



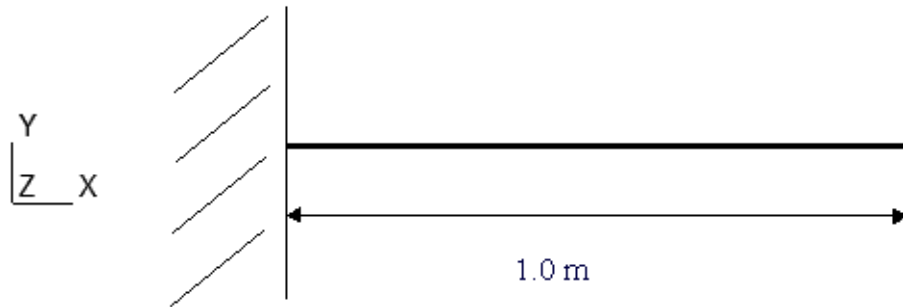
IngentaCal

Engineering Solutions

SCIFESOL Tutorial : Modal Analysis of Beam

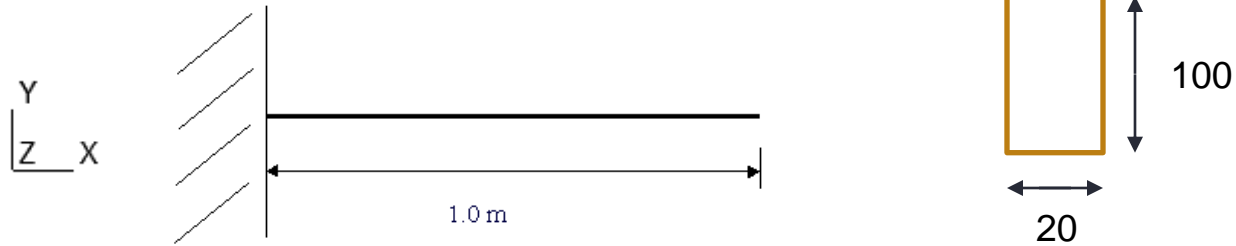
Problem Description

Modal analysis of a cantilever beam having uniform cross section is performed to calculate the mode shapes and natural frequencies of the beam.



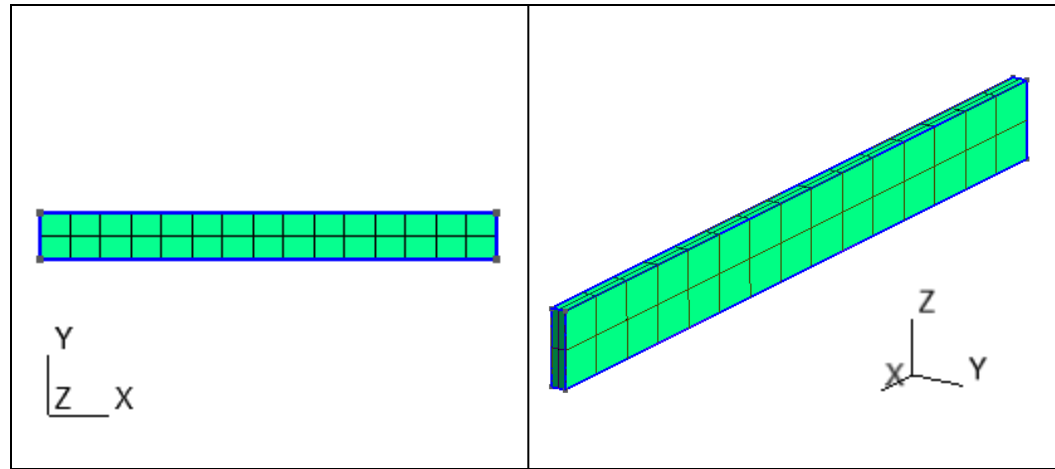
Geometry

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

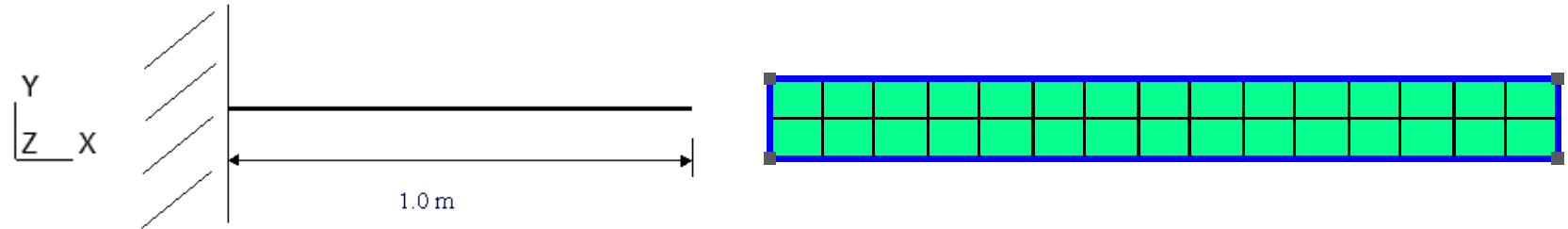


Mesh Generation

We model the beam using 2D plane stress quadrilateral element and 8 node hexahedral element.

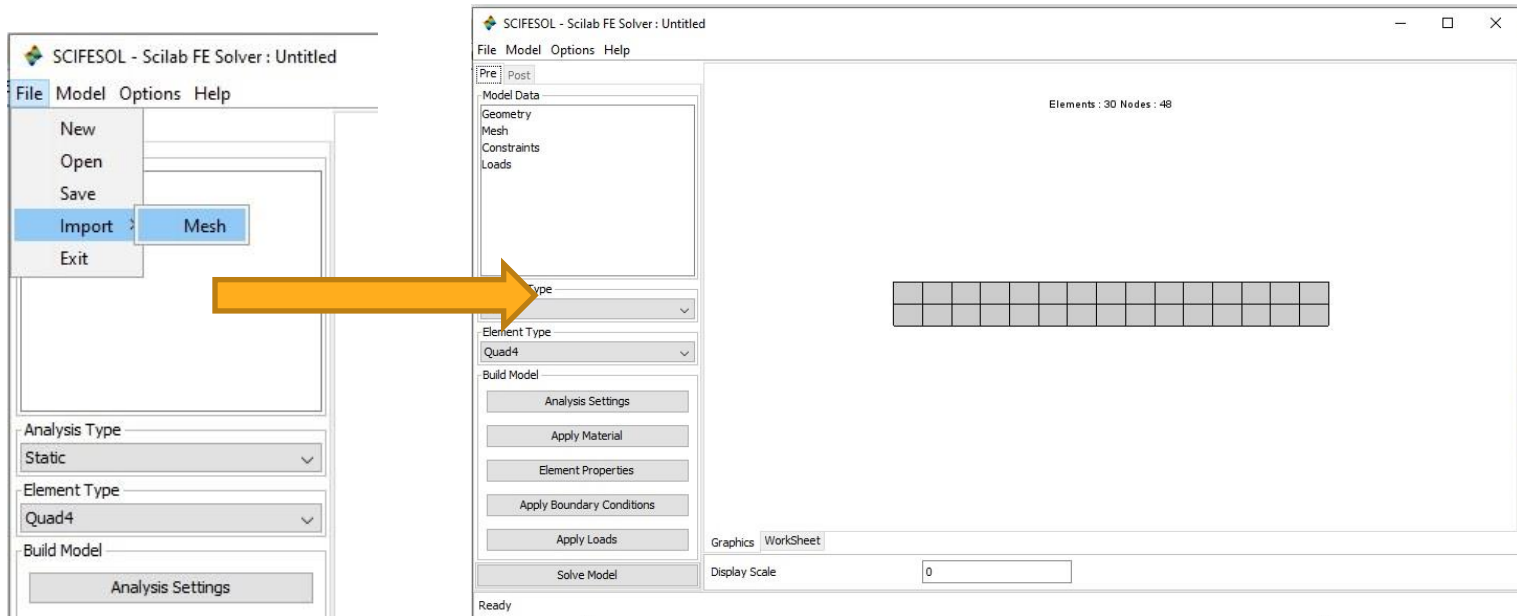


2D Plane Stress Model



Import Mesh

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



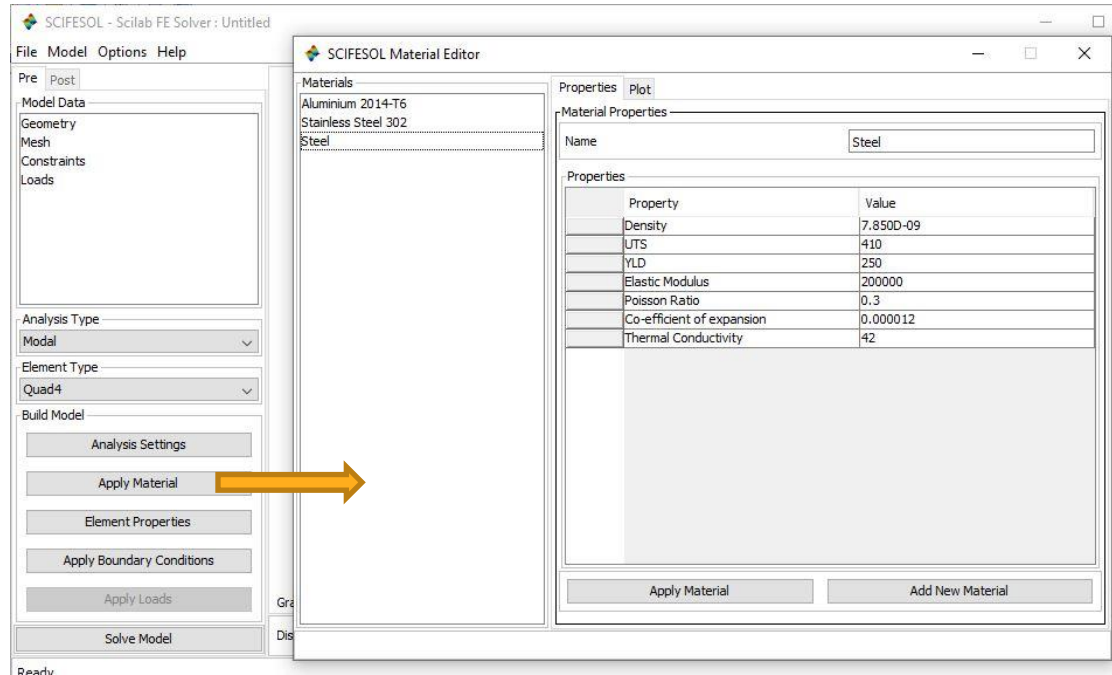
Define Analysis Settings

- Select analysis type to Modal.
- Using analysis settings , specify mode extraction method.

The image displays the SCIFESOL - Scilab FE Solver interface. The main window is titled "SCIFESOL - Scilab FE Solver : Untitled" and has a menu bar with "File", "Model", "Options", and "Help". Below the menu bar, there are tabs for "Pre" and "Post". The "Model Data" section is visible, containing "Geometry", "Mesh", "Constraints", and "Loads". A red box highlights the "Analysis Type" dropdown menu, which is open, showing options: "Static", "Modal", "Transient", and "Transient : Mode Superposition". The "Modal" option is selected. An orange arrow points from the "Analysis Settings" button in the "Build Model" section to a separate dialog box titled "Analysis Settings : Modal". This dialog box has a title bar with a close button and a maximize button. It contains a "No. of Mode Shapes" input field with the value "6". Under the "Method" section, there are two radio buttons: "Inverse Iteration" (which is selected) and "Subspace Iteration". At the bottom of the dialog box is an "Apply" button.

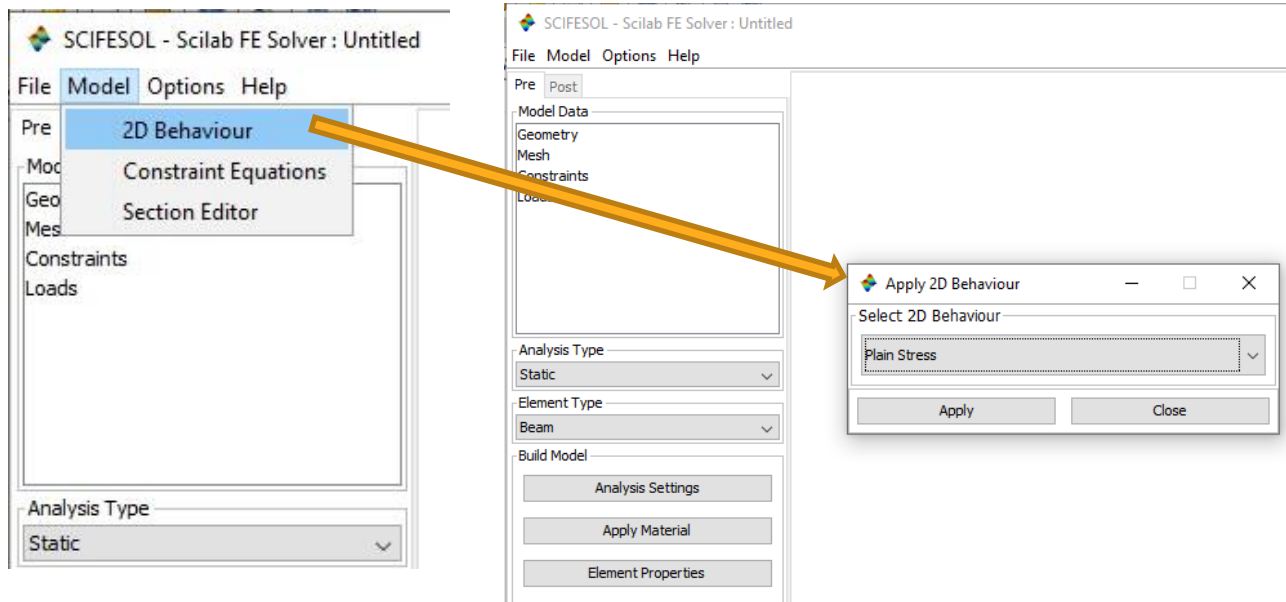
Define Material Properties

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



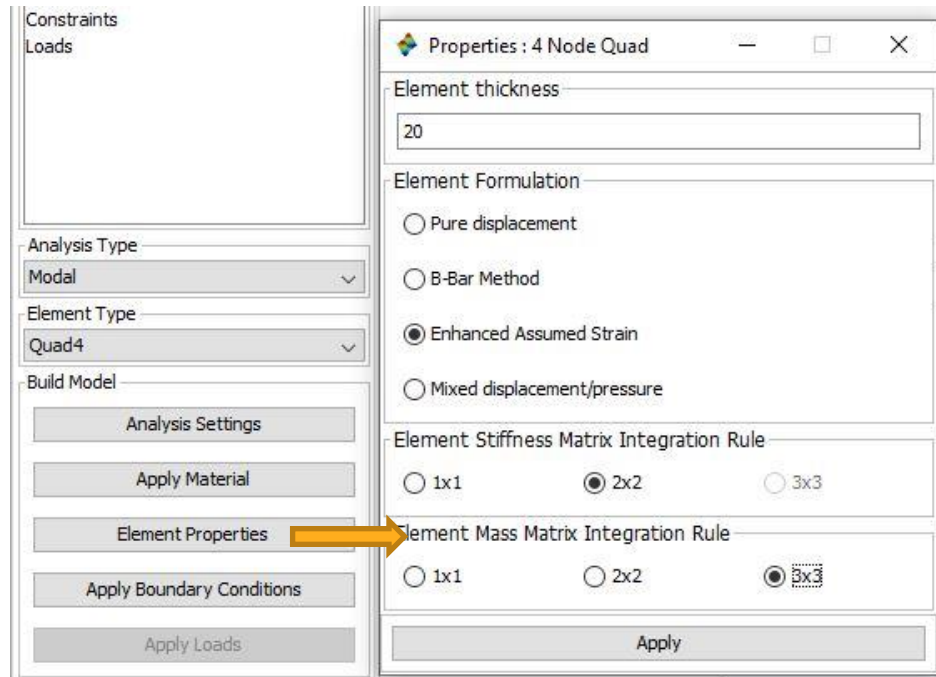
Define Element Properties

- 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 Strain method.
- Select the stiffness matrix integration rule as 2x2.
- Select the mass matrix integration rule as 3x3.
- Specify the thickness as 20 mm.



Apply Boundary Conditions

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

The screenshot displays the SCIFESOL - Scilab FE Solver interface. The main window shows a beam model with a coordinate system from 0 to 1000. A legend on the right identifies boundary conditions: Bid: 1 (black), Bid: 2 (blue), Bid: 3 (green), and Bid: 4 (cyan). A dialog box titled "Apply Fixed Boundary Condition..." is open, with "Select Region" selected and the value "4" entered. An arrow points from a yellow box labeled "Left end fixed" to the left end of the beam in the main window.

SCIFESOL - Scilab FE Solver : Untitled

File Model Options Help

Pre Post

Model Data

Geometry

Mesh

Constraints

Loads

Analysis Type

Modal

Element Type

Quad4

Build Model

Apply Fixed Boundary Condition...

Select Fixed Nodes / Region

Select Region

4

Select Node Id

0

Apply Close

Legend:

- Bid: 1
- Bid: 2
- Bid: 3
- Bid: 4

Left end fixed

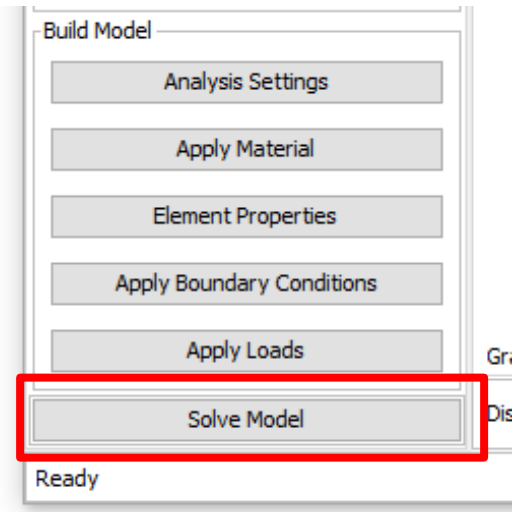
1000 500 0

1000 100 0

-200 -100 0 100 200 300 400 500 600 700 800 900 1000

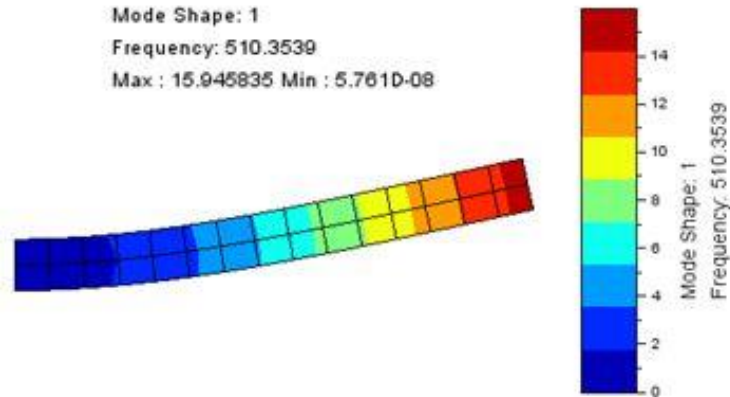
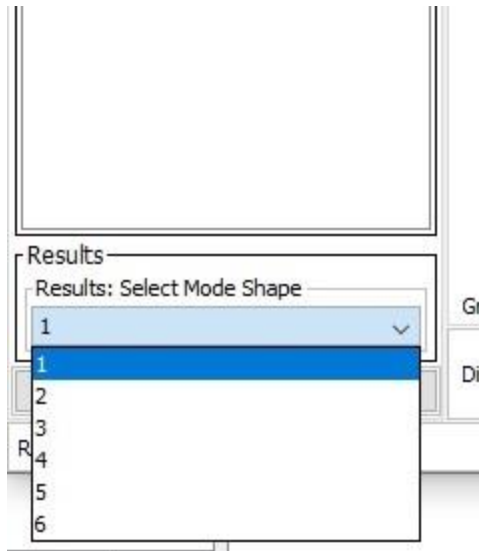
Solve

Start Modal solver to solve the model.

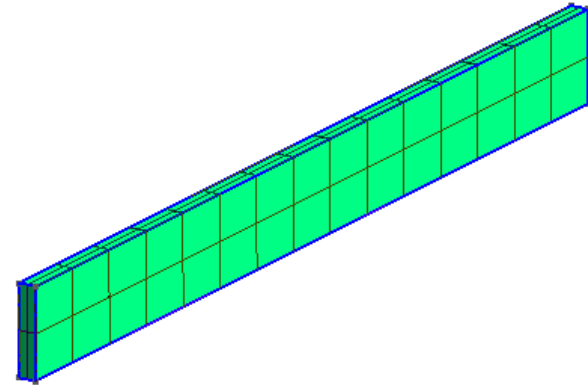
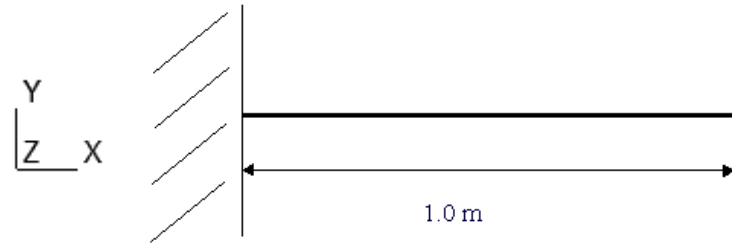


Results

Using the post processor tab, we plot the required Mode shapes.



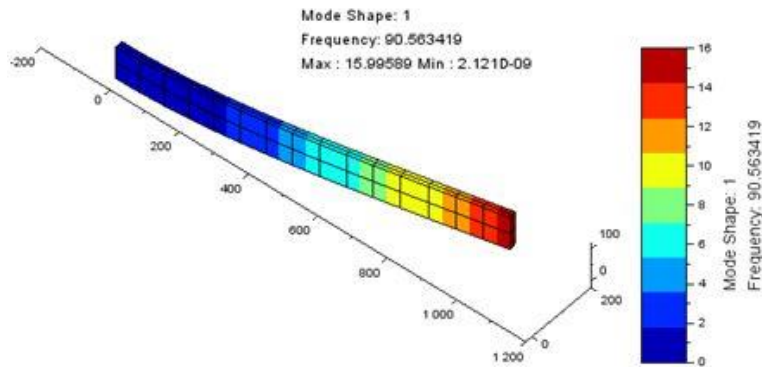
3D Model



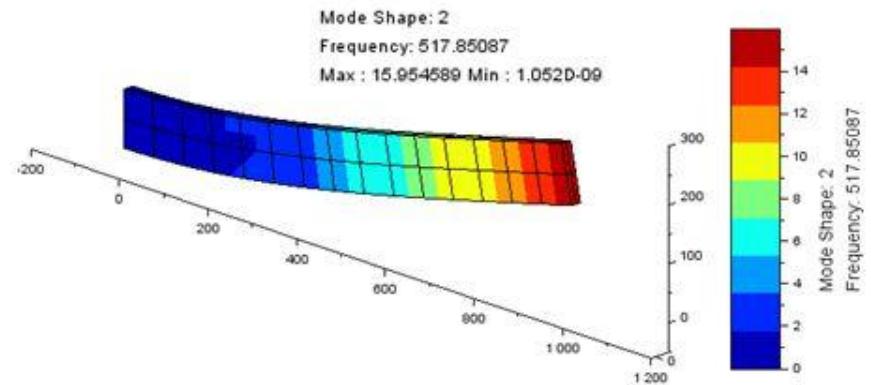
Using the same procedure we solve for 3D model of the beam using Hexahedral elements .

Results

In 3D analysis, the order of mode shapes change as degree of freedom of the model increases.



Lateral Bending Mode



Vertical Bending Mode



Thanks!