How to use Plane Truss Finite Element Solver

Developed by Samson Mano

email: saminnx@gmail.com

Github: https://github.com/Samson-Mano/

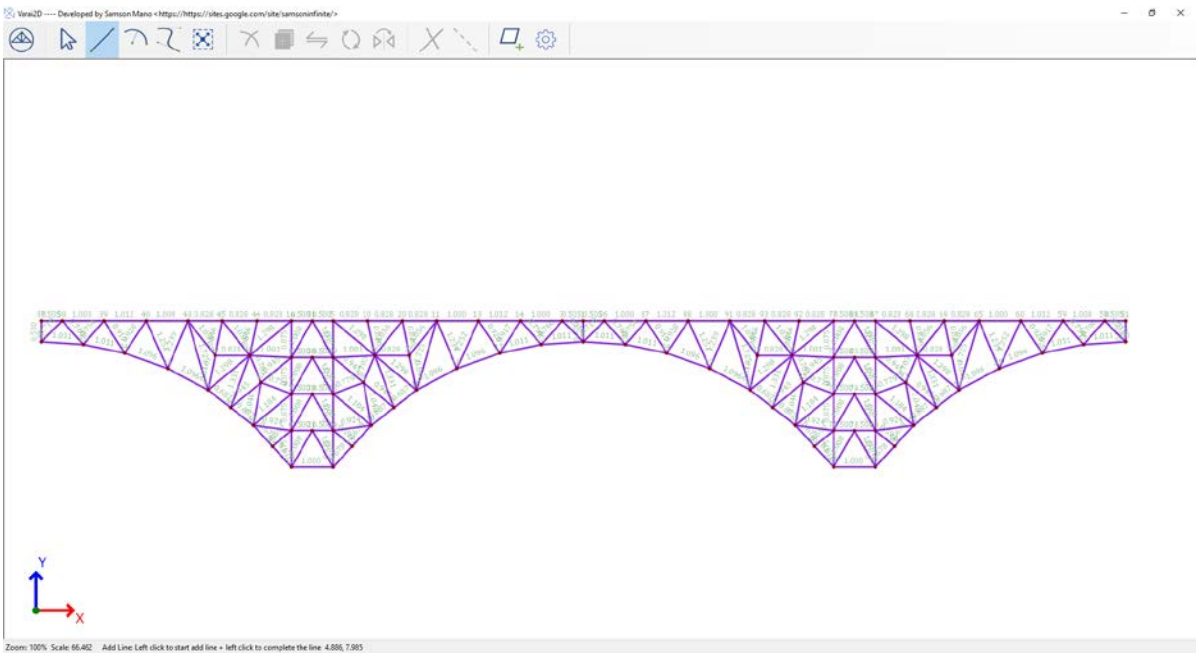
Content

1. Model creation using Draw2D Geometry

The first step is to create the 2D line geometry model. You can download the Draw2D geometry software from this link:

https://github.com/Samson-Mano/Draw2D_geometry/tree/main/Varai2D_portable

It is very easy to create the 2D model using this application.



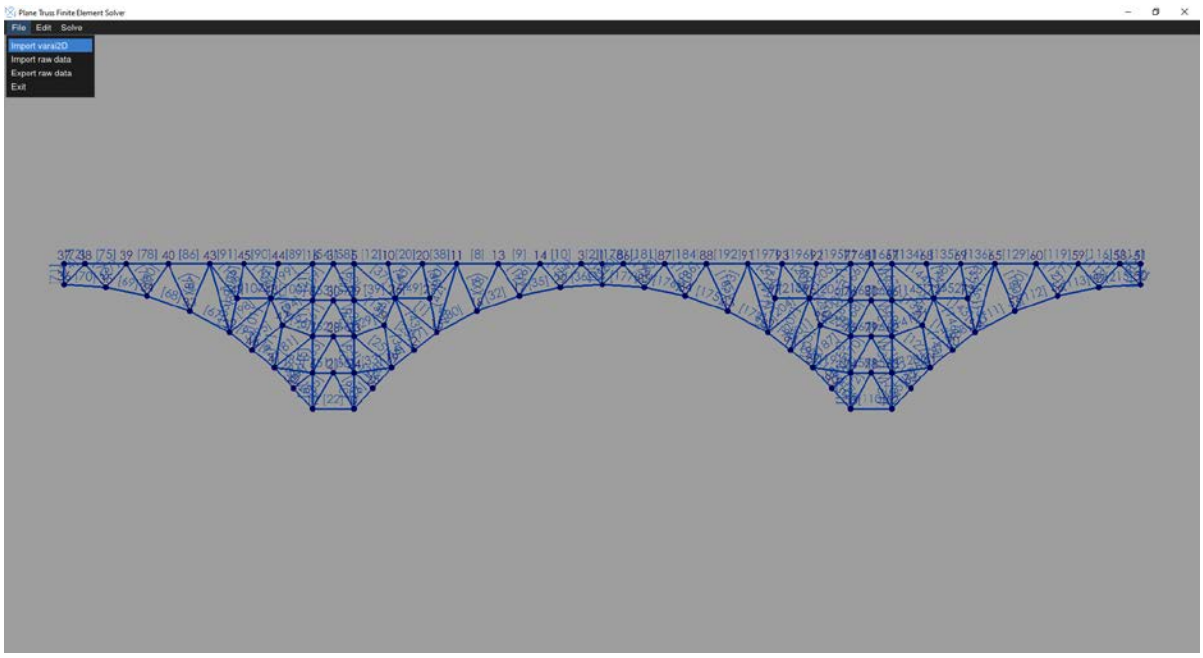
There is also other way to create the model using text inputs which can be imported as raw text data.

2. Model import and model view basics

Important GUI commands to note are,

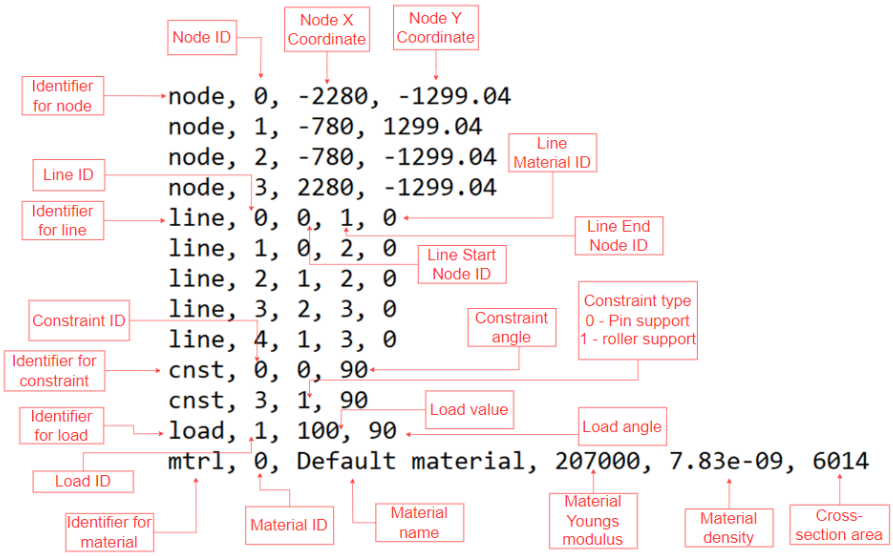
- Zoom in/ out => Ctrl + mouse scroll
- Pan operation => Ctrl + mouse right drag
- Zoom to fit => Ctrl + F

Use *File -> Import Varai2D* to open the model in the solver.

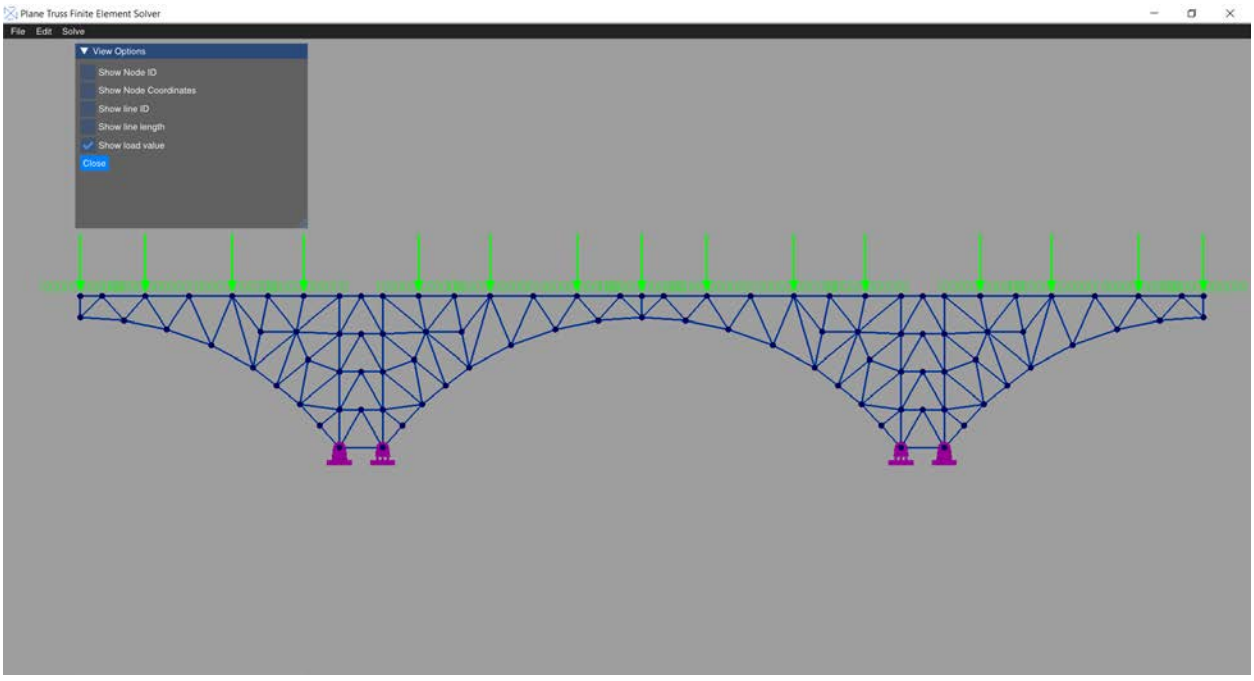


In-order to create model using text inputs use the following format and open the model using *File -> Import raw data*

Below is the definition of values for the raw data input

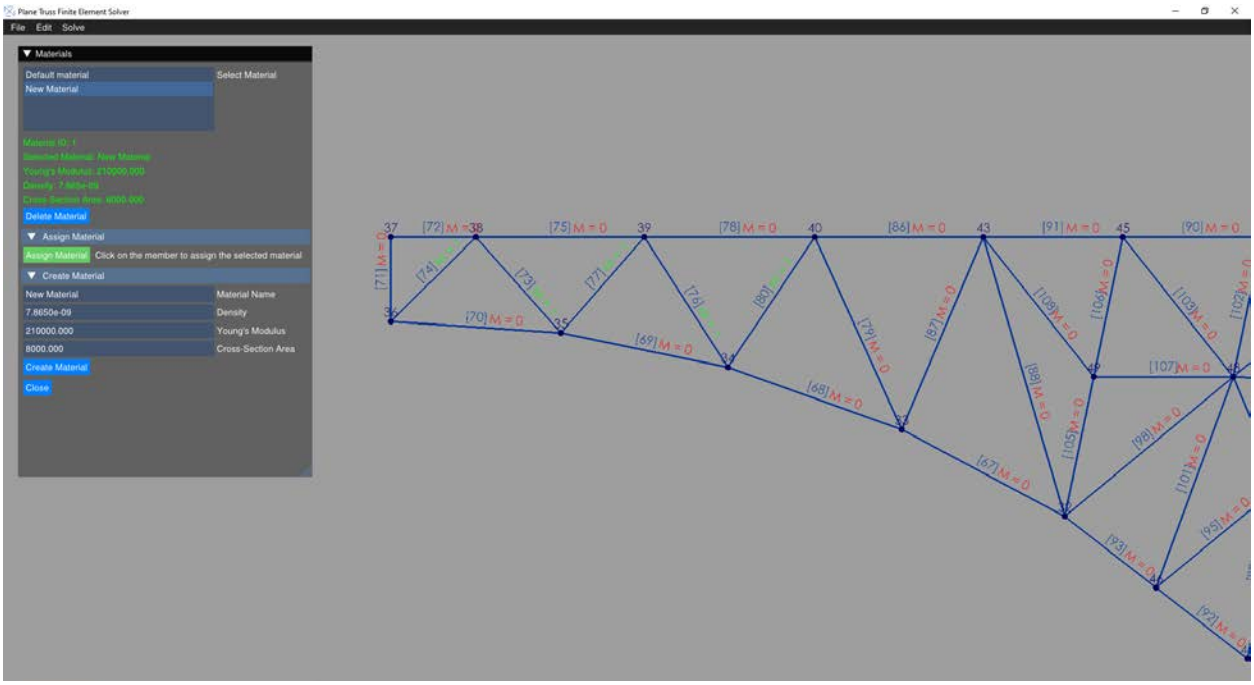


Use *Edit -> view options* to change the model view setting.



3. Assign material property

Use *Edit -> material* to update the material properties.



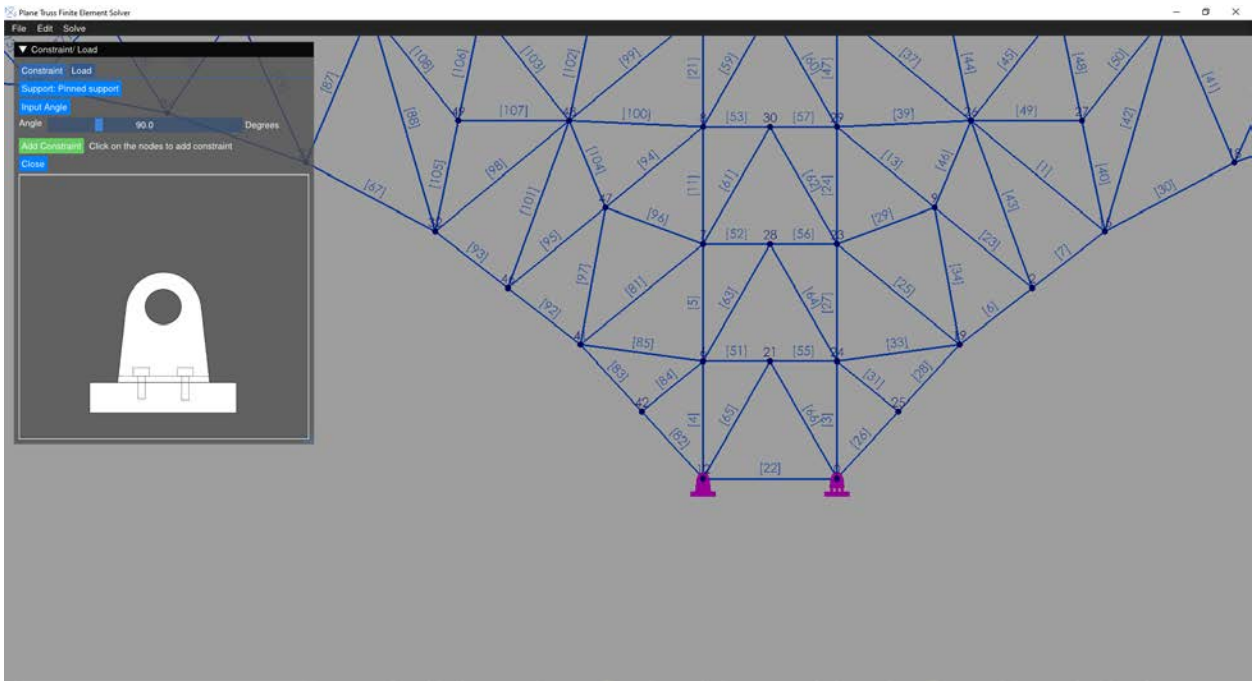
Create material button to create new material.

Select Assign Material button and click on the member to assign the selected material. Material ID will be displayed to show the assigned material properties.

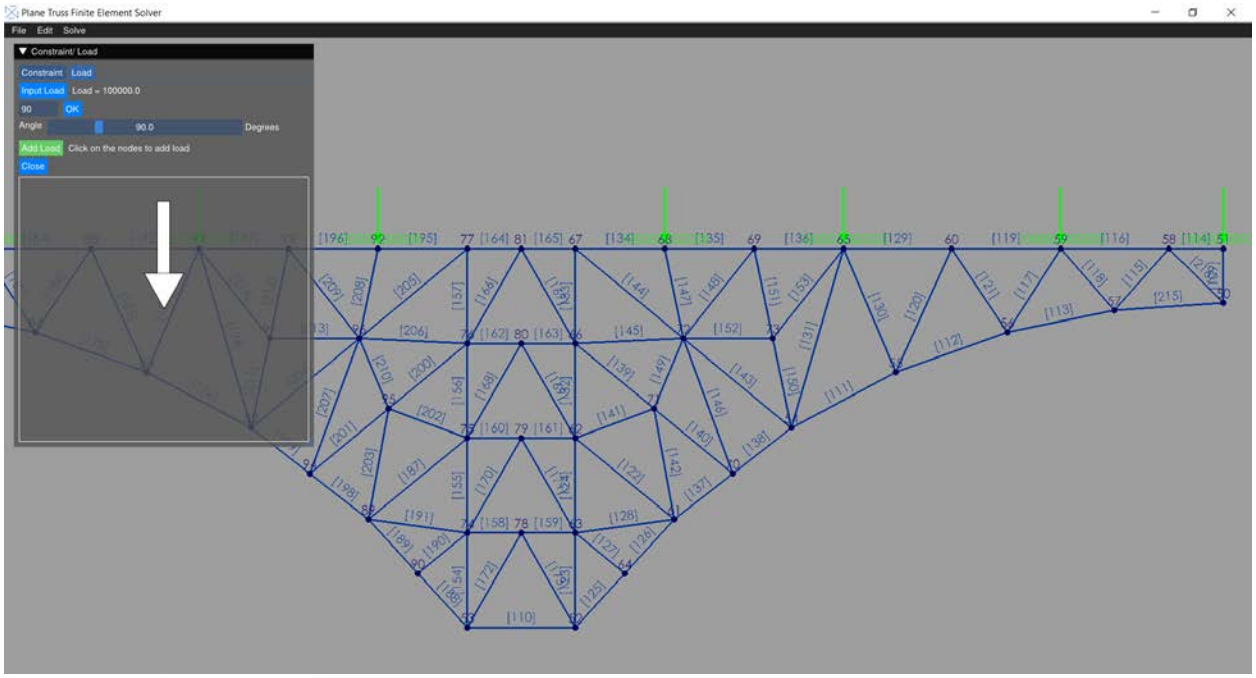
4. Apply boundary condition

Use *Edit -> constraint/ load* to add constraint and loads

Select the type of support, angle of support and Select Add constraint button and click on the node to assign the constraint.

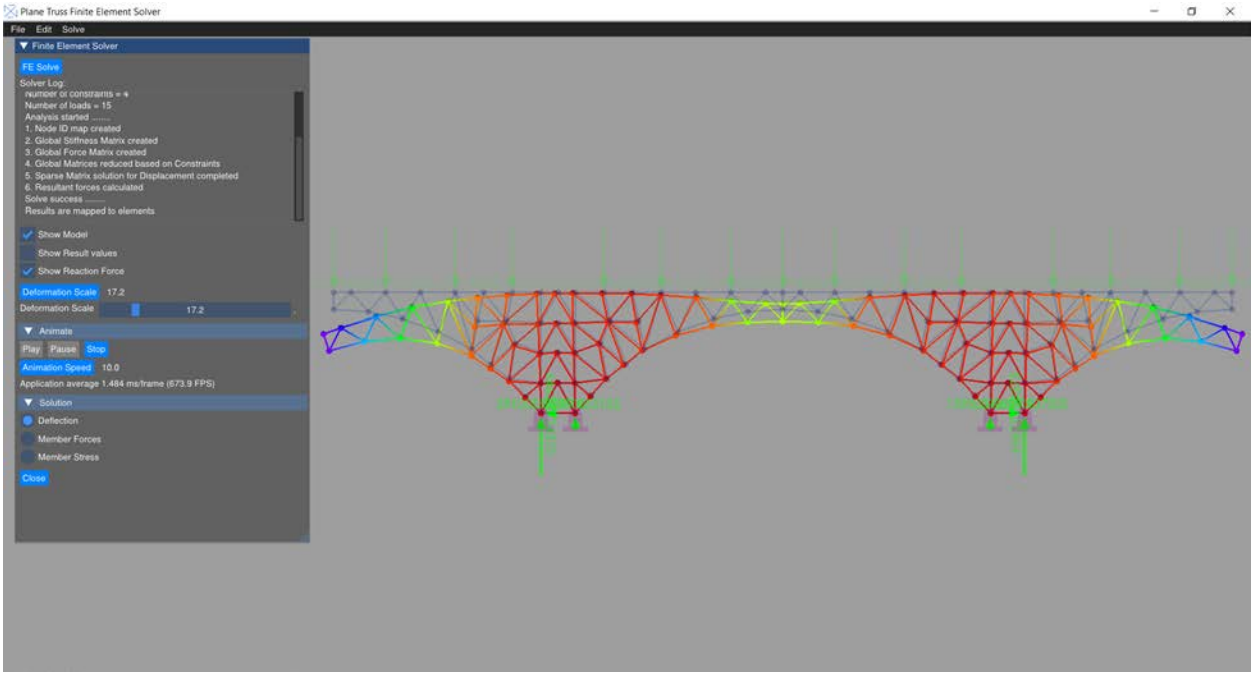


Similarly select the add load button and click on the nodes to add the load.

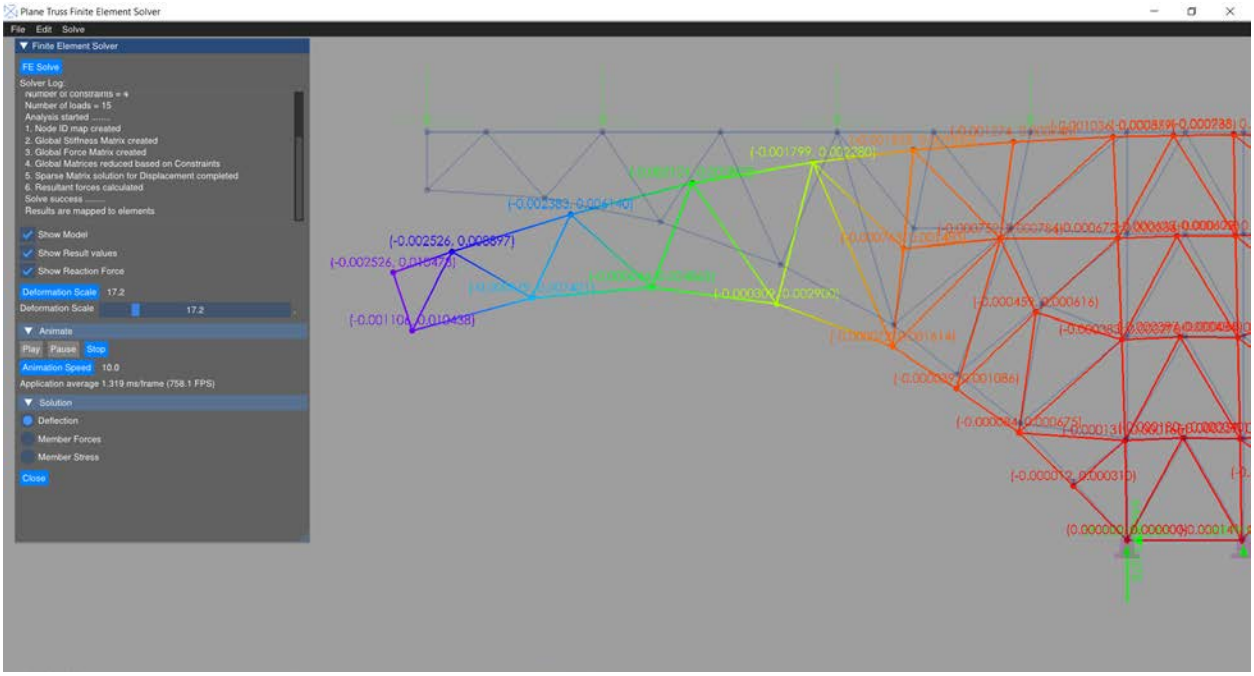


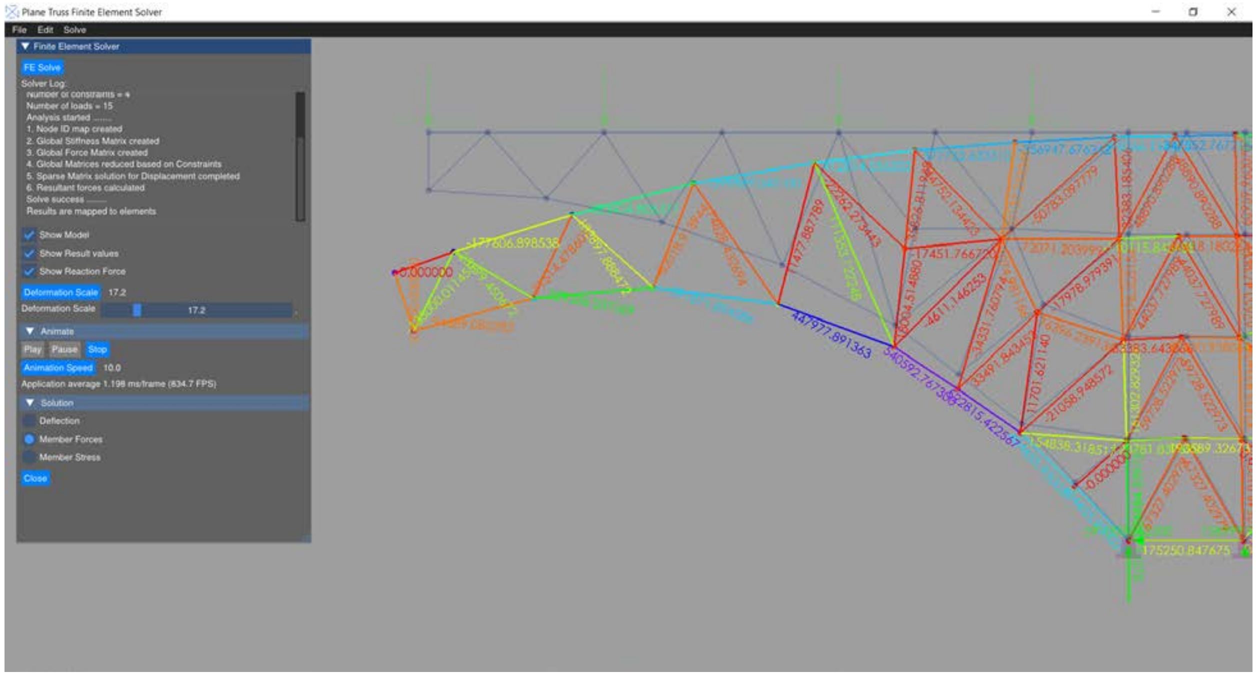
5. Solve and post-processing.

Once the model setup is done use *Solve -> FE Analysis* and click FE Solve button to solve the model.



Use the various options for post processing. For example, selecting Show Result values will show the nodal deflection values. Selecting Member Forces option will show the member force of individual member.




Plane Truss Finite Element Solver
File
Edit
Solve
Finite Element Solver
FE Solve
Solver Log:
Number of loads = 15
Analysis started ....
1. Node ID map created
2. Global Stiffness Matrix created
3. Global Force Matrix created
4. Global Matrices reduced based on Constraints
5. Sparse Matrix solution for Displacement completed
6. Resultant forces calculated
Solve success ....
Results are mapped to elements
Show Model
Show Result values
Show Reaction Force
Deformation Scale
17.2
Deformation Scale
17.2
Animate
Play
Pause
Stop
Animation Speed
10.0
Application average 1.198 ms/frame (834.7 FPS)
Solution
Deflection
Member Forces
Member Stress
Close
-177606.898538
447977.891363
175250.847675

- END -