ANSYS - Vibration Analysis of a Frame - Step 5

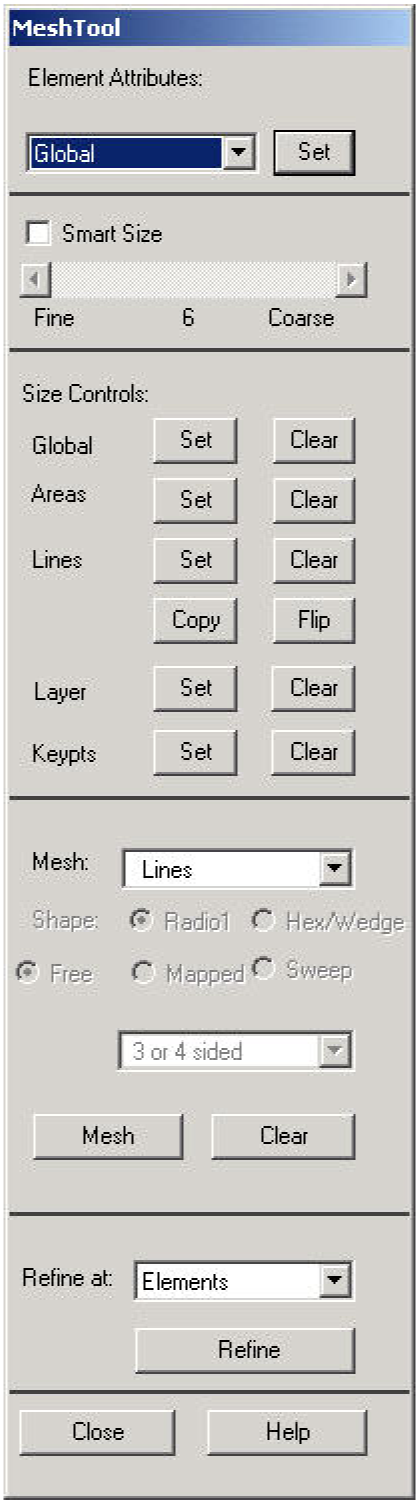
Step 5: Mesh geometry

Generate the Mesh

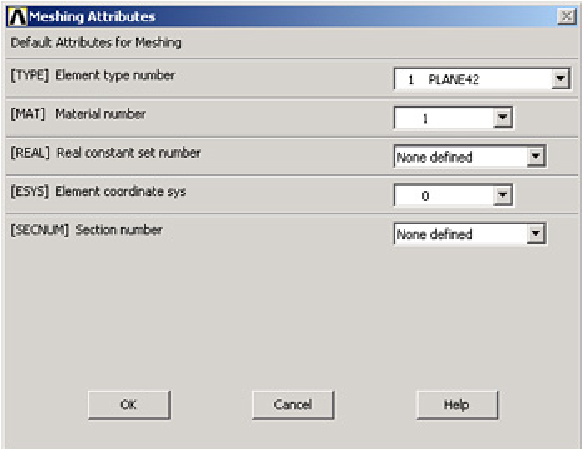
To bring up the MeshTool, Select:

Main Menu > Preprocessor > MeshTool.

Click on ***Set*** under ***Element Attributes*** in the MeshTool.



This brings up the *Meshing Attributes* menu. You will see that the correct element type, material number and real constants are already selected since we have only one of each.

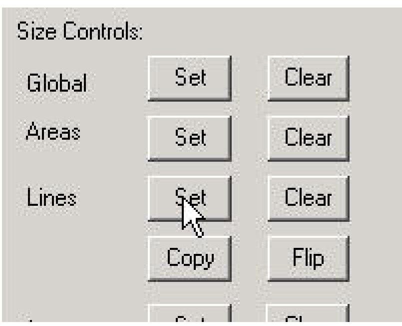


Close this menu by clicking ***OK***.

Define Number of Elements for Each Line

We'll use 20 elements for AB and 20 elements for BC to be consistent with Cook et al.

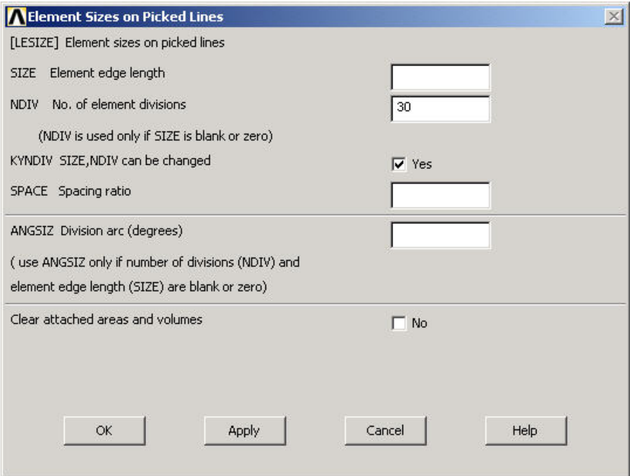
Under ***Size Control*** and ***Lines***, click ***Set***.



Select line AB.

Click ***OK***.

Enter `30` for ***NDIV***.

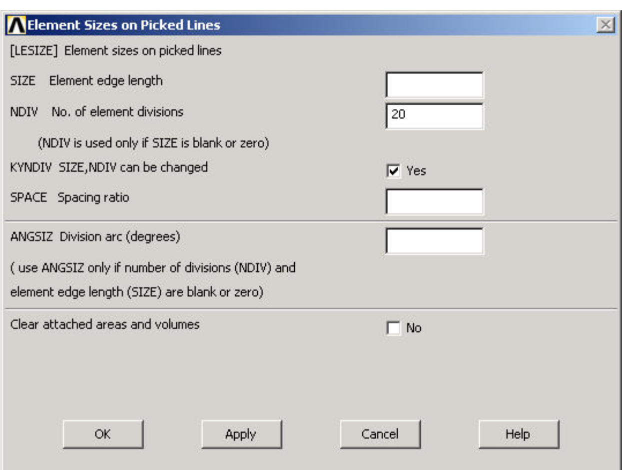


Click ***Apply***.

Select line BC.

Click ***OK***.

Enter `20` for ***NDIV***.



Click ***OK***.

Creating the Mesh

In the *MeshTool*, click on ***Mesh***. This brings up the *pick* menu. Click on ***Pick All***.

The geometry has been meshed and the elements are plotted in the graphics window. Close the *MeshTool*.

Save your work

Once you have successfully created the mesh, click on ***SAVE_DB*** in the *ANSYS Toolbar* to save the database.

Go to Step 6: Specify boundary conditions

Go to all ANSYS Learning Modules