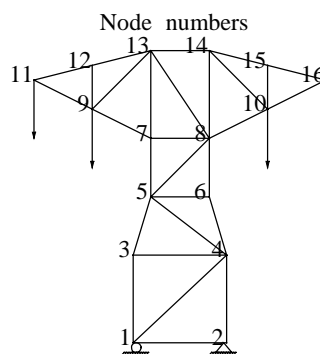
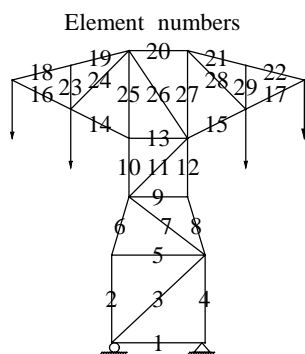
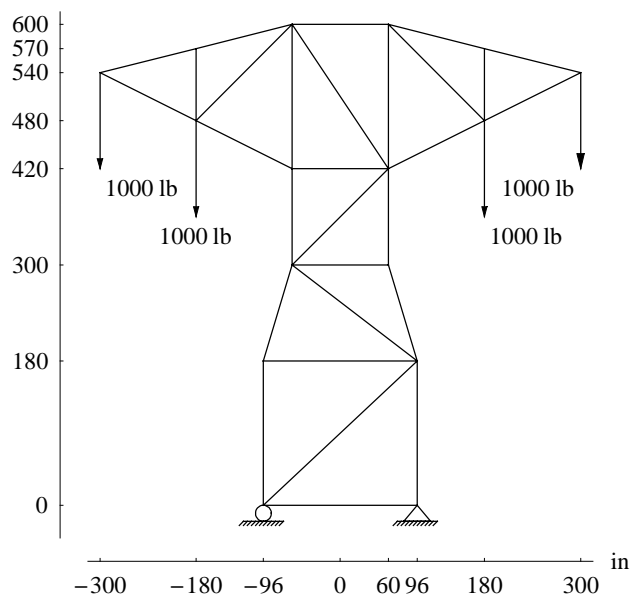


■ A.3.2 Truss Analysis

Determine nodal displacements and axial forces in the transmission tower shown in Figure A.1. The tower is made of steel ($E = 29 \times 10^6 \text{ lb/in}^2$) angle sections. The main vertical members (elements 2, 4, 6, 8, 10, and 12) have a cross sectional area of 20 in^2 . All other members have a cross-sectional area of 10 in^2 .



The simplest way to create model for structural frameworks is to manually create an input data file. For this example the model can be created by using the input as follows.

*Heading	*Element, type=T2D2	
Transmission tower	1,1,2	29,10, 15
*Node	2,1,3	** Element sets
1,-96,0,0	3,1,4	*Elset, elset=set1
2,96,0,0	4,2,4	2,4,6,8,10,12
3,-96,180,0	5,3,4	*Elset, elset=set2, generate
4,96,180,0	6,3,5	1,11,2
5,-60,300,0	7,4,5	13,29,1
6,60,300,0	8,4,6	*solidsection, elset=set1, material=steel
7,-60,420,0	9,5,6	20
8,60,420,0	10,5,7	*solidsection, elset=set2, material=steel
9,-180,480,0	11,5,8	10
10,180,480,0	12,6,8	*material, name=steel
11,-300,540,0	13,7,8	*elastic
12,-180,570,0	14,7,9	29000000, 0.3
13,-60,600,0	15,8,10	*Step, name=Step-1
14,60,600,0	16,9,11	Vertical load 1000 lb
15,180,570,0	17,10,16	*static
16,300,540,0	18,11,12	*cload
** Node sets for bc	19,12,13	loaded, 2, -1000
*Nset, nset=pin	20,13,14	*output, field, variable=preselect
2	21,14,15	*output, history, variable=preselect
*Nset, nset=roller	22,15,16	*elprint, frequency=1
1	23,9, 12	S,
*Nset, nset=loaded	24,9,13	*nodeprint, frequency=1
9,10,11,16	25,7, 13,	U,
*boundary	26,8, 13	RF,
pin,1,2	27,8, 14	*endstep
*boundary	28,10, 14	
roller,2		

The key-words starting with an asterisk are abaqus commands. The commands are not case sensitive. Lines starting with two asterisks are comment lines. The line following the *Heading command is used on output and plots to identify the analysis. The nodal coordinates are entered as node number, and x, y, and z coordinates, following the *Node command. In preparation for defining boundary conditions and loads, three nodal sets are created using the *Nset command. Nodes with pin supports are identified as a set called 'pin', those with roller support are called 'roller', and those with vertical load are called 'loaded'. Using the *boundary command, the degrees of freedom 1 and 2 (x and y displacements) are suppressed for node set 'pin'. The roller support is represented by suppressing only the second degree of freedom (y

displacement). The command `*Element` starts the definition of elements. The plane truss element in abaqus is identified as T2D2. The following lines define elements as element number, node at one end and the node at the other end. For assigning material and section properties the elements are classified into two sets identified as set1 and set2. Using the `*Elset` command the elements in set1 are assigned material as 'steel' and section as `*solidsection` with area = 20 and those in set2 are assigned the same material but an area = 10. The elastic material properties for steel are defined using `*material` and `*elastic` commands. A load of 1000 is applied in the -y direction to the 'loaded' nodes using the `*load` command.

The actual analysis commands are between the `*Step` and `*Endstep` commands. The `*static` specifies a static analysis. `*output`, `*elprint`, and `*nodeprint` commands control the results information written to the database, element results written to the .dat file and nodal results written to .dat file, respectively. The label S refers to stresses, U to displacements and RF to reaction forces.

Using this input file the following command is used to perform analysis.

```
abaqus job=tower
```

The abaqus output consisting of results and any error or warning messages are saved in a text file called tower.dat. The results can be viewed using the viewer as follows.

```
abaqus viewer database=tower
```

A plot of the axial stresses in the elements on a highly exaggerated deformed shape of the tower is shown in Figure A.12. The numerical results are exactly the same as those obtained from Ansys.

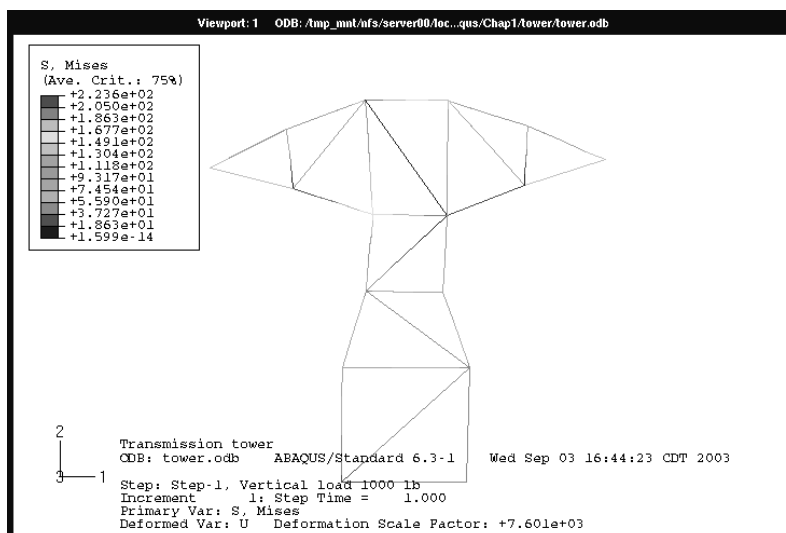


Figure A.12. Axial stresses shown on a deformed shape of the model