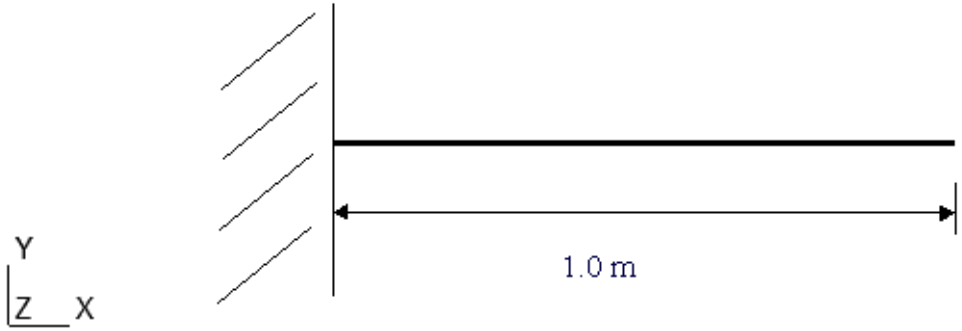
IngentaCal

Engineering Solutions

SCIFESOL Tutorial : Transient Analysis of Beam

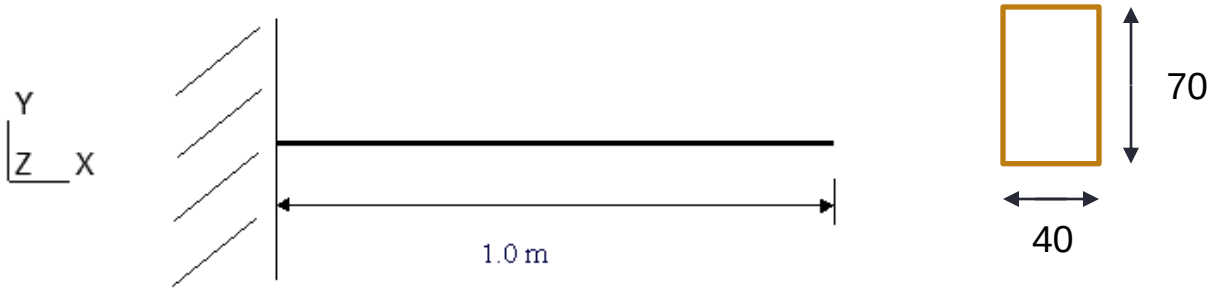
Problem Description

Transient analysis of a cantilever beam having uniform cross section is performed to calculate the dynamic response from time varying load.



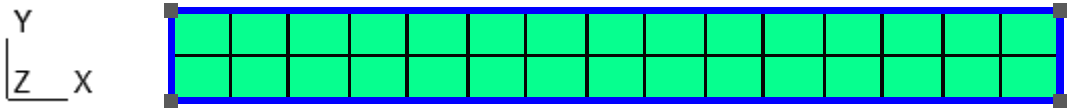
Geometry

A cantilever beam of length 1m , having rectangular cross section is studied.



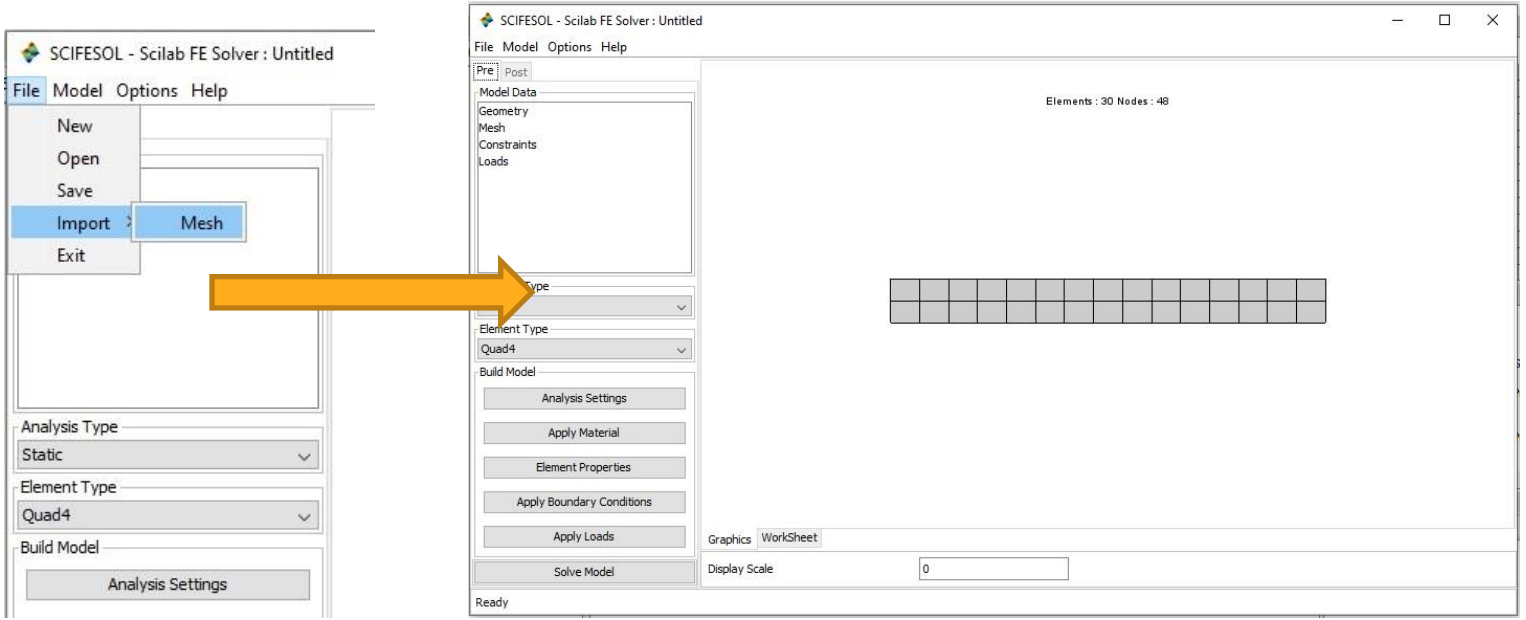
Mesh Generation

We model the beam using 2D plane stress quadrilateral element.



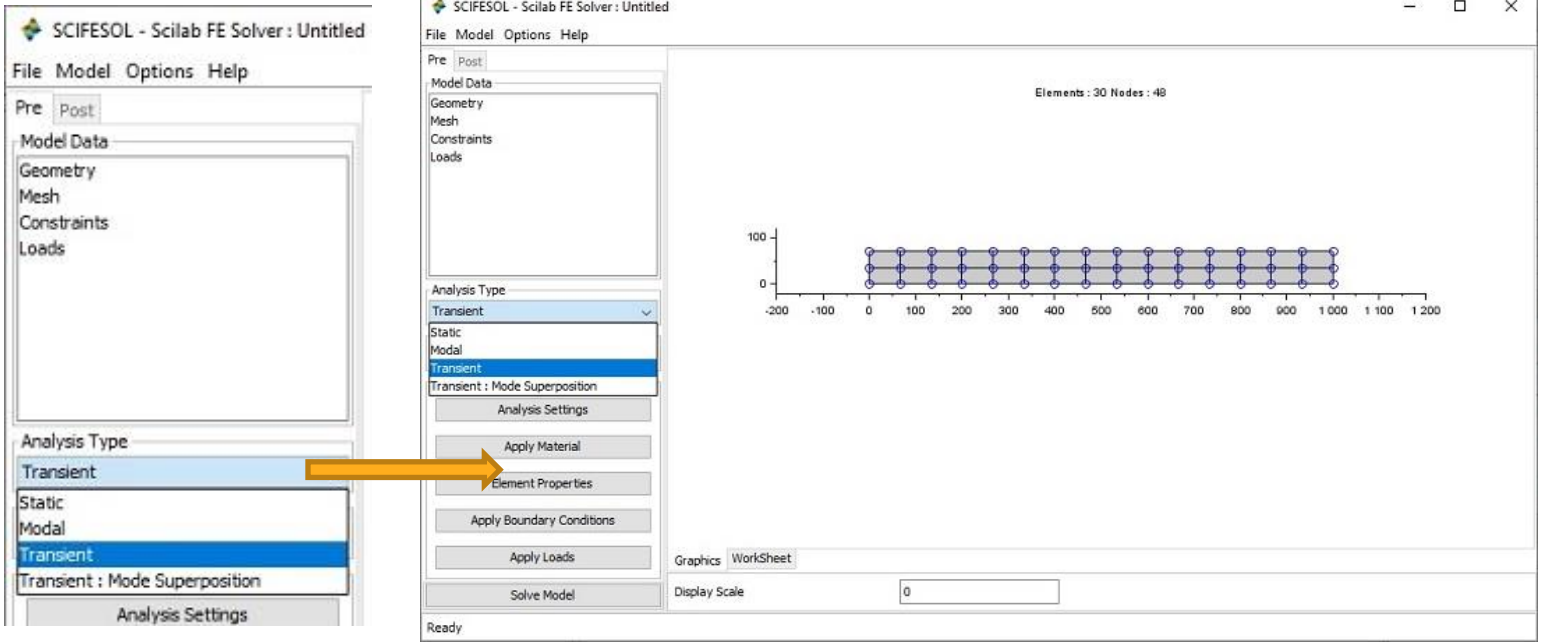
Import Mesh

- Now import the mesh file saved in file 2DPlaneBeam.m which is exported from GMSH.



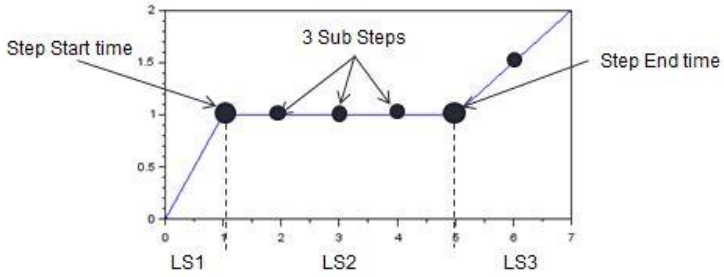
Define Analysis Settings

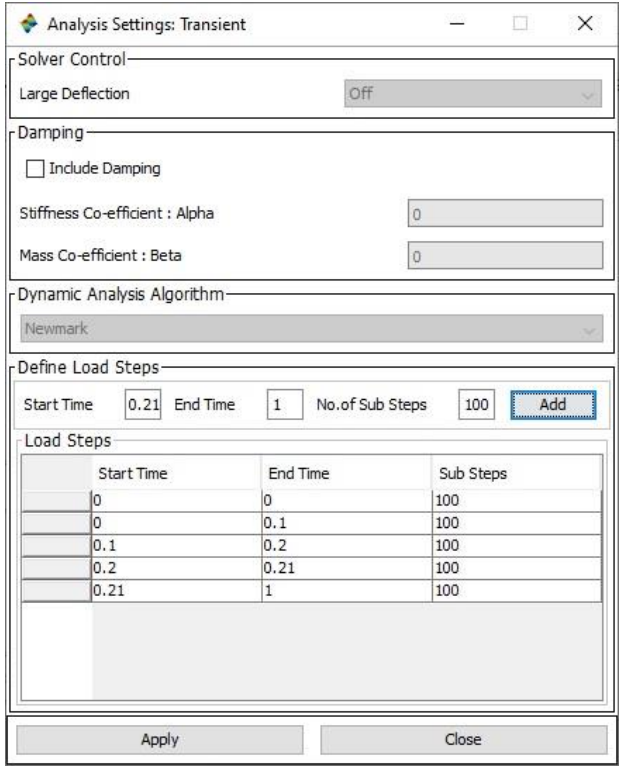
- Select analysis type to Transient.



Define Analysis Settings

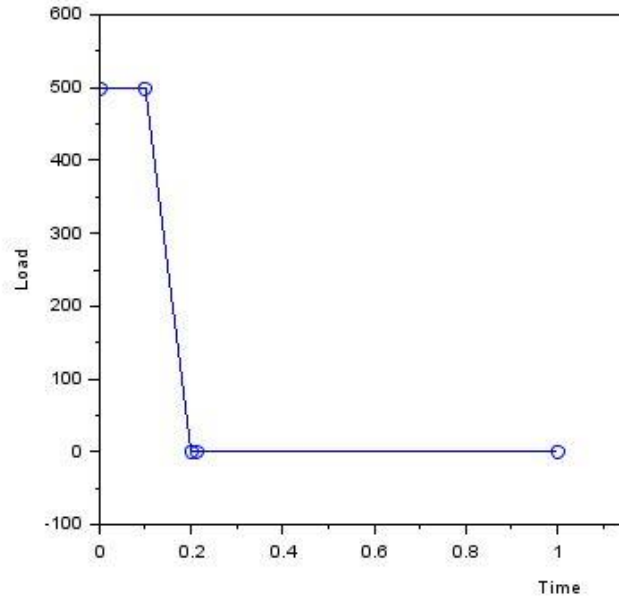
- Analysis Settings > Define Load Steps
- Load steps is used to define loading conditions for time varying loads.
- We can also specify Rayleigh damping coefficients to include damping.
- Newmark time integration is used as solver.

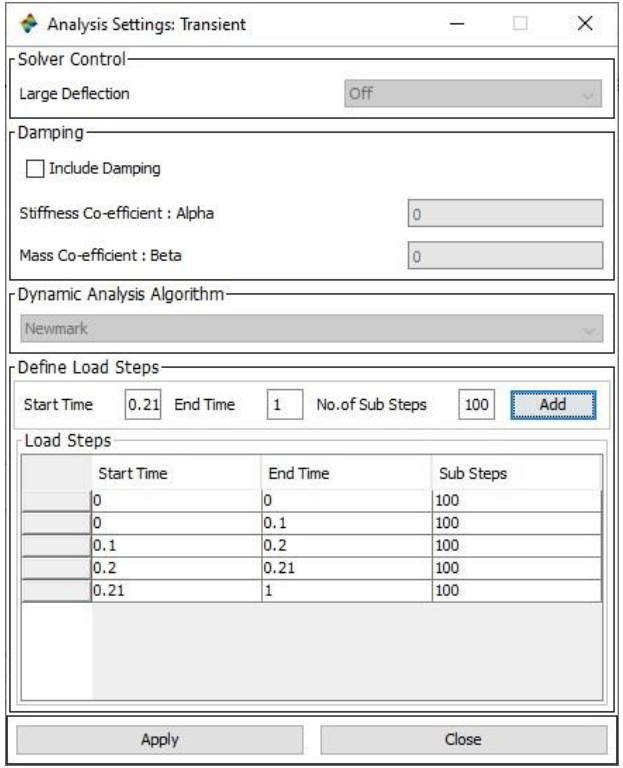




Define Analysis Settings

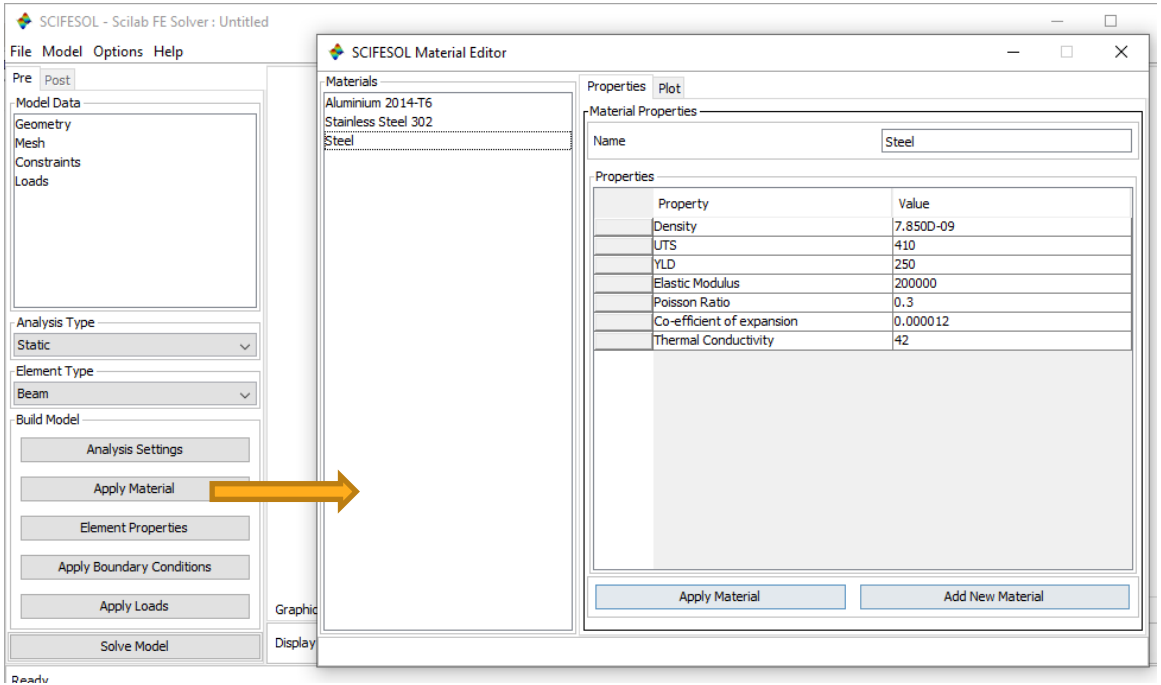
- Load steps are defined corresponding to the loading graph.





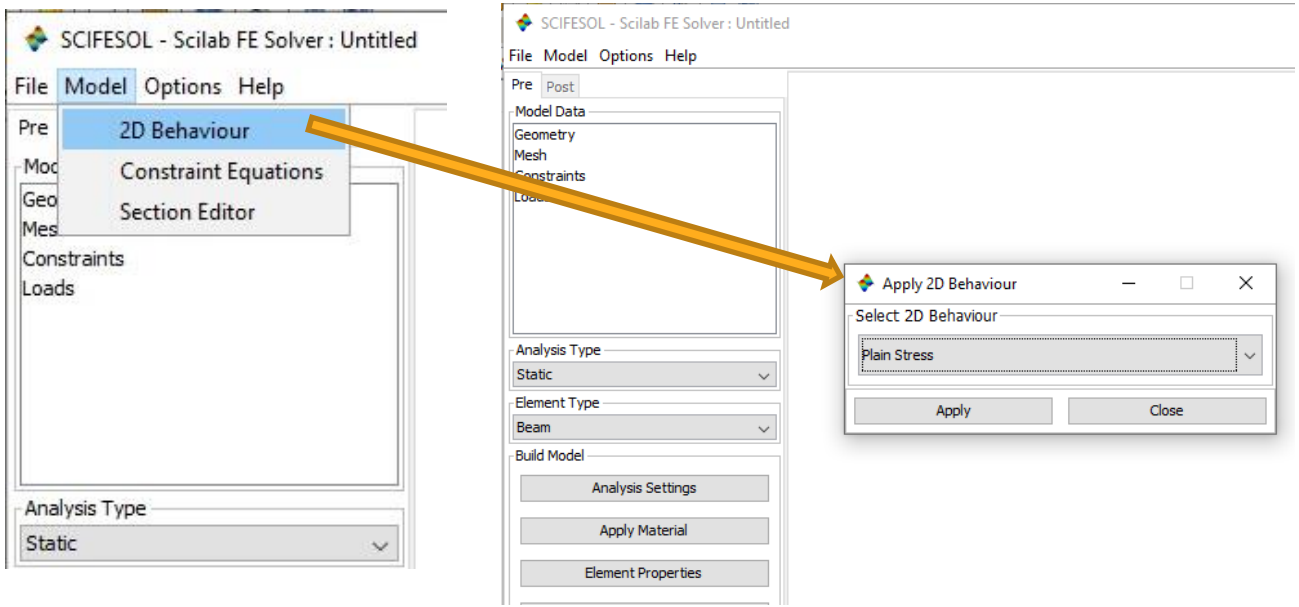
Define Material Properties

- Select material of beam from material editor. We can add new material if other material is required.



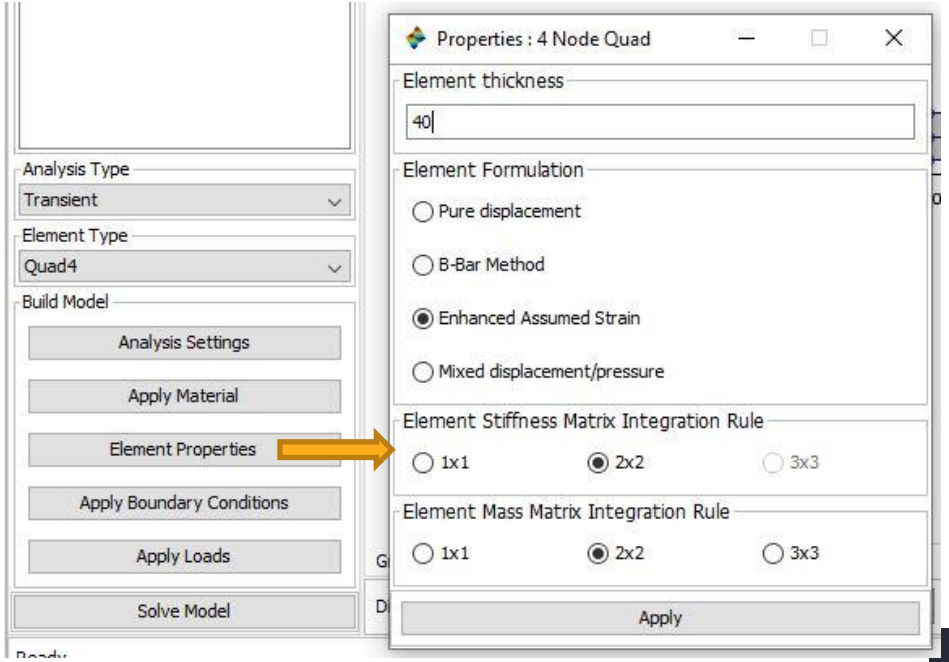
Define Element Properties

- We define the behavior of 2D 4 node quadrilateral elements as plane stress.



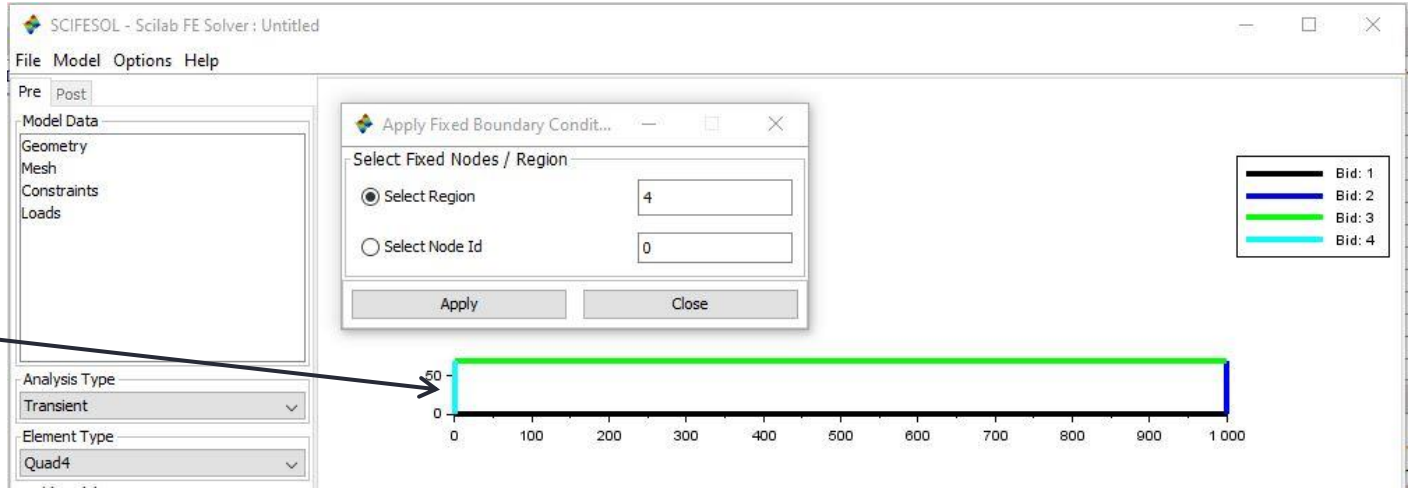
Define Element Properties

- Now we define the formulation of quadrilateral elements as Enhanced Assumed Strain
- Select the stiffness and mass matrix integration rule as 2x2.
- Specify the thickness as 40 mm.



Apply Boundary Conditions

The beam is fixed at its left end using boundary ID 4.

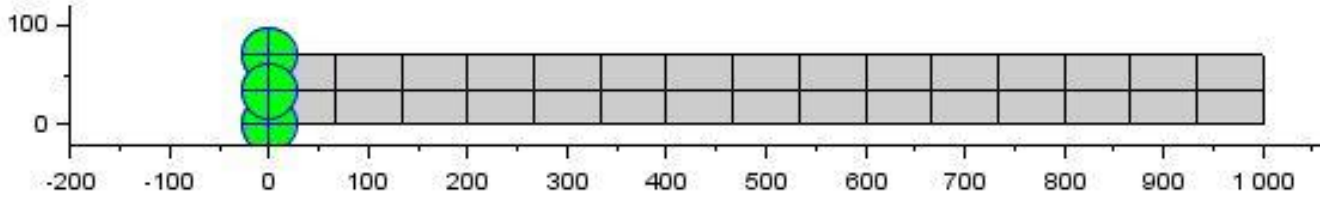


Left end fixed

Identify the boundary ID of the hole region from geometry link in the model data list.

Apply Boundary Conditions

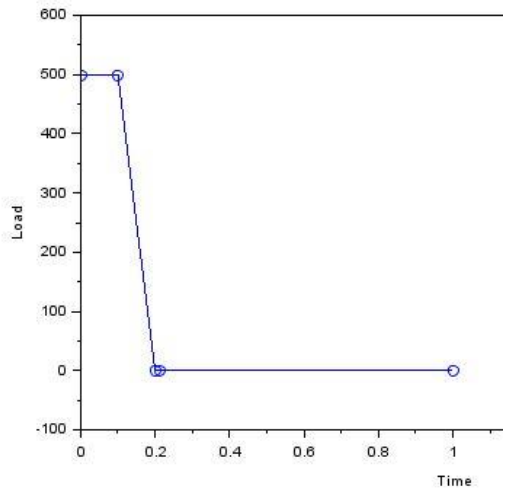
Fixed boundary condition is applied at the left boundary.

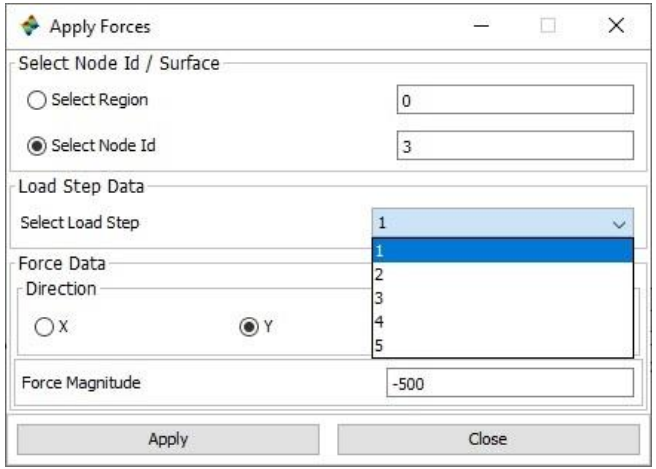


- After applying the boundary condition , first click on the mesh link in the model data list to activate the mesh then click on the constraints or loads link to display the constraints.

Apply Loads

- Time varying load is applied at the corner of the beam at node Id 3.
- We apply load of 500 N for time step 1 and 2 then force is reduced to zero.
- After applying the force we can cross check the FORCES array in the Scilab console to check the applied load.



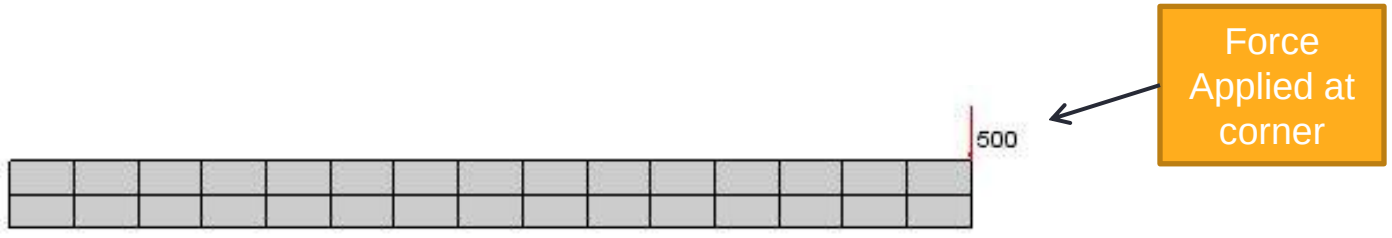



```
--> FORCES
FORCES  =

   3.    2.    0.    -500.
   3.    2.    0.1   -500.
   3.    2.    0.2    0.
   3.    2.    0.21   0.
   3.    2.    1.     0.
```

Apply Loads

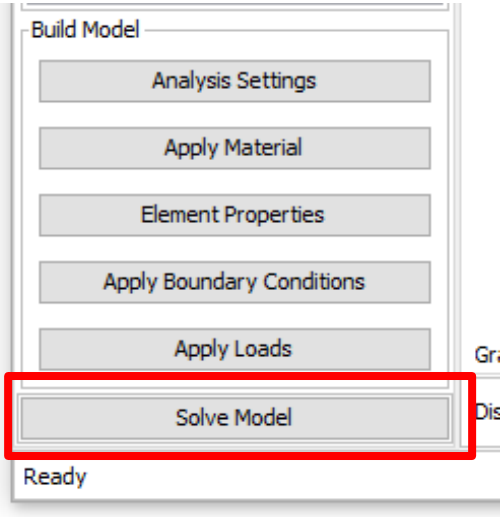
Time varying load is applied at the corner of the beam



- After applying the boundary condition , first click on the mesh link in the model data list to activate the mesh then click on the constraints or loads link to display the constraints.

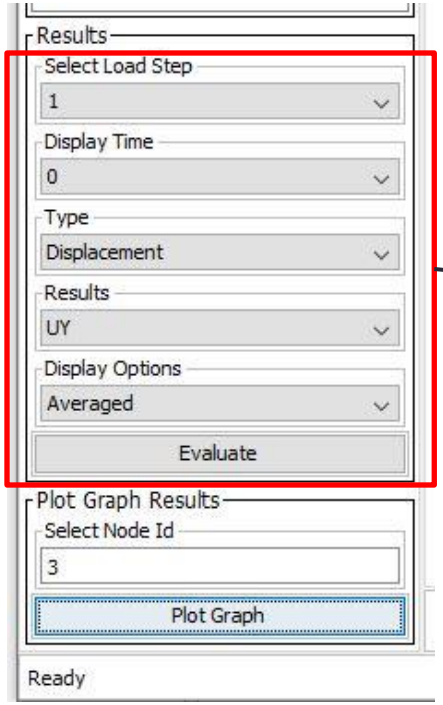
Solve

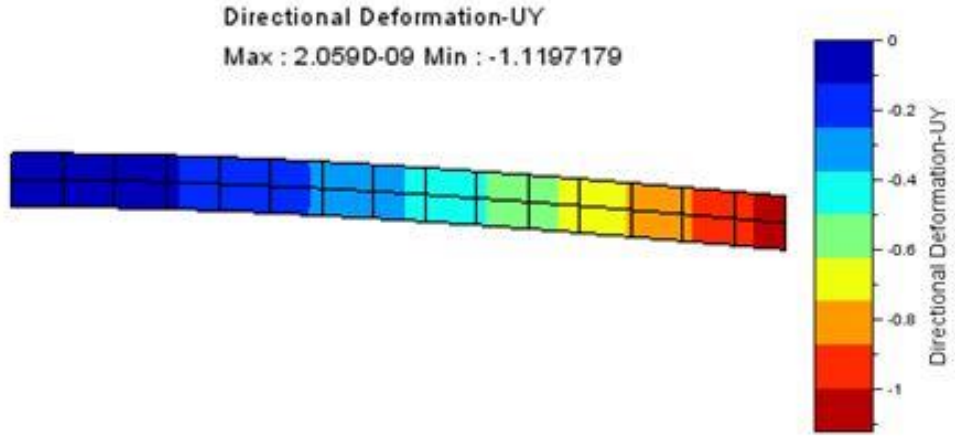
Start solver to solve the model.Due to large no.of time steps , the solving time of the analysis will be higher.



Post Process Results

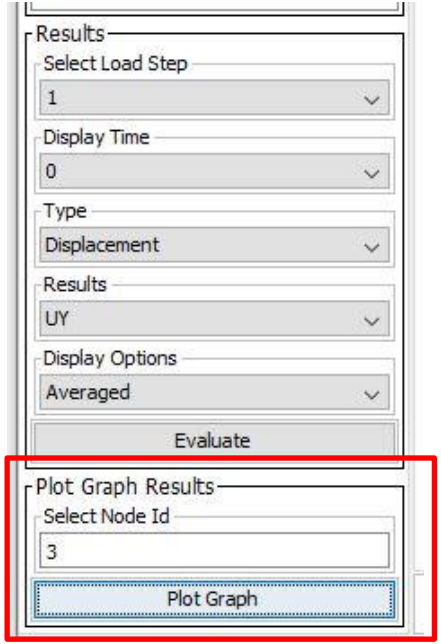
Using the post processor tab, we can plot the required results.

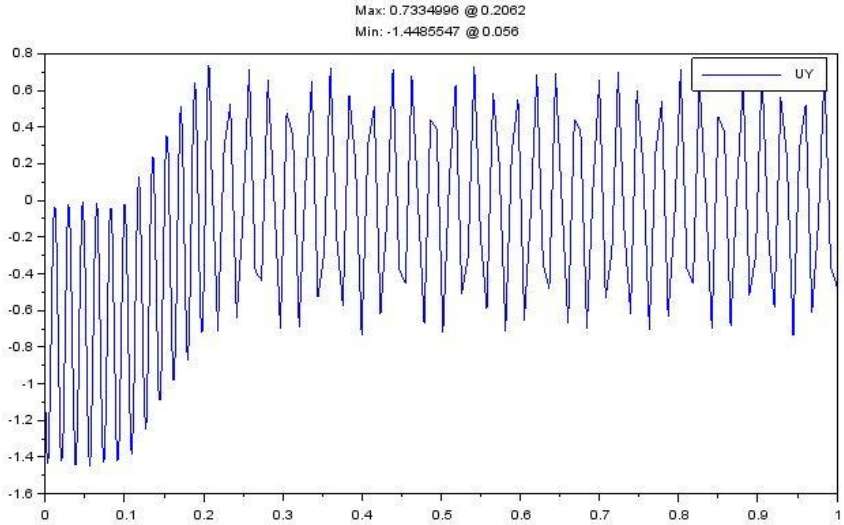




Post Process Results

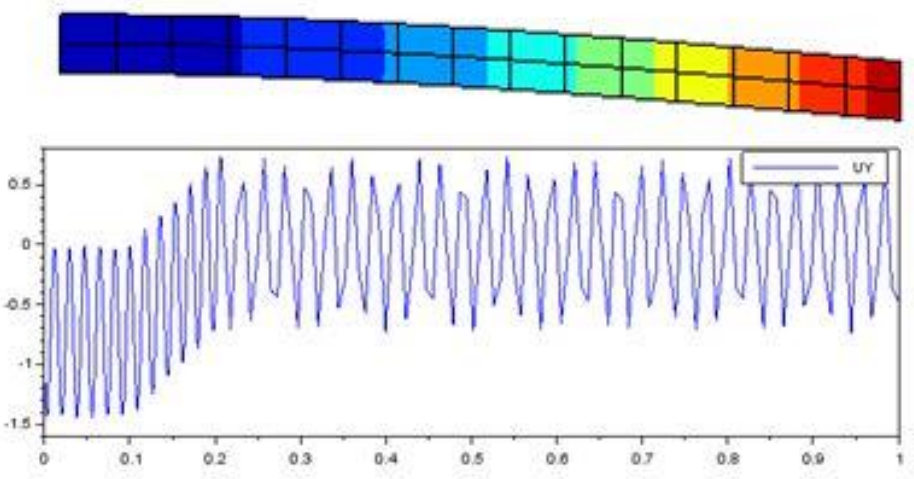
Using the post processor tab, we can plot the graph of any result quantity to study the dynamic response. To change the plot, select the type of results from results dropdown box.





Summary

In this tutorial we have prepared the geometry and the mesh in GMSH. The mesh is then imported in SCIFESOL to perform linear transient analysis. Damping is not included in the analysis , hence the dynamic response of the beam is similar to response of undamped spring mass system.



Thanks!