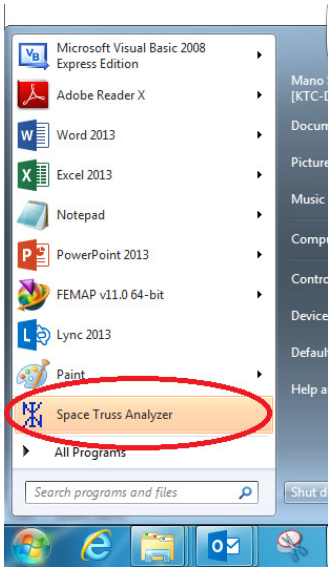
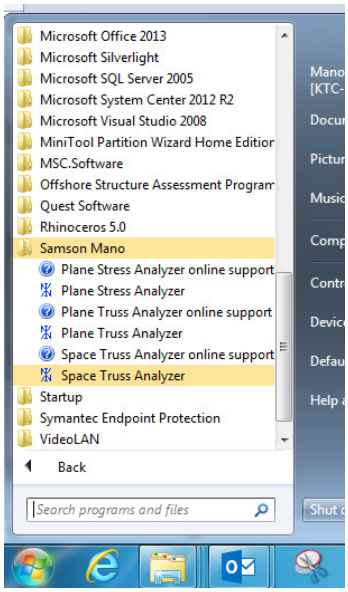
Space Truss Analyzer

Tutorial

Step: 0

After installing the file you can run the application from





Problem: 1 To demonstrate this application, I've solved this problem taken from: https://www.colorado.edu/engineering/CAS/courses.d/IFEM.d/IFEM.Ch21.d/IFEM.Ch21.pdf

EXERCISE 21.5 [C:25] Analyze the structure shown in Figure E21.1. This is a pin-jointed truss model of a 200-in-high (5m) transmission tower. The truss has 10 joints (nodes) and 25 members (elements). The truss geometry and node numbering are defined in Figure E21.1(a). Joints 1 and 2 at the top of the

tower lie on the {x,z} plane. The truss (but not the loads) is symmetric about the {y,z} and {x,z} planes. Figure E21.1(b) gives the element numbers.

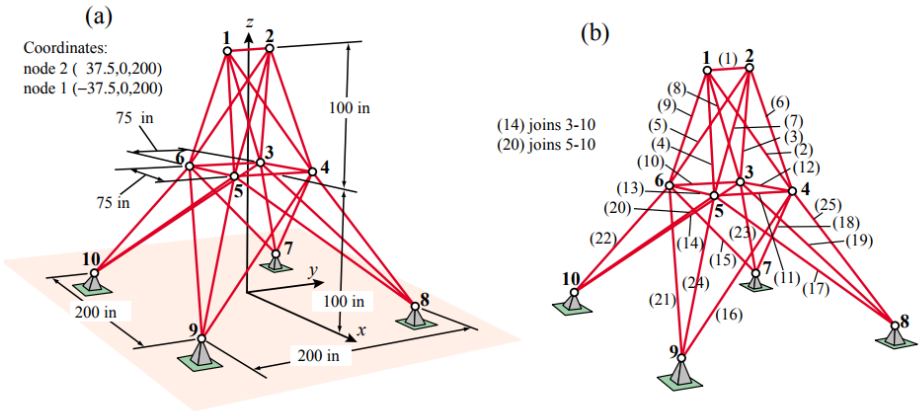


FIGURE E21.1. 25-member space truss model of a transmission tower. (a): Geometry definition and node numbers; (b) element numbers. For member properties and loads see Tables E21.1 and E21.2.

Table E21.1 Cross section areas of transmission tower members

Element	A (sq in)	Element	A (sq in)	Element	A (sq in)
1	0.033	10	0.010	19	1.760
2	2.015	11	0.010	20	1.760
3	2.015	12	0.014	21	1.760
4	2.015	13	0.014	22	2.440
5	2.015	14	0.980	23	2.440
6	2.823	15	0.980	24	2.440
7	2.823	16	0.980	25	2.440
8	2.823	17	0.980		
9	2.823	18	1.760		

Table E21.2 Applied load case for transmission tower

Node	x-load (lb)	y-load (lb)	z-load (lb)
1	1000	10000	−5000
2	0	10000	−5000
3	500	0	0
6	500	0	0

Applied forces at all other nodes are zero.
Own-weight loads not considered.

The members are aluminum tubes with the cross sections listed in Table E21.1.5 The modulus of elasticity is E = 10^7 psi for all members. The specific weight is 0.1 lb/in3. The applied load case to be studied is given in Table E21.2.

Step: 1

The given problem has 10 nodes

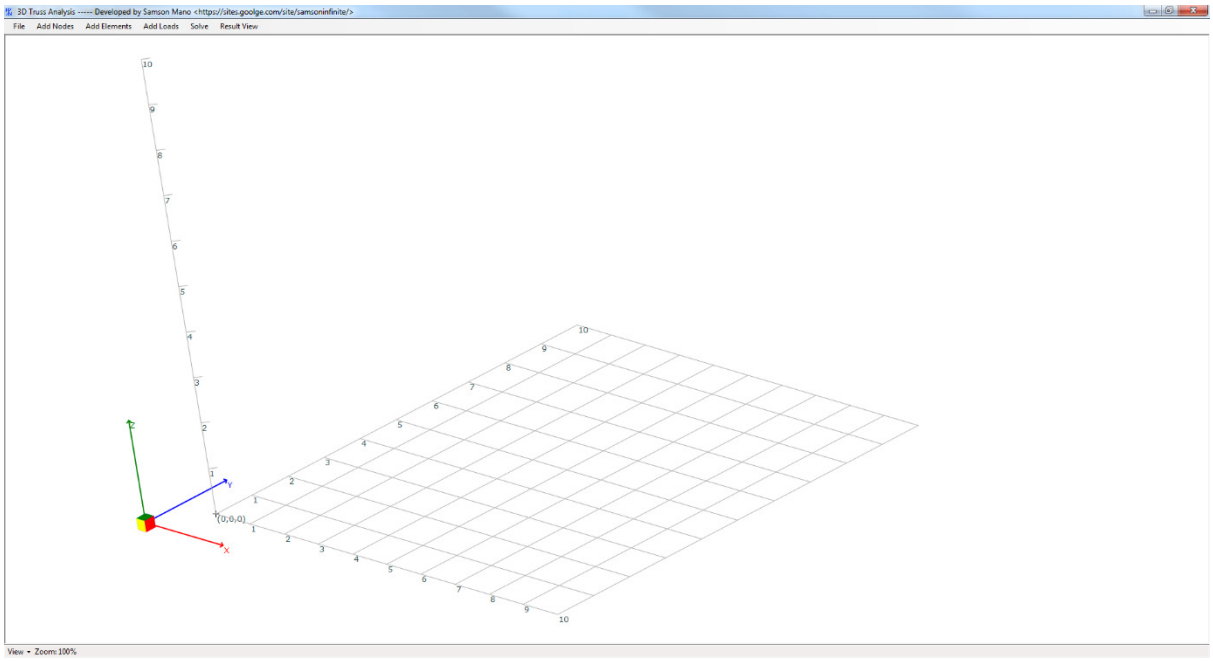
I have deduced the following nodal co-ordinates

Node	x	y	z
1	-37.5	0	200
2	37.5	0	200
3	-37.5	37.5	100
4	37.5	37.5	100
5	37.5	-37.5	100
6	-37.5	-37.5	100
7	-100	100	0
8	100	100	0
9	100	-100	0
10	-100	-100	0

Open the Spacetrussanalyzer.exe

Some basic controls to remember

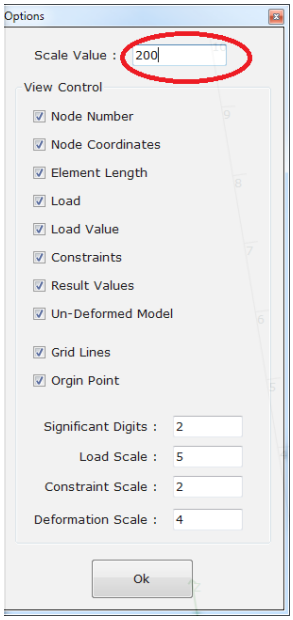
1. Scroll to zoom in and zoom out the view
2. Click and drag using mouse left button to rotate the view
3. Click and drag using mouse middle button to Pan the view
4. File -> save at any time to save the model



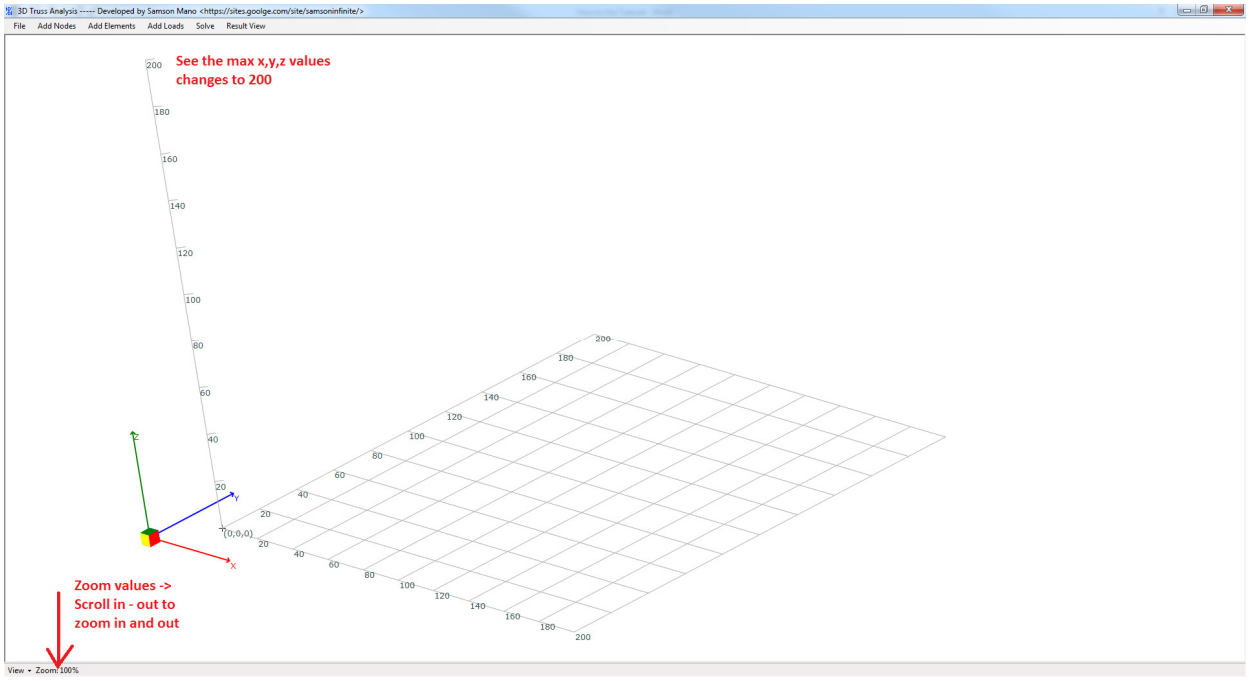
Goto File -> options

As you can see at 100% zoom the x,y,z axis has 10 units, but our node coordinates goes upto 200 units.

Change the Scale Value to 200

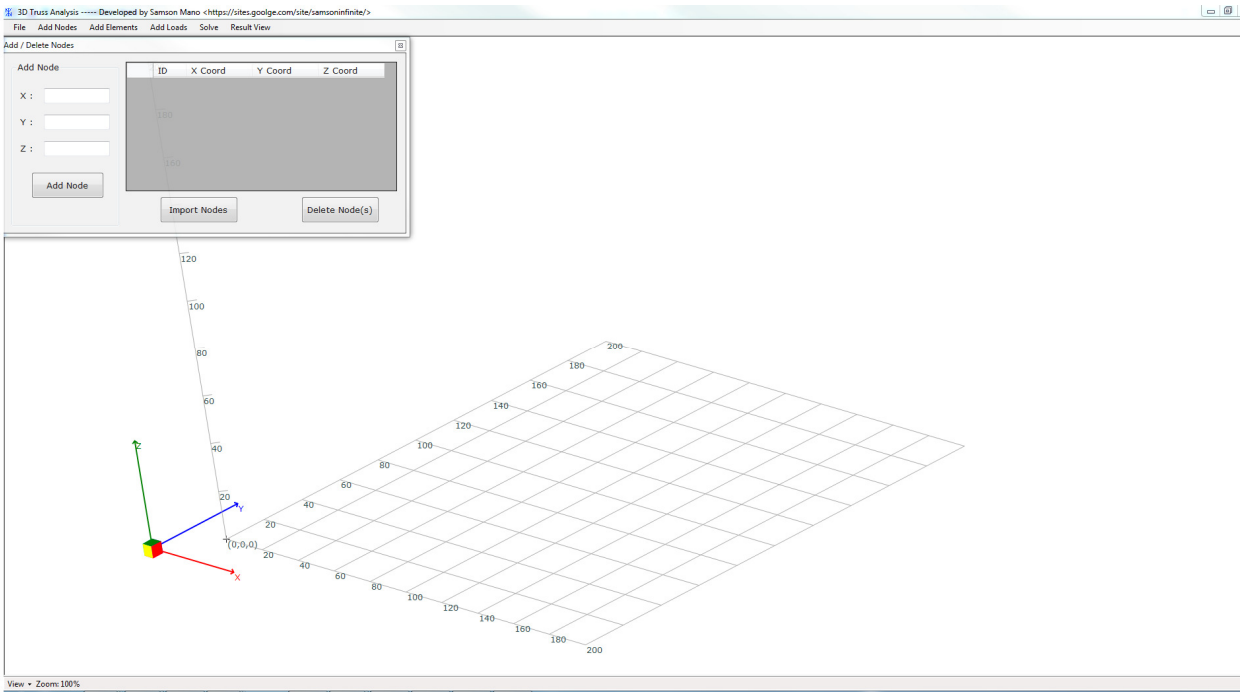


Now the main window becomes

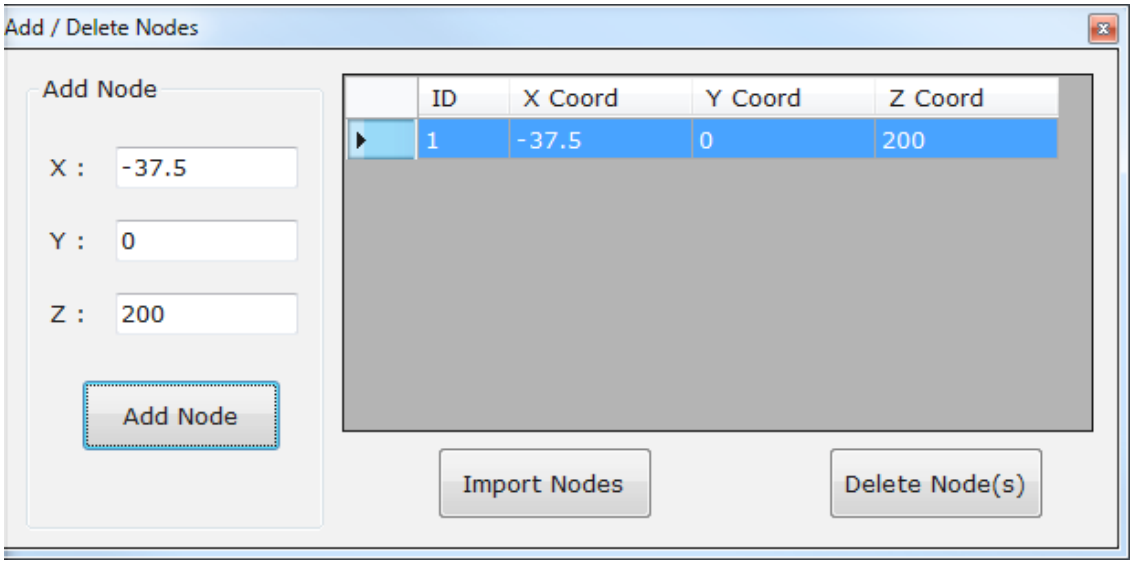


Step: 2 Adding Nodes

Click on the Add nodes button in the Menu strip

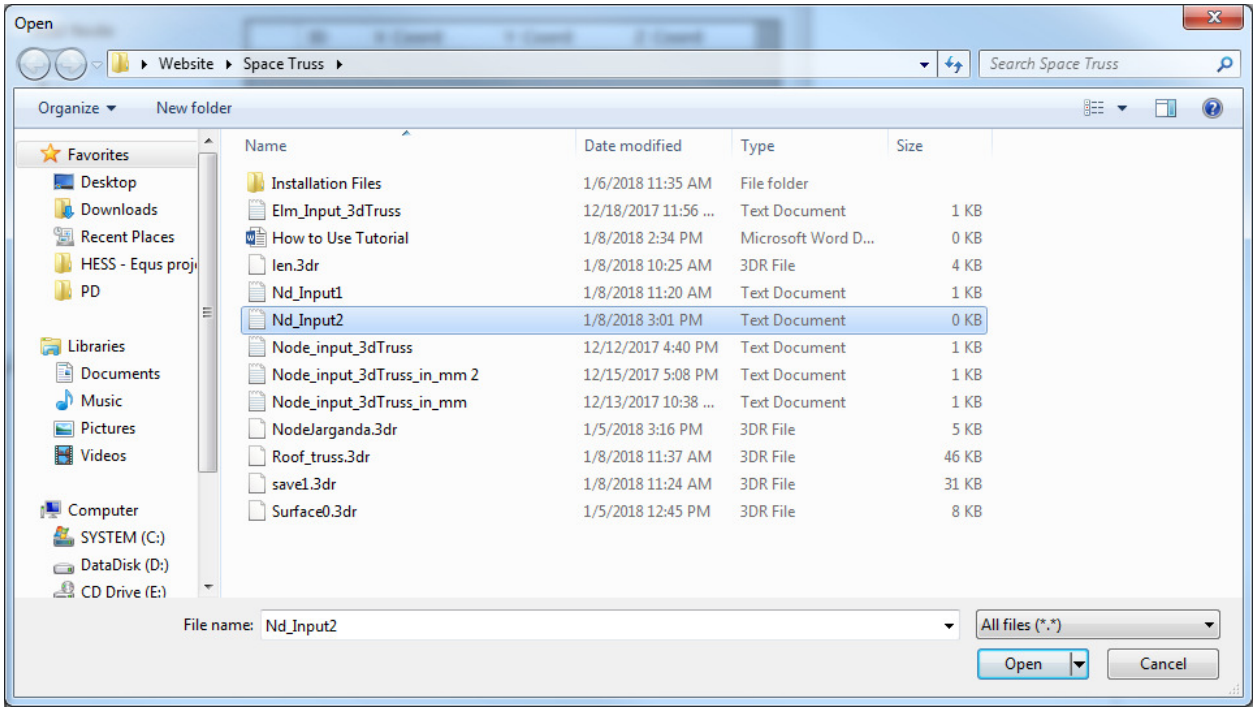


You can add each nodes by typing the coordinates and pressing Add Node button

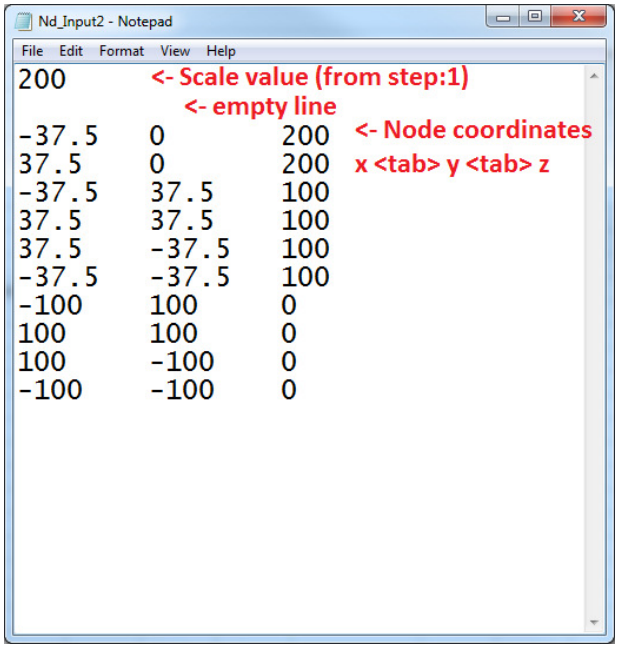


[Or]

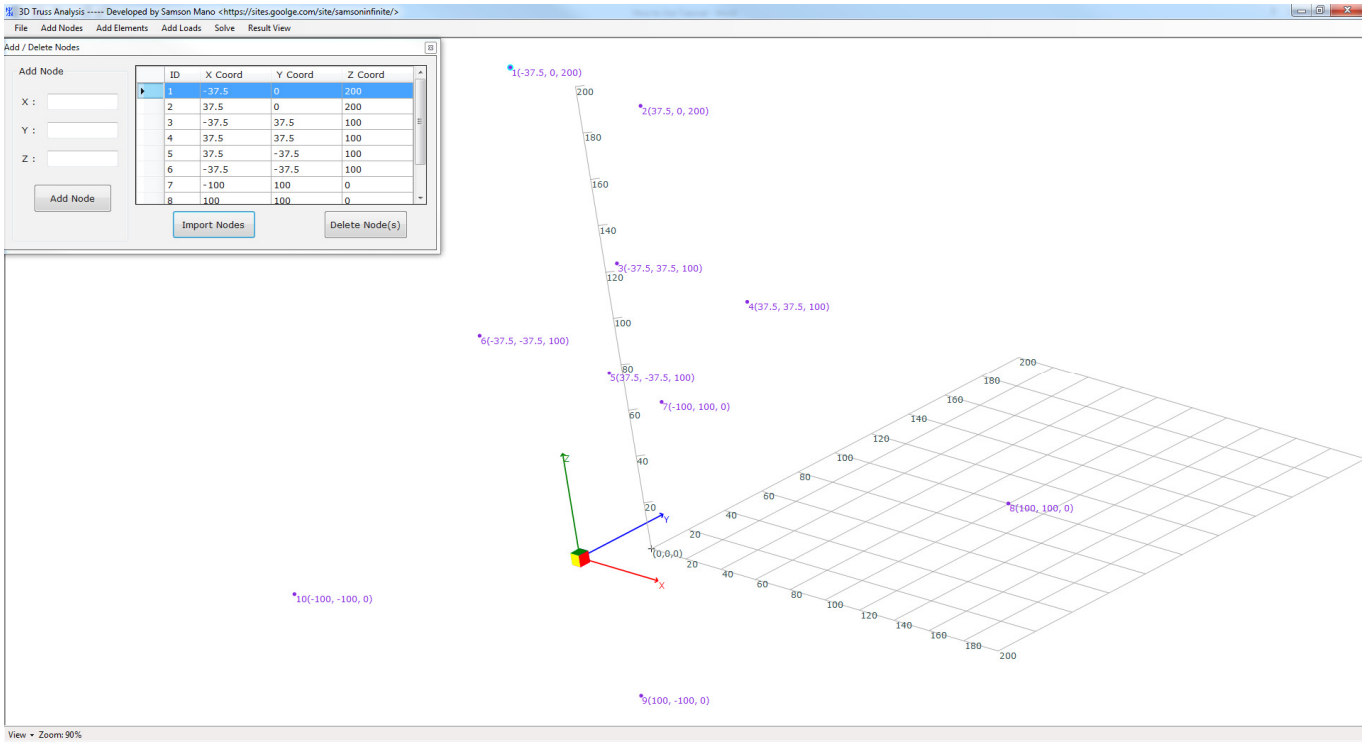
You can use Import Nodes button and import the nodes from a Notepad file.



These are the input format



After the addition of nodes the window looks like this



Note: 1

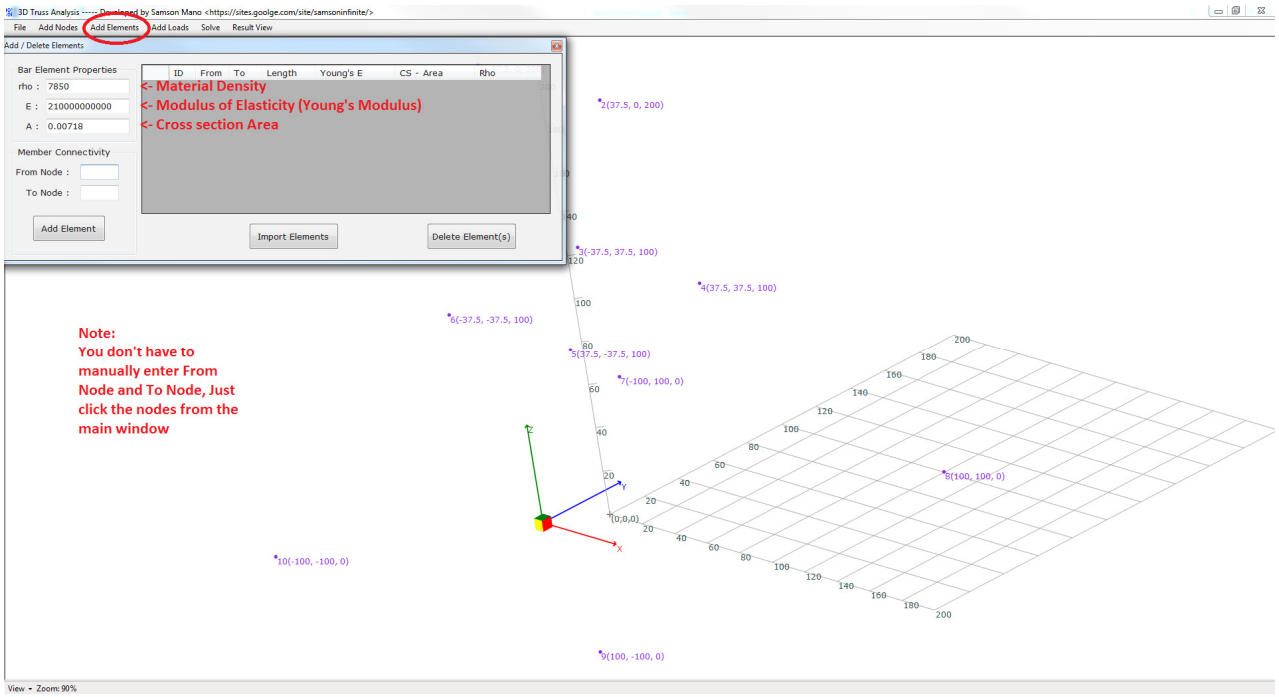
Move the cursor over the node to highlight the node and click on the node to select the node (shift + click allows you to multi select nodes). Alternatively you can select the nodes by selecting from the data grid view list. Selecting the nodes allows you to delete it.

Note: 2

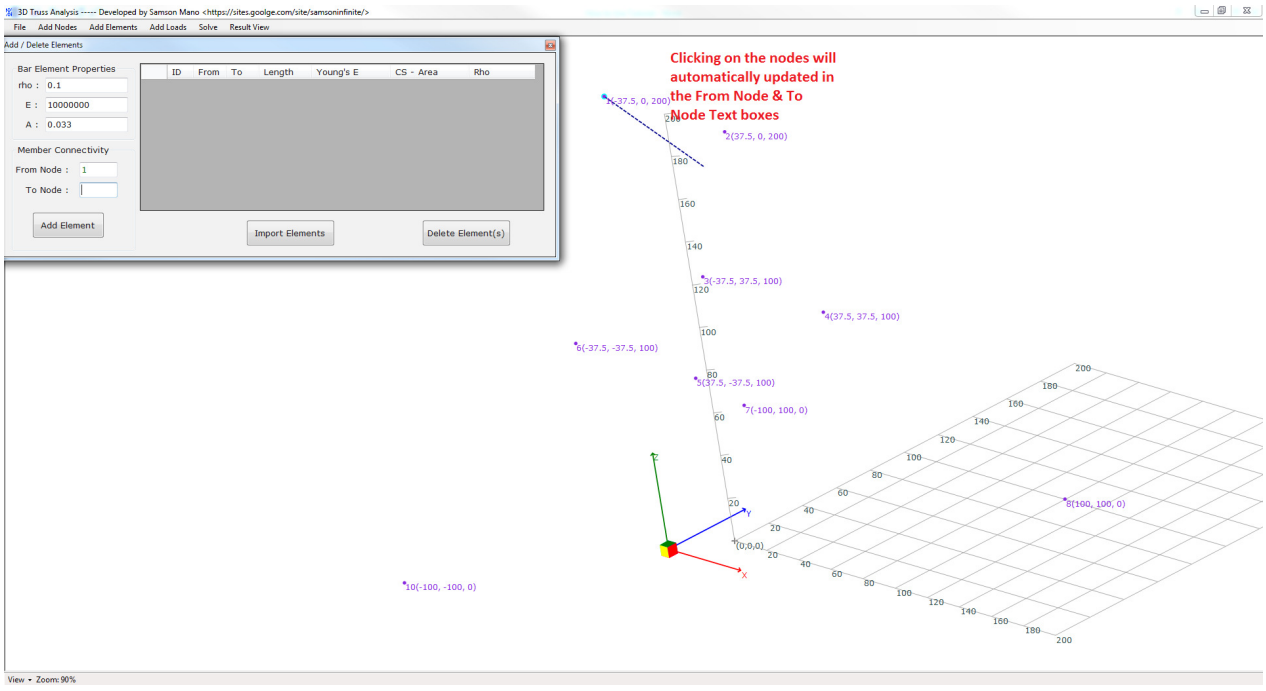
You cannot delete the nodes which are connected to element. Delete the element first and delete the node.

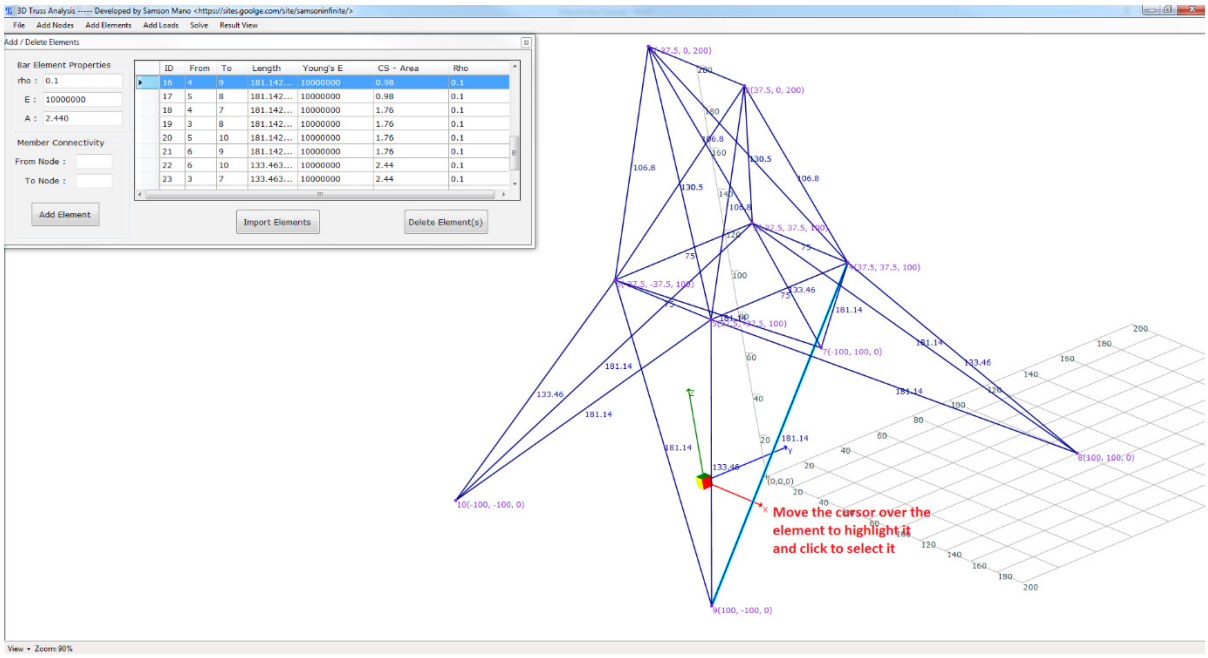
Step: 3 Adding Elements

Click on Add Elements button in menu strip



You can manually add elements by clicking the nodes from main window and press Enter

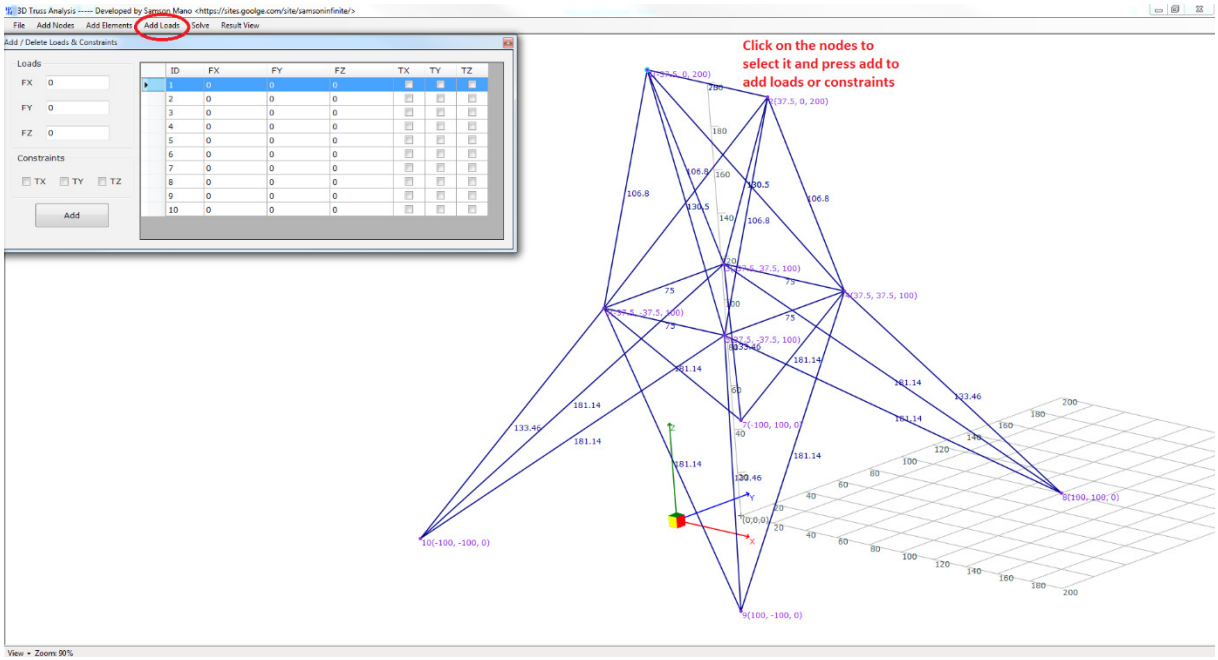




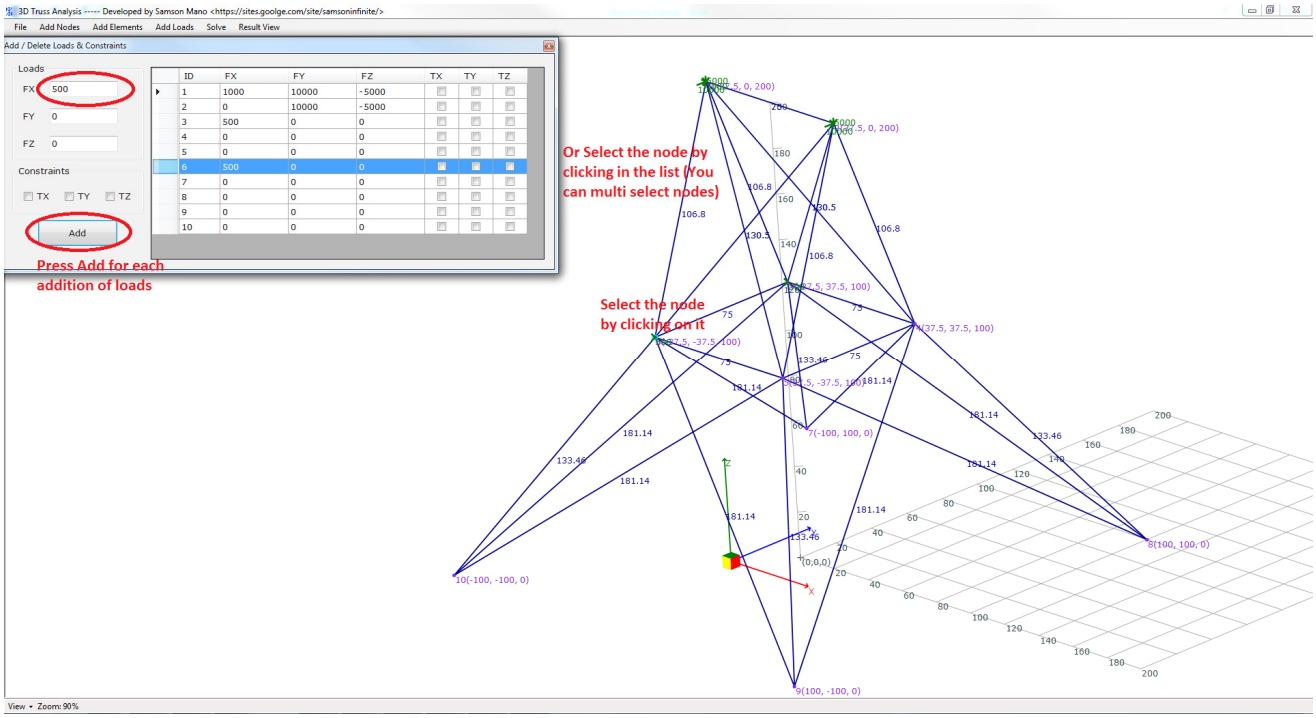
Note: Move the cursor over the element to highlight the element and click on the element to select the element (shift + click allows you to multi select elements). Alternatively you can select the nodes by selecting from the data grid view list. Selecting the element allows you to delete it.

Step: 4 Adding Loads and constraints

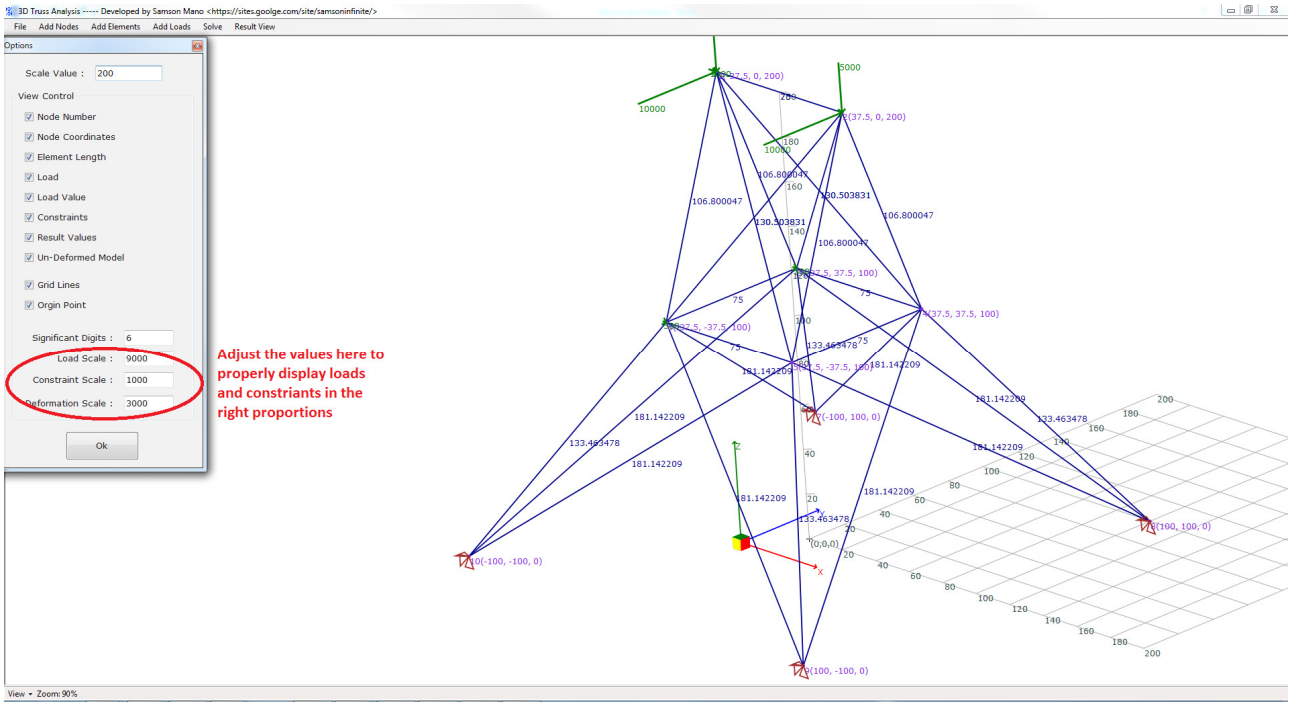
Click on Add Loads button in menu strip



Select the nodes by clicking on it in the main window or multi select nodes from the data grid view list. Add nodes and constraints appropriately

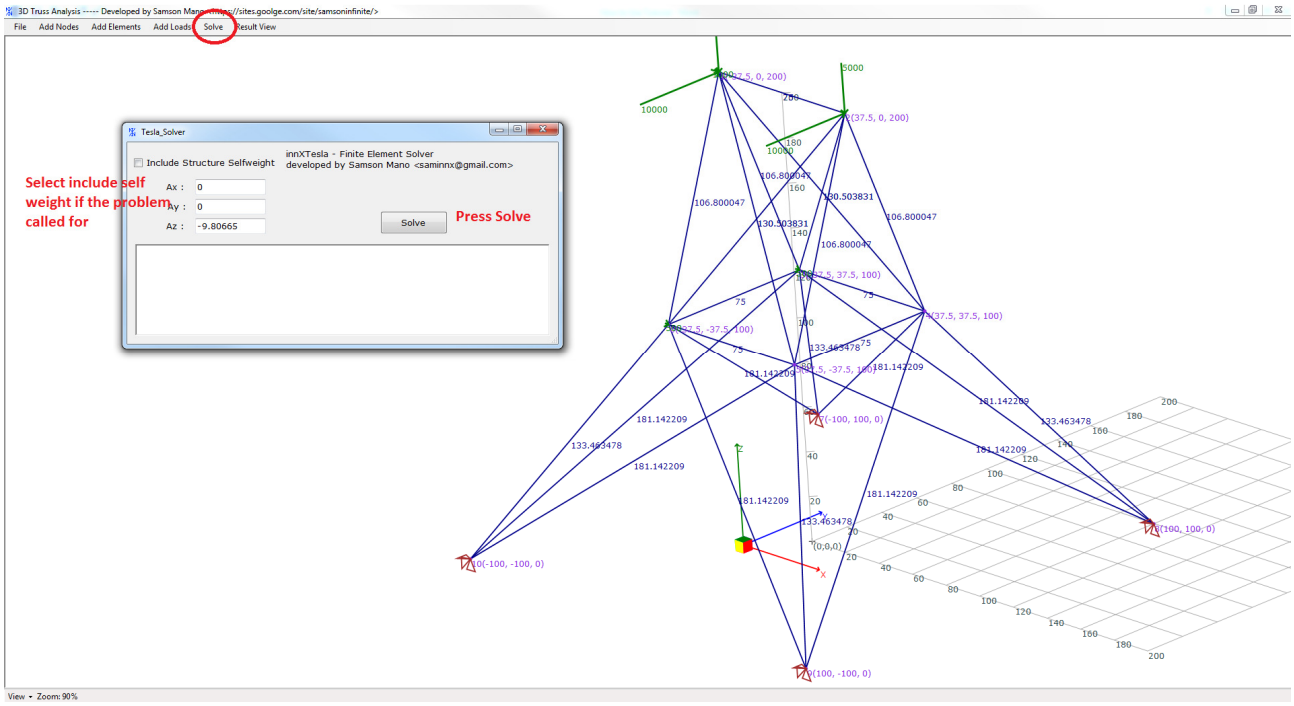


Note: As you can see the loads and constraints are not visible at right proportions in the model. You can modify the scale in which the loads and constraints are displayed by modifying it in File -> Options



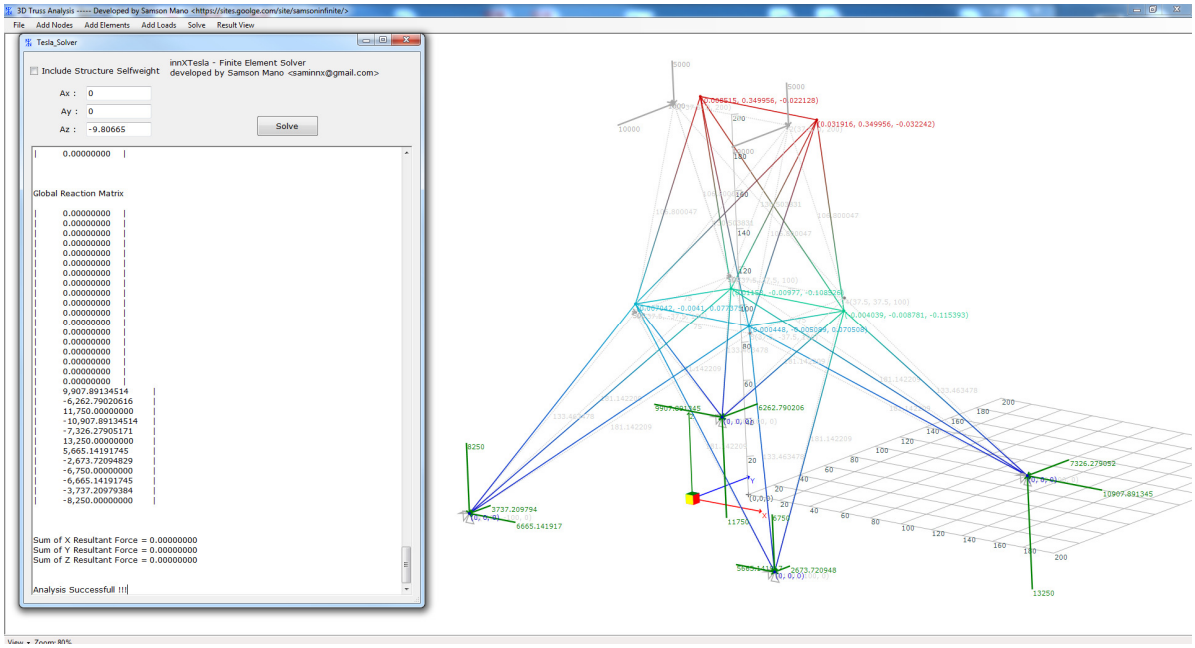
Step: 5 Solving the problem

Click on Solve button in menu strip



Step: 6 Post Processing the results

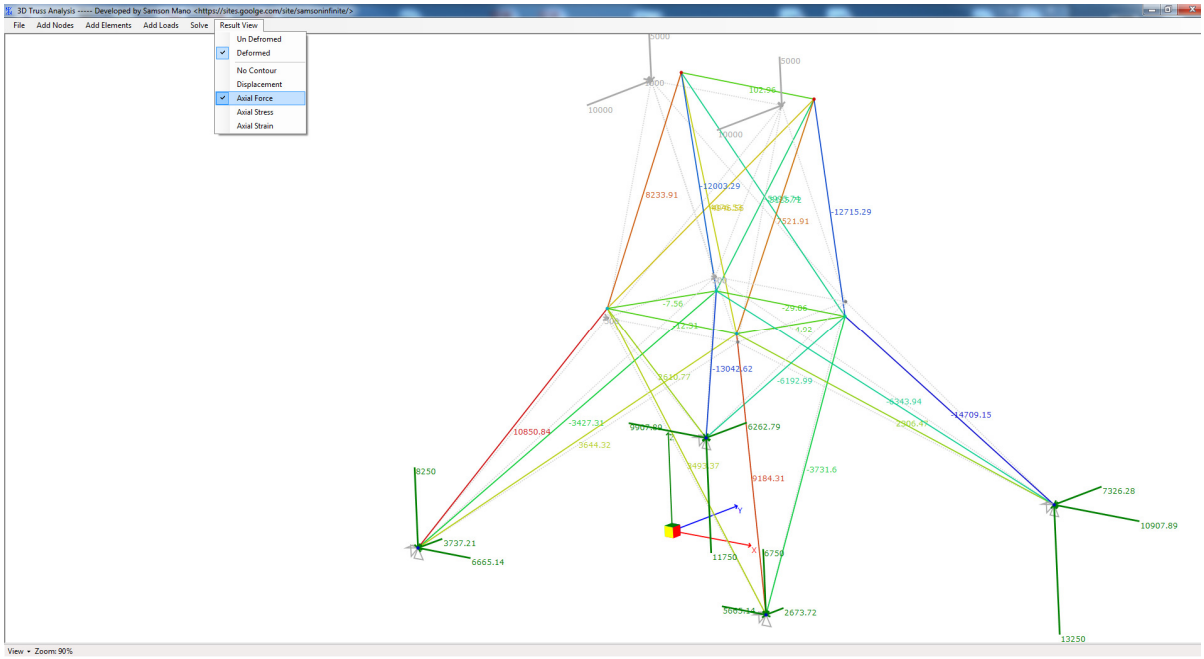
All the Matrices formed for the analysis is displayed. The displacement Contour will be displayed.



In theory, the sum of the resultant forces should be zero for the successful analysis. The solver is validated with Nastran student version.

Click on various result views from menu strip to visualize various results. Also change the option from file-> options to suite your need.

The member force view is shown below



Conclusion:

This software is free & validated. Just copy the spacetrussanalysis.exe file and it is good to run in any windows platform. I used VB.Net 2010 express edition to create this application.

Feel free to send me a mail saminnx@gmail.com if you found any bug or improvement ideas or any custom software solution requirement.

Visit my website: https://sites.google.com/site/samsoninfinite/

-

S Mano