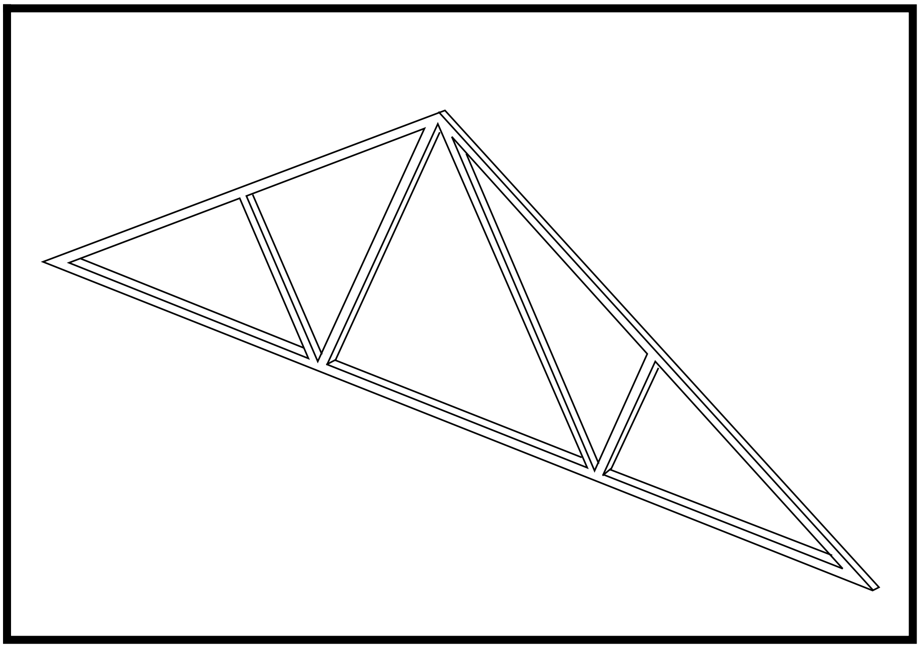
## WORKSHOP PROBLEM 1

# *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.

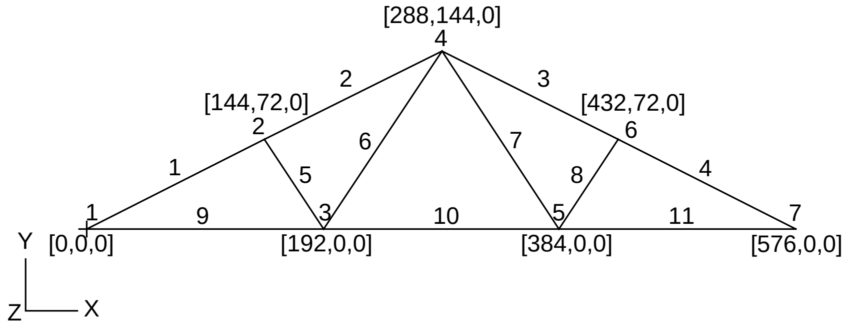
# Model Description:

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 contrained for all Grids.

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

Grid Coordinates and Element Connectivities




Loads and Boundary Conditions

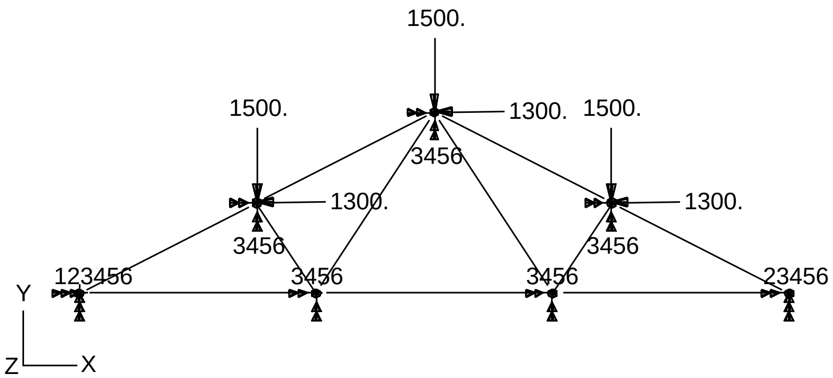



**Table 1.1**

| | |
|---|---|
| **Cross-Sectional Area** | **5.25 in$^2$** |
| **Elastic Modulus** | **1.76E6 psi** |
| **Poission's 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 loadcase (SUBCASE).
- Prepare the model for a linear static analysis (SOL 101).
- Submit it for a linear static analysis.
- Review results.

```
ID SEMINAR,PROB1
```

```
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 utilitizing 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:

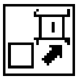
*Tolerance:* ◆ **Default**

*Analysis Code:* **MSC/NASTRAN**

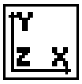
*Analysis Type:* **Structural**

**OK**

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


**Show Labels**

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



**Front View**

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

◆ **Finite Elements**

*Action:* **Create**

*Object:* **Node**

*Method:* **Edit**

❑ **Associate with Geometry**

*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 1**

*Node 2 =* **Node 2**

**Apply**

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

*Node 1 =* **Node 2**

*Node 2 =* **Node 4**

**Apply**

*Node 1 =* **Node 4**

*Node 2 =* **Node 6**

**Apply**

*Node 1 =* **Node 6**

*Node 2 =* **Node 7**

**Apply**

*Node 1 =* **Node 2**

*Node 2 =* **Node 3**

**Apply**

*Node 1 =* **Node 3**

*Node 2 =* **Node 4**

**Apply**

*Node 1 =* **Node 4**

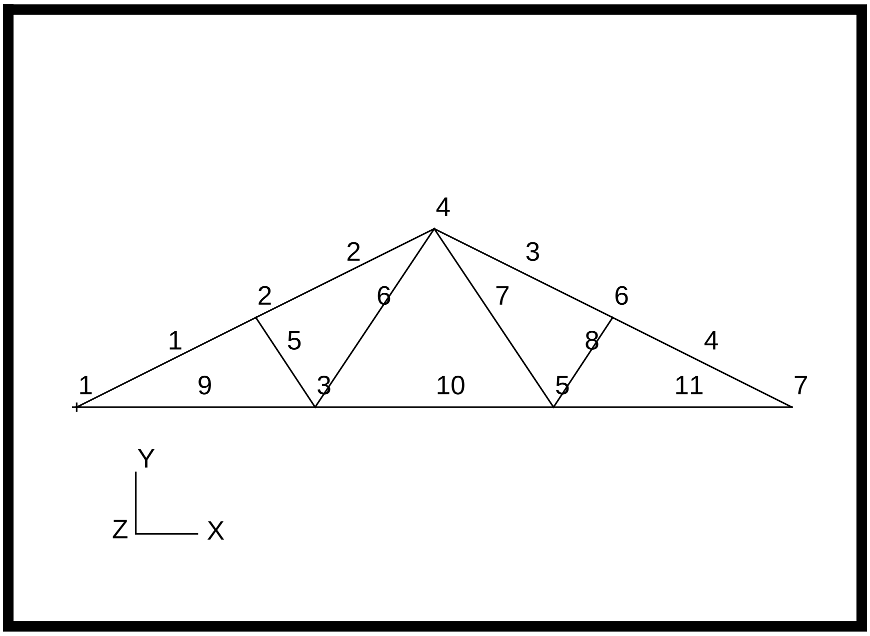
*Node 2 =* **Node 5**

**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 1.1** - Nodal and Element Locations



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

◆ **Materials**

| | |
|---|---|
| *Action:* | **Create** |
| *Object:* | **Isotropic** |
| *Method:* | **Manual Input** |
| *Material Name:* | **mat_1** |

**Input Properties ...**

| | |
|---|---|
| *Constitutive Model:* | **Linear Elastic** |
| *Elastic Modulus =* | **1.76E6** |
| *Poisson Ratio=* | **0.3** |

**Apply**

| | |
|---|---|
| *Constitutive Model:* | **Failure** |
| *Tension Stress Limit =* | **???** (Enter material limit) |
| *Compression Stress Limit =* | **???** (Enter material Limit) |

**Apply**

In the *Current Constitutive Models* data box, you will see **Failure - [n/a,,,,] - [Active]** and **Linear Elastic - [,,,,] - [Active]** appear. Click on **Cancel** to close the form.

**Cancel**

7. 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**

| | |
|---|---|
| *Action:* | **Create** |

| | |
|---|---|
| *Dimension:* | **1 D** |
| *Type:* | **Rod** |
| *Property Set Name:* | **rod** |
| **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** | |
| *Select Members:* | **Elm 1:11** |
| **Add** | |
| **Apply** | |

8. 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** | |
| **Cancel** | |

9. Create three nodal constraints and apply them to the analysis model. These boundary conditions represent the simply-supported ends of the truss, the fixed out of plane translations, and the constrained rotations.

9a. The left-hand support is defined as follows:

◆ **Loads/BCs**

| | |
|---|---|
| *Action:* | **Create** |

*Object:* **Displacement**

*Type:* **Nodal**

*New Set Name:* **pin**

**Input Data...**

*Translations < T1 T2 T3 >* **<0, 0, >**

**OK**

**Select Application Region...**

*Geometry Filter:* ◆ **FEM**

*Select Nodes:* **Node 1**

**Add**

**OK**

**Apply**

9b. The right-hand support is located at the opposite end of the truss.

◆ **Loads/BCs**

*Action:* **Create**

*Object:* **Displacement**

*Type:* **Nodal**

*New Set Name:* **roller**

**Input Data...**

*Translations < T1 T2 T3 >* **< , 0, >**

**OK**

**Select Application Region...**

*Geometry Filter:* ◆ **FEM**

*Select Nodes:* **Node 7**

**Add**

**OK**

**Apply**

9c. The out of plane translations and all rotations can be constrained as follows:

◆ **Loads/BCs**

| | |
|---|---|
| *Action:* | **Create** |
| *Object:* | **Displacement** |
| *Type:* | **Nodal** |
| *New Set Name:* | **out_of_plane** |

**Input Data...**

| | |
|---|---|
| *Translations < T1 T2 T3 >* | **< , , 0>** |
| *Rotations < R1 R2 R3 >* | **<0, 0, 0>** |

**OK**

**Select Application Region...**

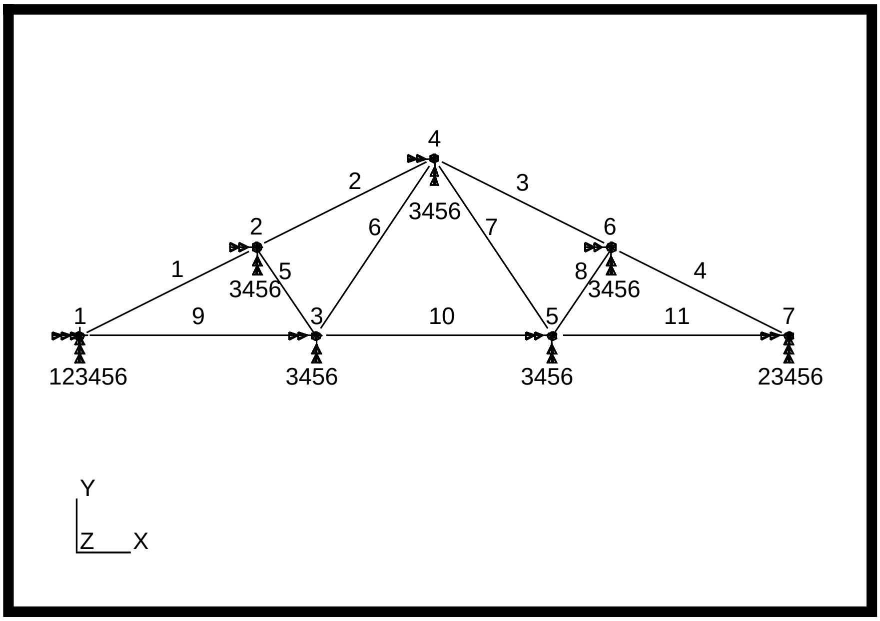
| | |
|---|---|
| *Geometry Filter:* | ◆ **FEM** |
| *Select Nodes:* | **Node 1:7** |

**Add**

**OK**

**Apply**

**Figure 1.2** - Displacement Constraints




10. 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.

10a. First, define the vertical load.

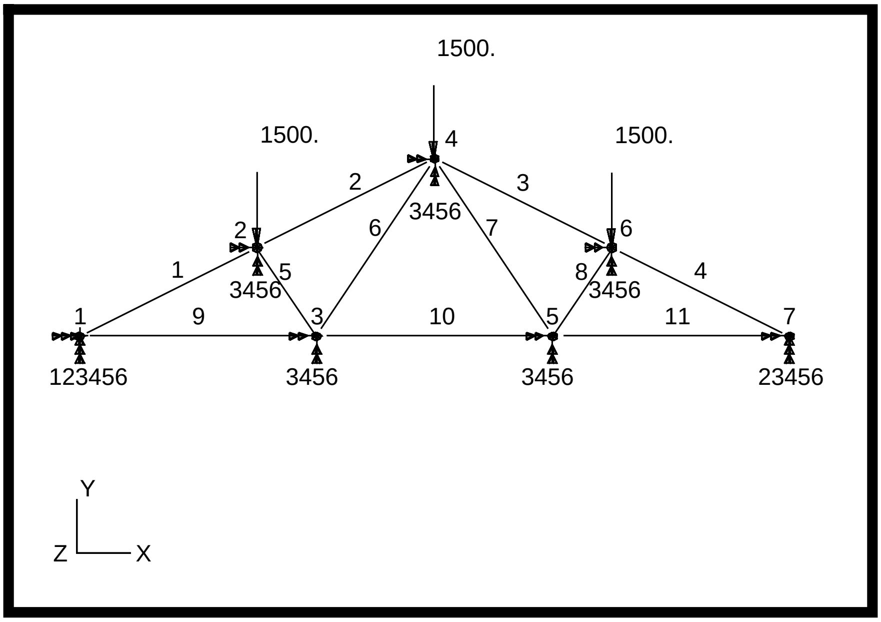
◆ **Loads/BCs**

| | |
|---|---|
| *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 1.3 -** Vertical Forces




11. 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** |

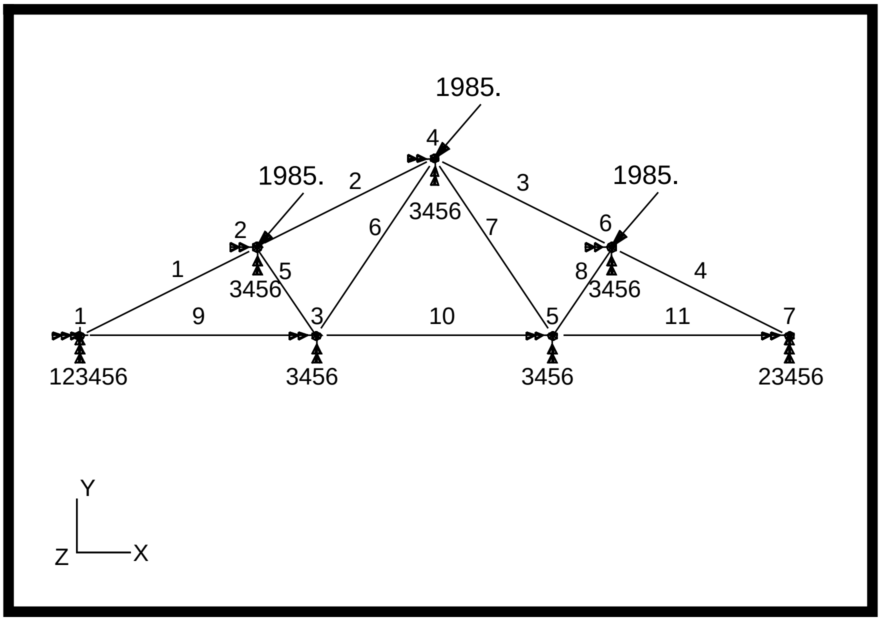
*Select Nodes:* **Node 2:6:2**

**Add**

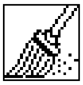
**OK**

**Apply**

**Figure 1.4** - Resultant Loads




11a. 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**

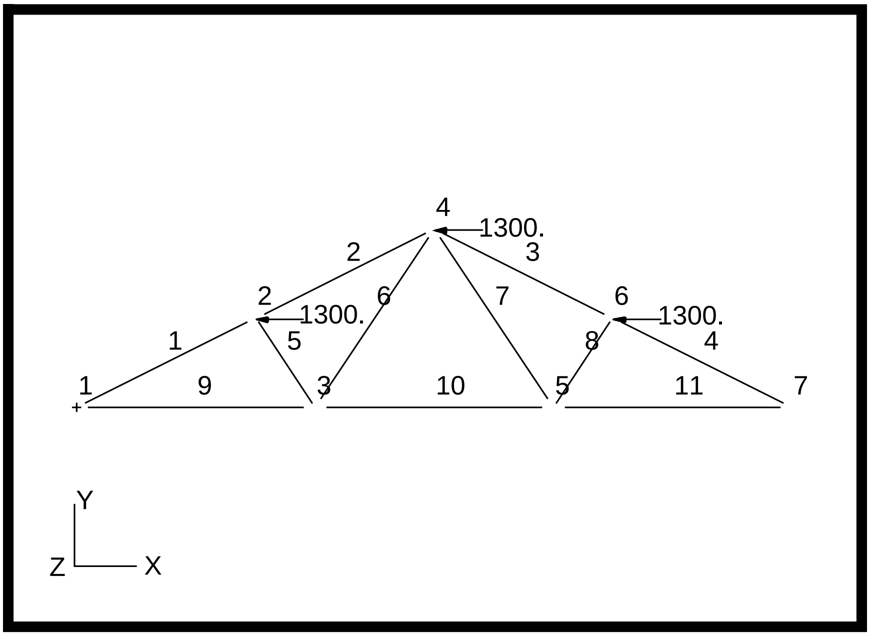
*Action:* **Plot Markers**

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**.

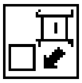
| | |
|---|---|
| *Assigned Load/BC Sets:* | **Force_force_2** |
| *Select Groups:* | **default_group** |

**Apply**

**Figure 1.5** - Horizontal Forces




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

 **Hide Labels**

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

◆ **Load Cases**

| | |
|---|---|
| *Action:* | **Create** |
| *Load Case Name:* | **truss_lbcs** |

*Load Case Type:* **Static**

**Assign/Prioritize Loads/BCs**

Select all the Load/BC sets in the *Select Loads/BCs to Add to Spreadsheet* box by clicking on all of them.

*Select Loads/BCs to Add to Spreadsheet*

**Displ_pin**
**Displ_roller**
**Displ_out_of_plane**
**Force_force_1**
**Force_force_2**

*** <u>NOTE</u>:** Be sure not to enter any load more than one time into the spreadsheet. Doing so will result in increasing the load by a factor equal to the number of times the load is entered into the spreadsheet. The increase in factor can be shown in two different manners. First, the LBC Scale Factor may show a value greater than one, or second, the spreadsheet may contain repeated entries of the same load. Either condition will result in erroneous loading conditions.

**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.

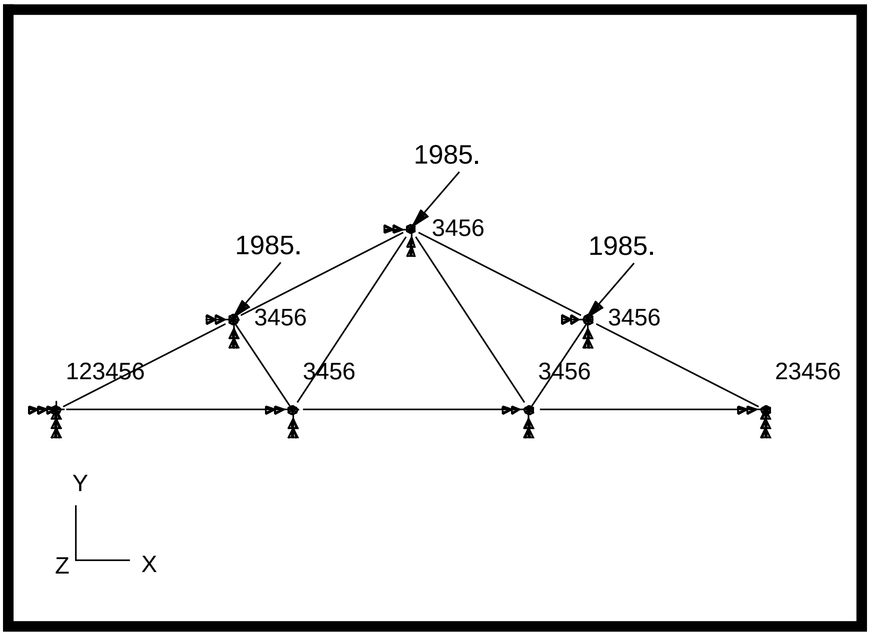
*Assigned Load/BCs Sets:*

**Displ_pin**
**Displ_roller**
**Displ_out_of_plane**
**Force_force_1**
**Force_force_2**

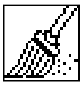
*Select Groups:* **default_group**

**Apply**

**Figure 1.6 -** Forces and Boundary Conditions




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



**Reset Graphics**

15. Display your model in its unshrunken 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.

16. 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...**

| | |
|---|---|
| *OUTPUT2 Format:* | **Binary** |
| *MSC/NASTRAN Version:* | **???** Set accordingly, here it is 70. |

**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_lbcs** |
| *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. MSC/PATRAN Users should proceed to **step 18**.

# Generating an input file for MSC/NASTRAN Users:

17. 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
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
+      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 MSC/NASTRAN and MSC/PATRAN USERS:

18. Submit the input file to MSC/NASTRAN for analysis.

    18a. 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.

    18b. 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.

19. 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.

    19a. 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:

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

# Comparison of Results:

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 DISPLACEMENT=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 SPCFORCES=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 | | | | | | |

- **This output generated by STRESS=ALL**

**21. MSC/NASTRAN Users have finished this exercise. MSC/PATRAN Users should proceed to the next step.**

22. Proceed with the Reverse Translation process, that is, importing the **prob1.op2** results file into MSC/PATRAN. To do this, return to the **Analysis** form and proceed as follows:

◆ **Analysis**

| | |
|---|---|
| *Action:* | **Read Output 2** |
| *Object:* | **Result Entities** |
| *Method:* | **Translate** |

**Select Results File...**

**Filter**

| | |
|---|---|
| *Available Files:* | **prob1.op2** |

**Ok**

**Apply**

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

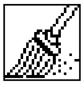
◆ **Results**

| | |
|---|---|
| *Action:* | **Create** |
| *Object:* | **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.