LESSON 8

Linear Static Analysis of a Simply-Supported Truss



Objectives:

- Create a finite element model by explicitly defining node locations and element connectivities.
- Define a MSC/NASTRAN analysis model comprised of CROD elements.
- Prepare a MSC/NASTRAN input file for a linear static analysis.
- Visualize analysis results.

MSC/NASTRAN 120 Exercise Workbook - Version 70 (MSC/PATRAN 7.5)

8-2

Model Description:

Below is a finite element representation of the truss structure shown on page 8-1. 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 is pinned at node 1 and supported by a roller at node 7. Point forces are applied at nodes 2, 4, and 6.

Grid Coordinates and Element Connectivities



Loads and Boundary Conditions



Compression Stress Limit 1900 psi

Suggested Exercise Steps:

- Open a new database.
- Explicitly generate a finite element representation of the truss structure without defining any geometry, i.e., the nodes (GRID) and element connectivities (CROD) should be defined manually.
- 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 and PARAMs).
- Generate an input file and submit it to the MSC/NASTRAN solver.
- Post-process results.
- Quit MSC/PATRAN.

Exercise Procedure:

1. Create a new database called **truss.db**.

File/New...

New Database Name:

Truce	
11433	

OK

In the New Model Preferences form set the following:

Tolerance:

Analysis Code:

Analysis Type:

ОК

◆ Default

MSC/NASTRAN

Structural

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



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

♦ Finite Elements

Action:

Object:

Method:

Create	
Node	
Edit	

□ Associate with Geometry

Auto Execute

Node Location List:

|--|

[140, 72, 0]

Apply

Repeat the previous operation to create the remaining nodes. Refer to the figure on page 8-3 for the nodal coordinates.

Node Location List:

Apply

[192, 0, 0]

[288, 144, 0]

[384, 0, 0]

[432, 72, 0]

Node Location List:

[576, 0, 0]

Apply

Next, manually define the truss segment connectivites with BAR2 elements using our newly created nodes. Again, refer to page 8-3 for connectivity information.

♦ Finite Elements

Action:	Create
Object:	Element
Method:	Edit
Shape:	Bar
Topology:	Bar2
□ Auto Execute	
<i>Node 1 =</i>	Node 1
<i>Node</i> 2 =	Node 2
Apply	

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

Node $1 =$	Node 1
<i>Node</i> 2 =	Node 2
Apply	
Node $1 =$	Node 2
<i>Node</i> 2 =	Node 4
Apply	
<i>Node 1 =</i>	Node 4
<i>Node</i> 2 =	Node 6
Apply	

LESSON 8

Node $1 =$	Node 6
<i>Node</i> 2 =	Node 7
Apply	
<i>Node 1</i> =	Node 2
<i>Node</i> 2 =	Node 3
Apply	
<i>Node 1</i> =	Node 3
<i>Node</i> 2 =	Node 4
Apply	
<i>Node 1</i> =	Node 4
<i>Node</i> 2 =	Node 5
Apply	
<i>Node 1</i> =	Node 5
<i>Node</i> 2 =	Node 6
Apply	
<i>Node 1</i> =	Node 1
<i>Node</i> 2 =	Node 3
Apply	
<i>Node 1</i> =	Node 3
<i>Node</i> 2 =	Node 5
Apply	
<i>Node 1</i> =	Node 5
<i>Node</i> 2 =	Node 7
Apply	

MSC/NASTRAN 120 Exercise Workbook - Version 70 (MSC/PATRAN 7.5)





3. 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
Apply	
Constitutive Model:	Failure
Tension Stress Limit =	??? (Enter material limit)
Compression Stress Limit =	(Enter material Limit)
Apply	

Current Constitutive Models:



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



Display/Finite Elements...



MSC/NASTRAN 120 Exercise Workbook - Version 70 (MSC/PATRAN 7.5)

LESSON 8

- 6. Create two displacement constraints and apply them to the analysis model. These boundary conditions represent the simply-supported ends of the truss.
- 6a. The left-hand support is defined as follows:

◆ Loads/BCs	
Action:	Create
Object:	Displacement
Method:	Nodal
New Set Name:	pin
Input Data	
Translation < T1 T2 T3 >	< 0, 0, >
ОК	
Select Application Region	
Geometry Filter:	♦ FEM
Select Nodes:	Node 1
Add	
ОК	
Apply	

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

◆ Loads/BCs	
Action:	Create
Object:	Displacement
Method:	Nodal
New Set Name:	roller
Input Data	
Translation < T1 T2 T3 >	< ,0, >
ОК	

MSC/NASTRAN 120 Exercise Workbook - Version 70 (MSC/PATRAN 7.5)



LESSON 8

The displacement constraints are shown below:



- 7. Apply forces to the upper joints of the truss as shown on page 8-3. Vertical forces of 1500 lbs and horizontal forces of 1300 lbs should be applied at the proper nodes.
- 7a. First, define the vertical forces.

◆ Loads/BCs	
Action:	Create
Object:	Force
Method:	Nodal
New Set Name:	force_1
Input Data	



The vertical forces should appear as follows:



7b. Next, define the horizontal forces.



OK

Select Application Region	
Geometry Filter:	♦ FEM
Select Nodes:	Node 2:6:2
Add	
ОК	

When you are done, resultant forces will be displayed as follows:



7c. Reset the display by selecting the broom icon on the **Top Menu Bar**.



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



Action:

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**.



The display should appear as follows:



8. 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	
(Click each selection until all Loads/BCs have one entry in the spreadsheet)*	Displ_pin Displ_roller Force_force_1 Force_force_2

OK	
Apply	

* **NOTE:** 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.

Reset the display by selecting the broom icon on the Top Menu Bar.



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 high-lighting all of them. Post the markers to the current group.

Assigned Load/BCs Sets:

Select Groups:

Apply



default_group

Here is how the display should appear:



9. Deactivate the entity labels by using the **Display/Entity Color/** Label/Render... option.

Display/Entity Color/Label/Render...

Hide All Entity Labels	
Apply	
Cancel	

Reset the display by selecting the broom icon on the Top Menu Bar.



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

Display/Finite Elements...

FEM Shrink:

0.0

Apply	
Cancel	

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

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



Apply

An MSC/NASTRAN input file called **Truss.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.

Submit the input file to MSC/NASTRAN for analysis. To do this, find an available UNIX shell window and at the command prompt enter:

nastran Truss.bdf scr=yes

Monitor the run using the UNIX **ps** command.

- 10a. When the run is completed, edit the **Truss.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.
- 10b. While still editing **Truss.f06**, search for the word:

DISPLACE (spaces are necessary).

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



Search for the word:

SINGLE (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. =

MSC/NASTRAN 120 Exercise Workbook - Version 70 (MSC/PATRAN 7.5)

8-18

What is the Axial Stress for CROD 7?

Axial Stress =

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

♦ Analysis	
------------	--

Action:

Object:

Method:

Read Output2	
Result Entities	
Translate	

Select Results File...

Filter

Selected Results File

select the desired .op2 file

OK	
Apply	

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



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. And hit **Apply** to view the result(s) in the viewport.

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 broom icon.



Quit MSC/PATRAN when you have completed this exercise.