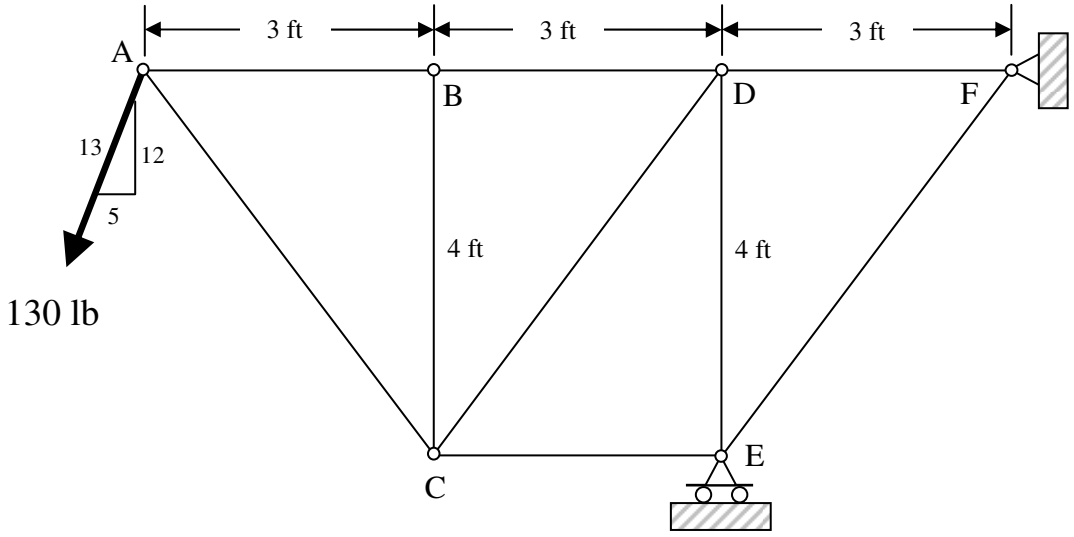
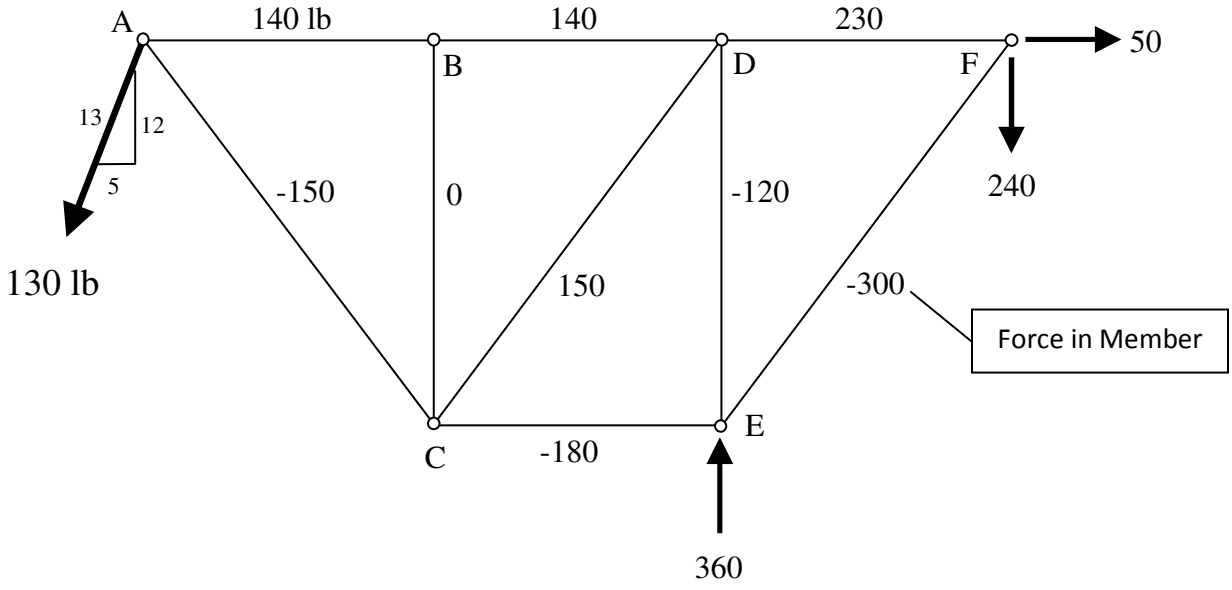
NASTRAN PROJECT FOR STATICS

The truss below is Problem 6-3 on Page 287 of Hibbeler, R.C., Engineering Mechanics: Statics, Fourteenth Edition, Pearson, 2016.



This truss is *statically determinate*; i.e., all of the internal forces in each member and all of the external reactions can be found by using the static equilibrium equations.

The answers from the hand calculations are given below.



" - " means " Compression "

It is desired to check the answers obtained from the hand calculations by analyzing this truss in Nastran.
Nastran uses an approximation technique called *finite elements* to solve problems. It does not use the same method as the hand calculations. The finite element method subdivides each member of the structure into a finite number of finite-sized elements.

The version of Nastran that we will be using is called NX Nastran. It is produced by Siemens Corporation. It comes with a modeler called Femap. Figures can be produced in Femap that will be later analyzed by NX Nastran.

In order for Nastran to solve any problems, it needs the following basic information.

1. Material Properties
   (Details concerning material properties will be discussed in Solid Mechanics)
2. Cross-Sectional Area Information
3. Types of Elements
4. Number of Elements
5. Geometry
6. Loads
7. Constraints

For the hand calculations, only items 5, 6, and 7 were needed since our truss is statically determinate.

<u>Additional Information Needed for Nastran</u>

- <u>Material</u>
  AISI 4340 Steel
- <u>Cross-Section</u>
  Solid Circular: Diameter = 3/4”

" - " means to "click the left mouse button" on the indicated menu choice.

Opening Femap - NX Nastran
- Start
- All Programs
- Engineering Applications
- FEMAP

Material Properties
- Model
- Material
(Input title)
- Type
- Isotropic
- OK
- Load
- AISI 4340 Steel
- OK
- OK
- Cancel

Cross-Sectional Properties and Types of Elements
- Model
- Property
(Input title)
- Material Down Arrow
(Click on the name that appears)
- Elem/Property Type
- Rod
[Rod Elements are used for 2-force members & our truss consists of all 2-force members.]
- OK
(Input cross-sectional area in square inches. Leave all other values as zero.)
- OK
- Cancel

Geometry
- Geometry
- Curve - Line
- Coordinates
(Input coordinates 0,0,0 for x,y,z respectively. This will be the coordinates of Point A.)
- OK
(Input coordinates 36,0,0 for x,y,z respectively. This will be the coordinates of Point B in inches.)
- OK
[Line AB has now been created. Repeat this process to create all other lines in the truss. Then Cancel.]

(Use the "Pan" arrows at the top of the screen to center the figure.)
(Use the "Magnification" magnifying glass symbols at the top of the screen to enlarge or decrease the size of the figure.)

Ctrl + A
If the figure that you are working on gets "Lost in Space", Press "Ctrl + A".

Mesh Size
- Mesh
- Mesh Control
- Size Along Curve
- Select All
- OK

(Input # of elements. Use 1. For rod elements which represent 2-force members, always use 1 rod element for each 2-force member.)
- OK
- Cancel

[We now have each 2-force member meshed into 1 subdivision.]

Creating Nodes and Elements
- Mesh
- Geometry
- Curve
- Select All
- OK
- Property Down Arrow

(Click on name that appears)
- OK

[We now have each 2-force member represented as one rod element, bounded by 2 nodes.]

Constraints
- Model
- Constraint
- Nodal

(Input title)
- OK

(Move the cursor on the figure and click the left button on the mouse to select the node corresponding to **Point F**.)
- Pinned

[Notice that T_x, T_y, T_z have been checked. This means that Point F has been stopped from translating in the x, y, and z directions.]

- OK
(Move the cursor on the figure and click the left button on the mouse to select the node corresponding to **Point E**.)
- OK
(Input title)
- T_y
- T_z
[This means that Point F has been stopped from translating in the y or z directions, but is free to translate in the x direction.]
- OK
(Move the cursor on the figure and click the left button on the mouse to select the node corresponding to **Point A**.)
(Move the cursor on the figure and click the left button on the mouse to select the node corresponding to **Point B**.)
(Move the cursor on the figure and click the left button on the mouse to select the node corresponding to **Point C**.)
(Move the cursor on the figure and click the left button on the mouse to select the node corresponding to **Point D**.)
- OK
(Input title)
- T_z
[This means that Points A, B, C, D have been stopped from translating in the z direction, but are free to translate in the x and y directions.]
- OK
- Cancel
[The numbers 1,2,3 will appear at the node representing Point F, the numbers 2,3 will appear at the node representing Point E, and the number 3 will appear at the nodes representing Points A, B, C, D. These indicate the Degrees of Freedom that have been stopped for that node. 1 = Translation in the x-direction, 2 = Translation in the y-direction, 3 = Translation in the z-direction, 4 = Rotation about the x-axis, 5 = Rotation about the y-axis, 6 = Rotation about the z-axis.]

<u>Load</u>
- Model
- Load
- Nodal
(Input title)
- OK
(Move the cursor and click the left button on the mouse to select the node corresponding to Point A.)
- OK
- Force
- Components
(For F_x, type -130*5/13)
(For F_y, type -130*12/13)
- OK
- Cancel

[The model is now complete.]

Running the Program
- File
- Analyze
- Create/Edit Set
- New
(Input title)
- Static
("Static" analysis = Linear analysis with quasi-static loading)
- OK
- Done
- OK
[Information about the analysis will appear on the left side of the screen as it is running.]

[If no error messages appear, then the run is complete and you will have output.]

Disk Space to Run NX Nastran

1. NX Nastran will save your Femap Model as a file with a ".modfem" suffix.

2. When running your file, 6 additional files will be created with the same filename as your model file but with different suffixes.

3. There must be enough disk space for the creation of these 6 additional files.

4. The run will not complete if there is not enough space for these 6 additional files.

5. Once the run has completed, the Output will be placed back in your original ".modfem" file.

6. At that point, the 6 additional files can be deleted.

7. Every time a run is executed, 6 additional files will be created. If not deleted, they will accumulate.

A. Check for:

1. Fatal Errors
2. Warning Messages with the words
   a. Singularities,
   b. Non-Positive Definite, or
   c. Autoconstrained

B. If any of the above messages are found:

1. there will be no output, or
2. the output will not be correct.

C. Singularities or Non-Positive Definite Warnings

1. These warning messages mean that there are one or more values that have become infinite (very large).
2. Usually it is due to the fact that one or more nodes have an infinite displacement.
3. This means that one or more nodes
   a. do not have proper constraints, or
   b. are not connected properly to other nodes in the system.

   Hence they are free to move or rotate in some direction with no restrictions.
4. This will most likely be accompanied by a Fatal Error which means that there should be no output.

D. Autoconstrained Warning

1. This warning means that there was an error (probably a singularity) and NASTRAN has automatically constrained the system in a way that the singularity will be removed.
2. There should be no fatal error in this case, but the output is of no use since we don't know how NASTRAN has constrained the system.

* If all went well, you should now have Output.

* It can be viewed by using the commands below.

Output

Figure of Model, Showing Constraints and Load
- View
- Options
- Labels, Entities and Color
- Label Parameters
- 3..14pt MS Sans Serif
[This will make values on the figure large enough to read.]
- Palette...
(Choose white)
- OK
- 1..Use View Color
[This will make labels on the figure white in color.]
- Node
- 0..No Labels
[This will remove the node numbers from the view.]
- OK
(Use the "Pan" arrows at the top of the screen to center the figure.)
(Use the "Magnification" magnifying glass symbols at the top of the screen to enlarge or decrease the size of the figure.)
- File
- Print
- Landscape
- OK

[This will give the 1st output page shown in the upcoming pages.]

Figure Displaying Internal Force in Each 2-Force Member
- View
- Select
- None –Model Only (for Deformed Style)
- Criteria
- Deformed and Contour Data
- Contour Down Arrow
- 3036..Rod Axial Force
- OK
- Contour Options
- Max Value
- OK
- OK

[The force in each 2-force member should appear by the member.
If they do not appear, follow these commands: View; Options; PostProcessing; Criteria – Elements That Pass; 1..Output Value; OK.]
- File
- Print
- Landscape
- OK

[This will give the 2nd output page shown in the upcoming pages.]

Figure Displaying External Reactions in X-Direction
- View
- Select
- None –Model Only (for Deformed Style)
- Criteria
- Deformed and Contour Data
- Contour Down Arrow
- 52..T1 Constraint Force
- OK
- OK
- File
- Print
- Landscape
- OK

[The external x-direction forces at each constraint will be displayed on each member coming into that constraint.]

[This will give the 3rd output page shown in the upcoming pages.]

Figure Displaying External Reactions in Y-Direction
- View
- Select
- None –Model Only (for Deformed Style)
- Criteria
- Deformed and Contour Data
- Contour Down Arrow
- 53..T2 Constraint Force
- OK
- OK
- File
- Print
- Landscape
- OK

[The external y-direction forces at each constraint will be displayed on each member coming into that constraint.]

[This will give the 4th output page shown in the upcoming pages.]

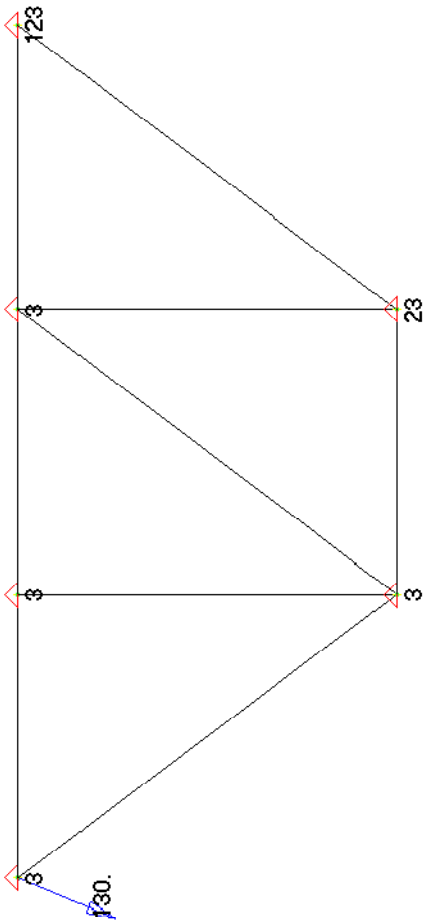
Deformed View of Model
- View
- Select
- Deform
- None –Model Only (for Contour Style)
- Deformed and Contour Data
- Deform Down Arrow
- 1..Total Translation
- OK
- OK
- File
- Print
- Landscape
- OK

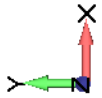
[The largest deformation will be given in inches in parentheses after the word "Deformed" at the bottom left of the screen.]

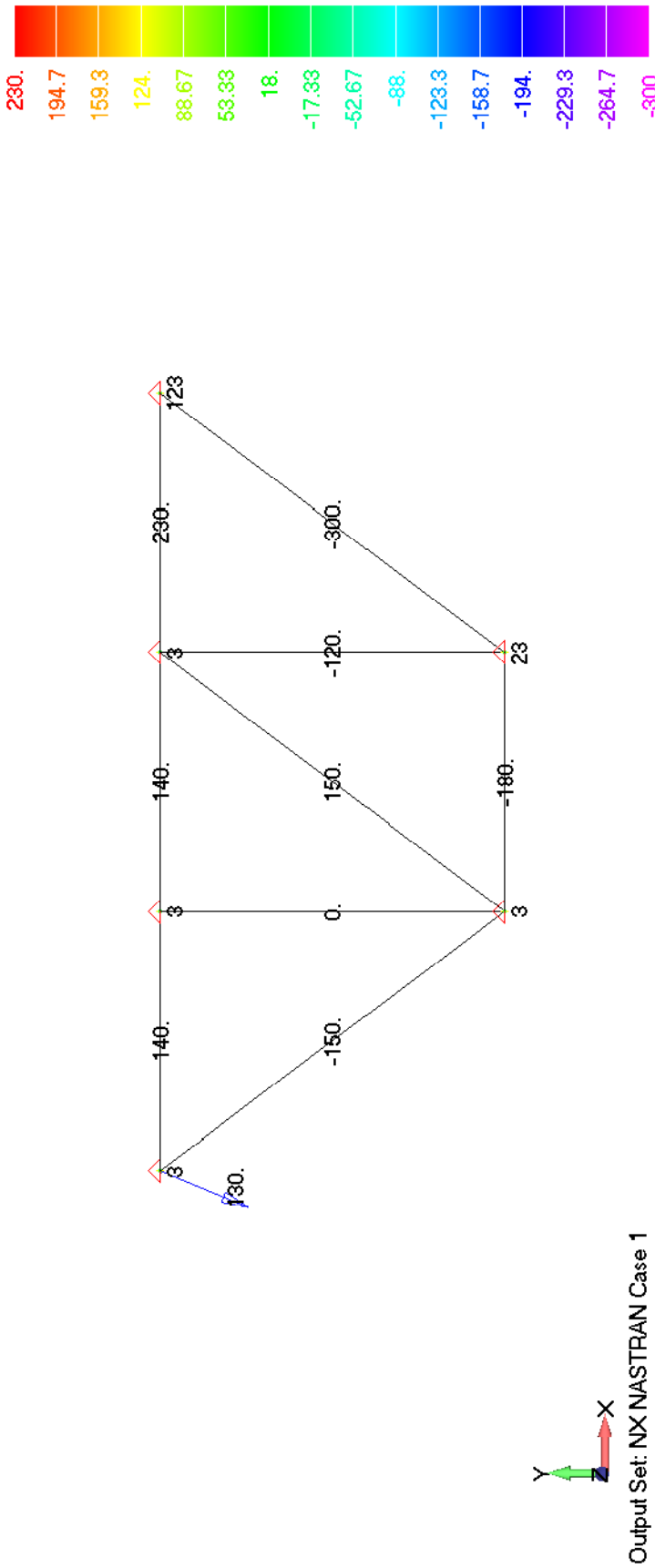
[This will give the 5th output page shown in the upcoming pages.]

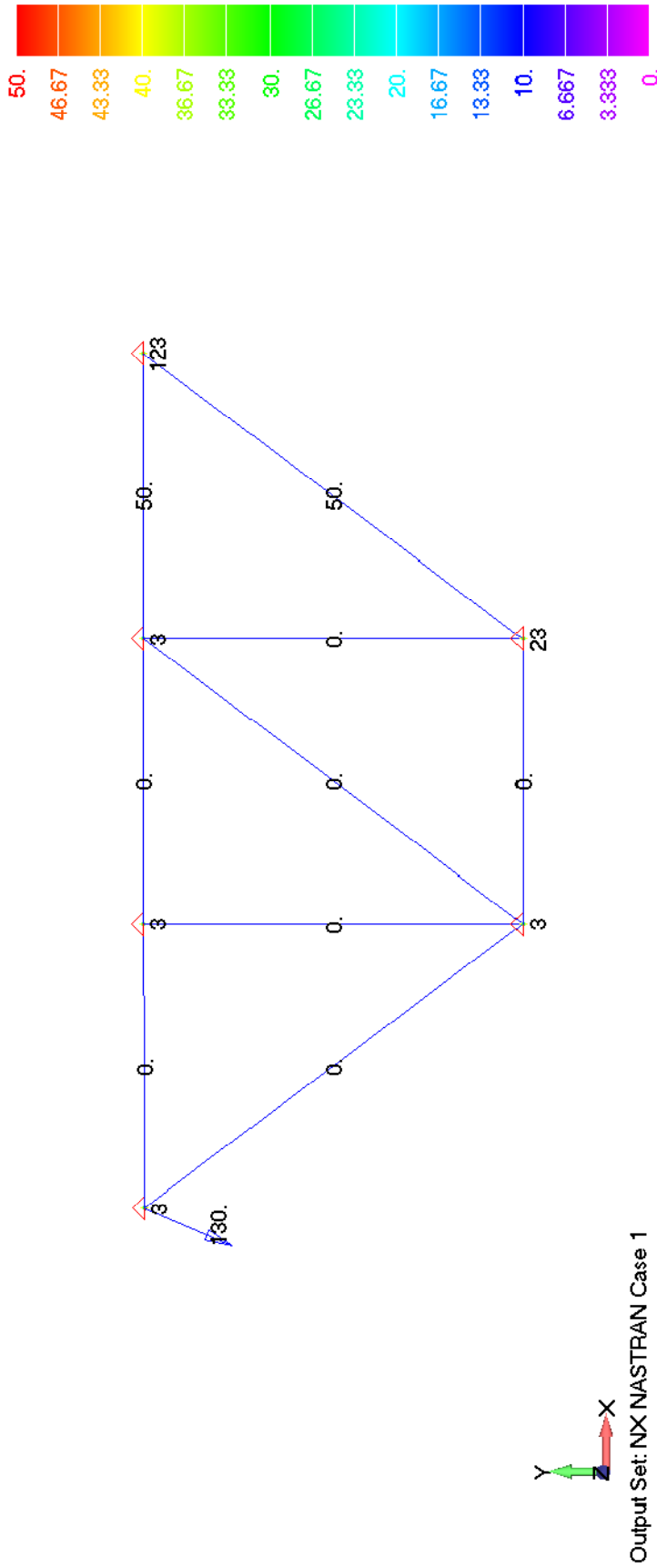
Turn-Ins

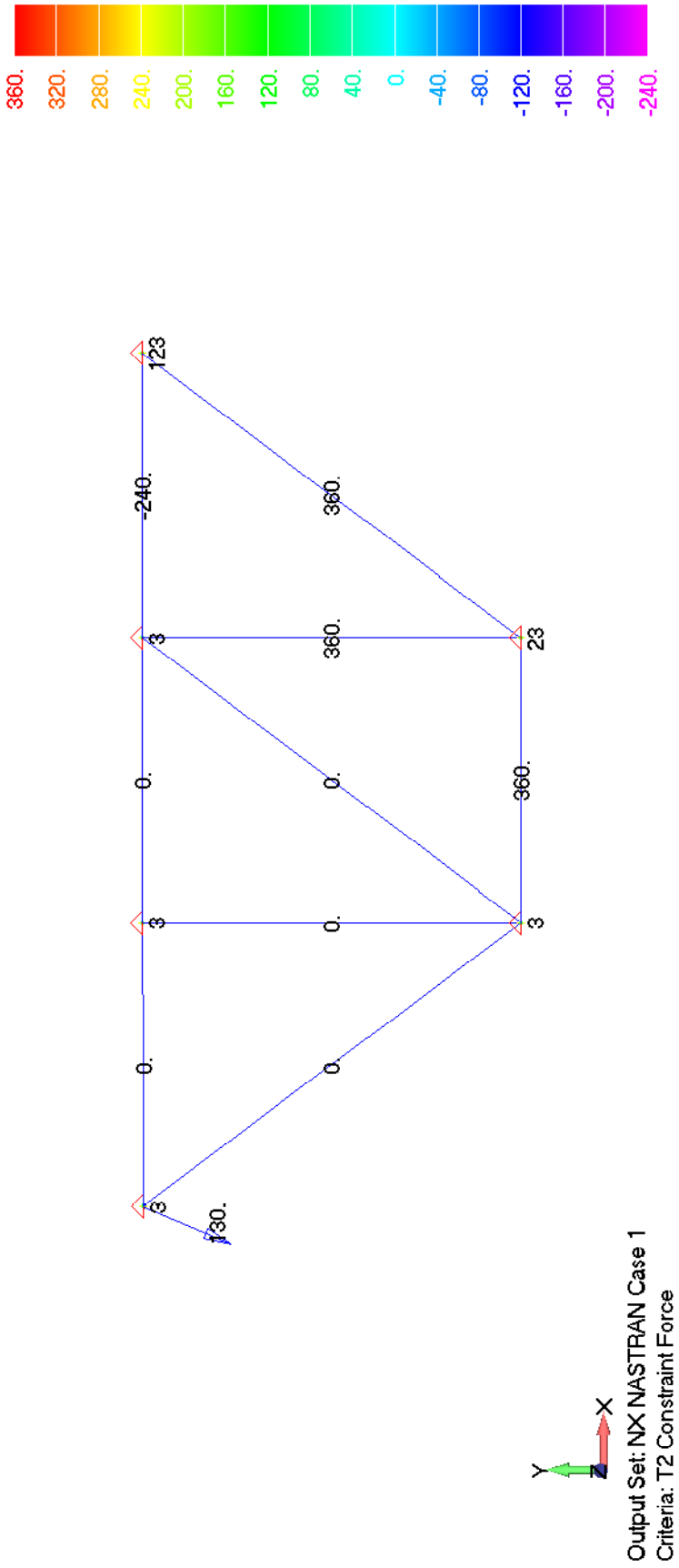
Turn in pages like those shown on Pages 11-16.

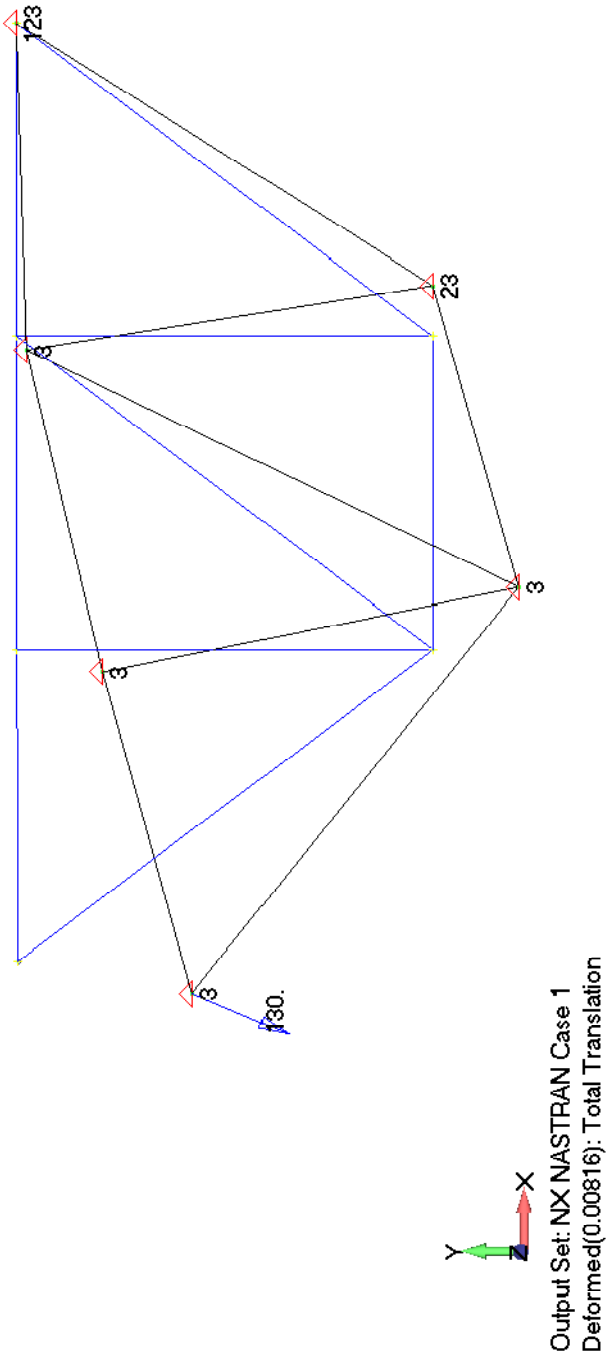

123
3
23
3
3
3
F30.


X
Y
Z


230.
194.7
159.3
124.
88.67
53.33
18.
-17.33
-52.67
-88.
-123.3
-158.7
-194.
-229.3
-264.7
-300.
123
230.
-300.
3
-120.
23
140.
150.
-180.
3
0.
3
140.
-150.
3
130.
Y
X
Output Set: NX NASTRAN Case 1
Criteria: Rod Axial Force


50.
46.67
43.33
40.
36.67
33.33
30.
26.67
23.33
20.
16.67
13.33
10.
6.667
3.333
0.
123
50.
50.
0.
0.
0.
23
3
3
3
0.
0.
0.
0
730.
Y
X
Output Set: NX NASTRAN Case 1
Criteria: T1 Constraint Force


360.
320.
280.
240.
200.
160.
120.
80.
40.
0.
-40.
-80.
-120.
-160.
-200.
-240.
123
-240.
360.
3
360.
23
0.
0.
360.
3
0.
3
0.
0
0
130.
Y
X
Output Set: NX NASTRAN Case 1
Criteria: T2 Constraint Force


123
23
3
3
3
30.
Y
X
Output Set: NX NASTRAN Case 1
Deformed(0.00816): Total Translation

Fill in the table below using your values from your Nastran Output.

	Theory (lb)	NASTRAN (lb)	% Error
F_{AB}	140		
F_{AC}	-150		
F_{BC}	0		
F_{BD}	140		
F_{CD}	150		
F_{CE}	-180		
F_{DE}	-120		
F_{DF}	230		
F_{EF}	-300		
Y_E	360		
X_F	50		
Y_F	-240		

Force In Member: " - " means " Compression "

Reaction: " - " means " – Global Axis Direction "