input Data...

Force < F1 F2 F3 >

<0, -1500, 0>

OK

Select Application Region...

Geometry Filter:

◆ **FEM**

Select Nodes:

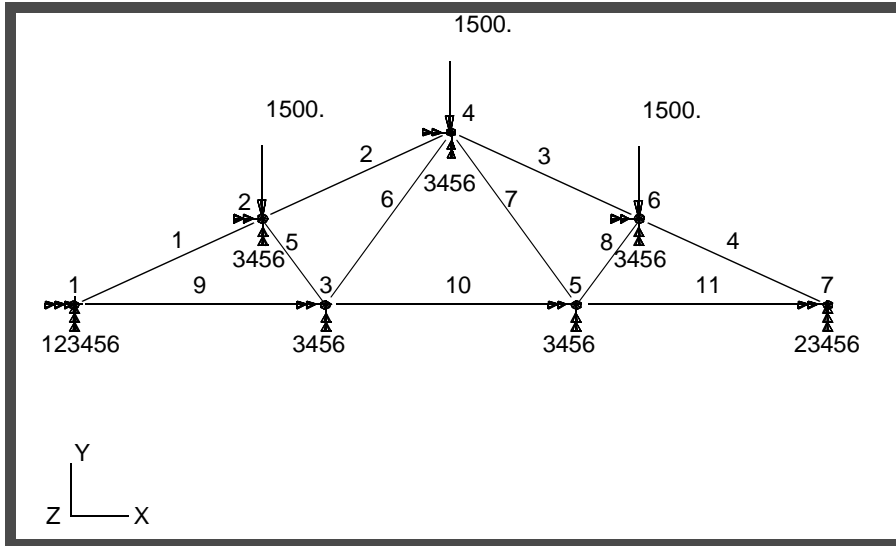
Node 2:6:2

Add

OK

Apply

Figure 1.5 - Vertical Forces



8. Next, define the horizontal forces.

◆ **Loads/BCs**

Action:

Create

Object:

Force

Type:

Nodal

New Set Name:

force_2

Input Data...

Force < F1 F2 F3 >

<-1300, 0, 0>

OK

Select Application Region...

Geometry Filter:

◆ **FEM**

Select Nodes:

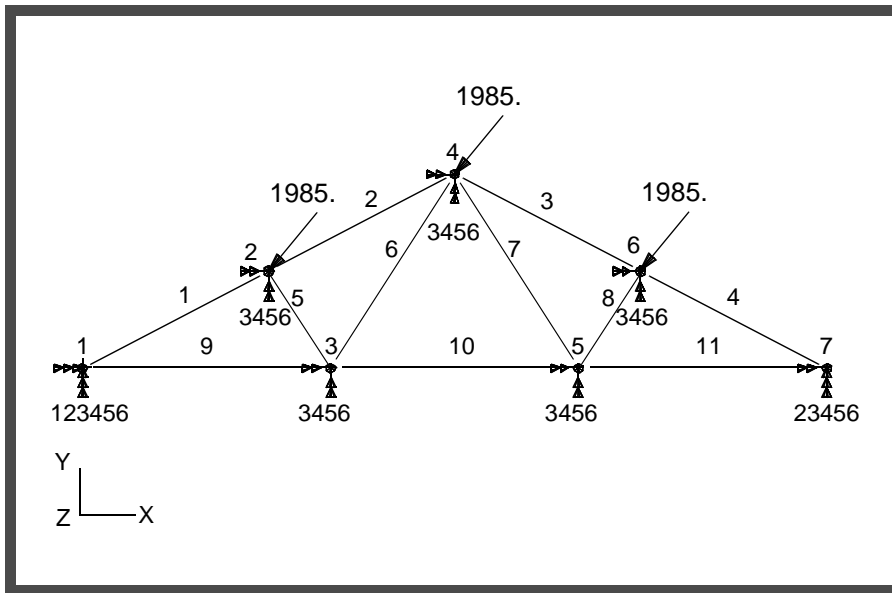
Node 2:6:2

Add

OK

Apply

Figure 1.6 - Resultant Loads



- 8a. Reset the display by selecting the **Reset Graphics** icon on the **Top Menu Bar**.



Reset Graphics

To display only the horizontal forces, change the Action on the **Load/BCs** form to **Plot Markers**.

◆ **Loads/BCs**

Action:

Plot Markers

Select the **Force_force_2** set in the *Assigned Load/BC Sets* box by highlighting it. Also apply the markers to the current group **default_group**.

Assigned Load/BC Sets:

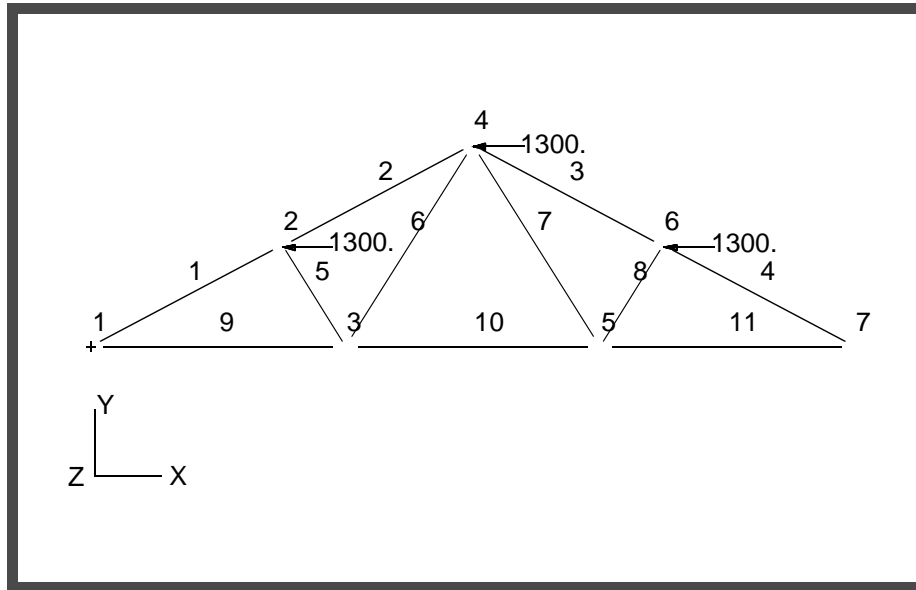
Force_force_2

Select Groups:

default_group

Apply

Figure 1.7 - Horizontal Forces



Deactivate the entity labels by selecting the **Hide Labels** button on the toolbar.



Hide Labels

9. Create a load case that references the forces and boundary conditions that have already been defined.

◆ **Load Cases**

Action:

Create

Load Case Name:

truss_lbc

Load Case Type:

Static

Assign/Prioritize Loads/BCs

Select Individual Load/BCs
(Select from menu)

Displ_permanent_constraint
Displ_pin
Displ_roller
Force_force_1
Force_force_2

OK

Apply

Plot the Load/BCs markers and post them to the current group.

◆ **Loads/BCs**

Action:

Plot Markers

Select all the Load/BC sets in the *Assigned Load/BC Sets* box by highlighting all of them. Post the markers to the current group.

Assigned Load/BCs Sets:

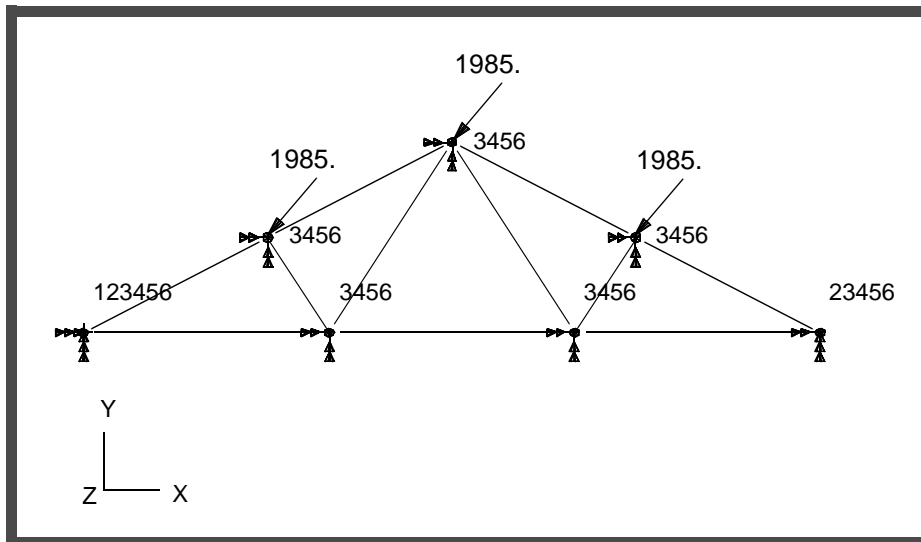
Displ_out_of_plane
Displ_pin
Displ_roller
Force_force_1
Force_force_2

Select Groups:

default_group

Apply

Figure 1.8 - Forces and Boundary Conditions



Reset the display by selecting the **Reset Graphics** icon on the **Top Menu Bar**.



Refresh Graphics

10. Display your model in its unshrunk state using the **Display/Finite Elements...** option.

Display/Finite Elements...

FEM Shrink:

0.0

Apply

Cancel

Now you are ready to generate an input file for analysis.

11. Click on the **Analysis** radio button on the **Top Menu Bar** and complete the entries as shown here:

◆ **Analysis**

Action:

Analyze

Object:

Entire Model

Method:

Analysis Deck

Job Name:

prob1

Translation Parameters...

Data Output:

XDB and Print

OK

Solution Type...

Solution Type:

◆ **Linear Static**

Solution Parameters...

■ **Database Run**

■ **Automatic Constraints**

Data Deck Echo:

Sorted

OK

OK

Subcase Select...*Subcases For Solution Sequence:***truss_ibcs***Subcases Selected:***Default***(Click on this to deselect)***OK****Apply**

An MSC.Nastran input file called **prob1.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.Nastran Users:

12. MSC.Nastran users can generate an input file using the data from 1-3. The result should be similar to the output below (**prob1.dat**):

```

ID SEMINAR, PROB1
SOL 101
TIME 600
CEND
TITLE = PROB1
SUBCASE 1
  SUBTITLE=TRUSS_LBCS
  SPC = 2
  LOAD = 2
  DISPLACEMENT = ALL
  SPCFORCES = ALL
  STRESS = ALL
BEGIN BULK
PARAM, POST, 0
PROD   1   1   5.25
CROD   1   1   1   2
CROD   2   1   2   4
CROD   3   1   4   6
CROD   4   1   6   7
CROD   5   1   2   3
CROD   6   1   3   4
CROD   7   1   4   5
CROD   8   1   5   6
CROD   9   1   1   3
CROD  10   1   3   5
CROD  11   1   5   7

MAT1   1   1.76+6
+      A 1900. 1900.

GRID   1   0.   0.   0.   3456
GRID   2  144.  72.  0.   3456
GRID   3  192.  0.   0.   3456
GRID   4  288. 144.  0.   3456
GRID   5  384.  0.   0.   3456
GRID   6  432.  72.  0.   3456
GRID   7  576.  0.   0.   3456
SPCADD 2   1   3
LOAD   2   1.  1.  1   1.   3
SPC1   1  12   1
SPC1   3   2   7
FORCE  1   2   0  1500.  0.  -1.  0.
FORCE  1   4   0  1500.  0.  -1.  0.
FORCE  1   6   0  1500.  0.  -1.  0.
FORCE  3   2   0  1300. -1.  0.  0.
FORCE  3   4   0  1300. -1.  0.  0.
FORCE  3   6   0  1300. -1.  0.  0.
ENDDATA

```

Submitting the Input File for Analysis:

13. Submit the input file to MSC.Nastran for analysis.
 - 13a. To submit the MSC.Patran **.bdf** file, find an available UNIX shell window. At the command prompt enter **nastran prob1.bdf scr=yes**. Monitor the run using the UNIX **ps** command.
 - 13b. To submit the MSC.Nastran **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran prob1 scr=yes**. Monitor the run using the UNIX **ps** command.
14. When the run is completed, edit the **prob1.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.
15. While still editing **prob1.f06**, search for the word:

D I S P L A C E (spaces are necessary).

What are the components of the displacement vector for GRID 7 (translation only)?

Disp. X = _____
Disp. Y = _____
Disp. Z = _____

Search for the word:

S I N G L E (spaces are necessary).

What are the components of the reaction force at GRID 1?

Force X = _____
Force Y = _____
Force Z = _____

Search for the word:

S T R E S S (spaces are necessary).

What is the margin of safety for CROD 2?

M.S. = _____

What is the Axial Stress for CROD 7?

Axial Stress = _____

Comparison of Results:

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

■ This output generated by DISPLACEMENT=ALL

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
1	G	0.0	0.0	0.0	0.0	0.0	0.0
2	G	1.102585E-01	-4.731638E-01	0.0	0.0	0.0	0.0
3	G	3.948052E-02	-5.117254E-01	0.0	0.0	0.0	0.0
4	G	2.850379E-02	-4.871603E-01	0.0	0.0	0.0	0.0
5	G	6.129870E-02	-5.089226E-01	0.0	0.0	0.0	0.0
6	G	-3.558523E-02	-4.661461E-01	0.0	0.0	0.0	0.0
7	G	1.277922E-01	0.0	0.0	0.0	0.0	0.0

■ This output generated by SPCFORCES=ALL

F O R C E S O F S I N G L E - P O I N T C O N S T R A I N T

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1	G	3.900000E+03	2.900000E+03	0.0	0.0	0.0	0.0
7	G	0.0	1.600000E+03	0.0	0.0	0.0	0.0

■ This output generated by STRESS=ALL

S T R E S S E S I N R O D E L E M E N T S (C R O D)

ELEMENT ID.	AXIAL STRESS	SAFETY MARGIN	TORSIONAL STRESS	SAFETY MARGIN	ELEMENT ID.	AXIAL STRESS	SAFETY MARGIN	TORSIONAL STRESS	SAFETY MARGIN
1	-1.235161E+03	5.4E-01	0.0		2	-8.678073E+02	1.2E+00	0.0	
3	-7.293841E+02	1.6E+00	0.0		4	-6.814683E+02	1.8E+00	0.0	
5	-1.459390E+02	1.2E+01	0.0		6	1.459390E+02	1.2E+01	0.0	
7	3.691398E+02	4.1E+00	0.0		8	-3.691398E+02	4.1E+00	0.0	
9	3.619048E+02	4.2E+00	0.0		10	2.000000E+02	8.5E+00	0.0	
11	6.095238E+02	2.1E+00	0.0						

17. MSC.Nastran Users have finished this exercise. MSC.Patran Users should proceed to the next step.
18. Proceed with the Reverse Translation process, that is, attaching the **prob1.xdb** results file into MSC.Patran. To do this, return to the **Analysis** form and proceed as follows:

◆ **Analysis**

<i>Action:</i>	Attach XDB
<i>Object:</i>	Result Entities
<i>Method:</i>	Local
Select Results File...	
Filter	
<i>Available Files:</i>	prob1.xdb
OK	
Apply	

19. When the translation is complete and the Heartbeat turns green, bring up the **Results** form.

◆ **Results**

<i>Action:</i>	Create
<i>Object:</i>	Quick Plot

Choose the desired result case in the **Select Result Cases** list and select the result(s) in the **Select Fringe Result** list and/or in the **Select Deformation Result** list.

Apply

If you wish to reset your display graphics to the state it was in before you began post-processing your model, remember to select the **Reset Graphics** icon.



Reset Graphics

Quit MSC.Patran when you have completed this exercise.