ANSYS - Plate with a Hole - Step 5

Step 5: Mesh geometry

Bring up the *MeshTool*:

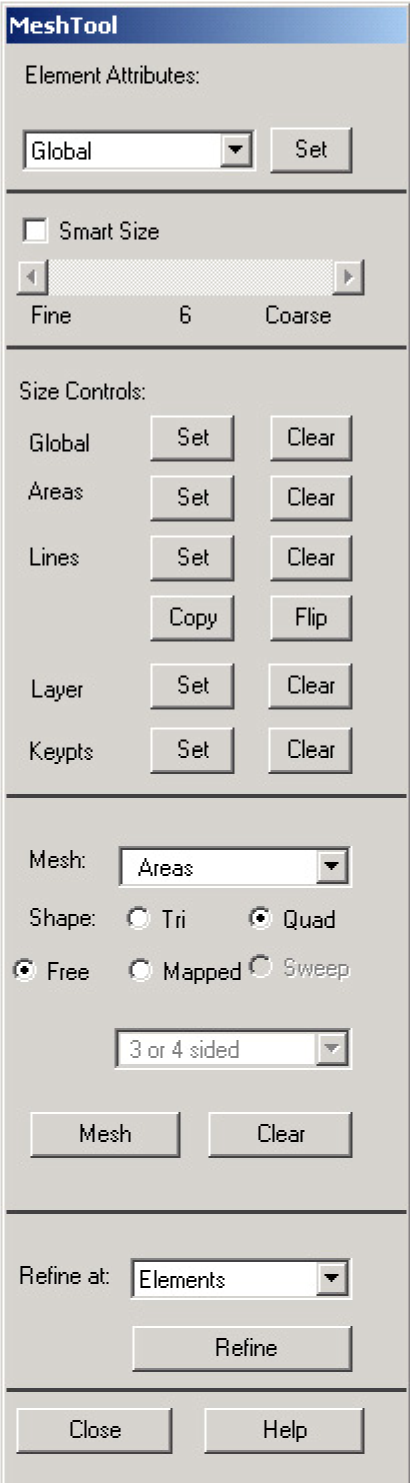
Main Menu > Preprocessor > Meshing > MeshTool

The *MeshTool* is used to control and generate the mesh.

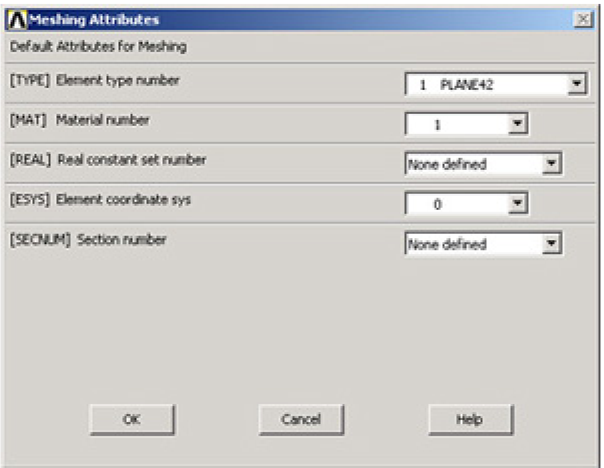
Set Meshing Parameters

We'll now specify the element type, real constant set and material property set to be used in the meshing. Since we have only one of each, we can assign them to the entire geometry using the ***Global*** option under ***Element Attributes***.

Make sure ***Global*** is selected under ***Element Attributes*** and click on Set.



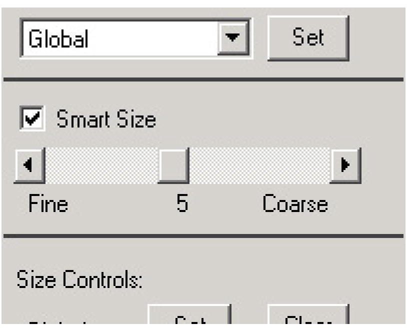
This brings up the *Meshing Attributes* menu. You will see that the correct element type and material number are already selected since we have only one of each. Recall that no real constants need to be defined for *PLANE42* element type with the plane stress keyoption.



Click ***OK***. ANSYS now knows what element type and material type to use for the mesh.

Set Mesh Size

Instead of setting the mesh size at each boundary, we'll use the *SmartSize* option which enables automatic element sizing. Click on the ***SmartSize*** checkbox so that a tickmark appears in it.

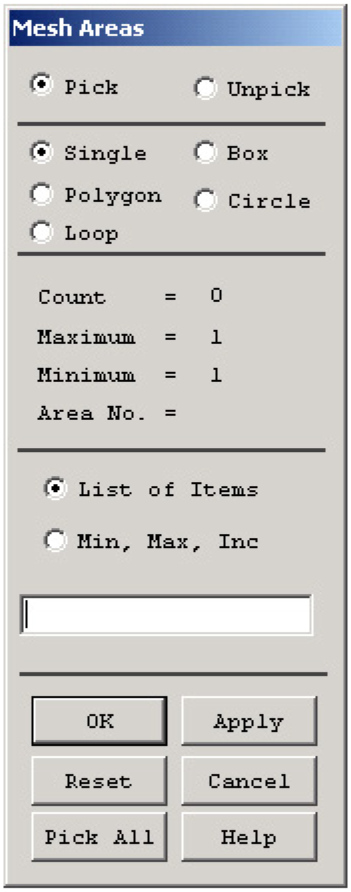


The only input necessary for the ***SmartSize*** option is the overall element size level for meshing. The element size level determines the fineness of the mesh. Its value is controlled by the slider shown in the above picture. Change the setting for the overall element size level to ***5*** by moving the slider under ***S martSize*** to the left.

Mesh Areas

In the *MeshTool*, make sure ***Areas*** is selected in the drop-down list next to ***Mesh***. This means the geometry components to be meshed are areas (as opposed to lines or volumes). We'll use quadrilateral elements. So make sure the default option of ***Quad*** is selected under ***Shape***. We'll also use the default of ***Free*** meshing.

Click on *the* ***Mesh*** button. This brings up the pick menu.



In the *Input* window, ANSYS tells you to "pick or enter areas to be meshed". Since we have only one area to be meshed, click on ***Pick All***. The geometry has been meshed and the elements are plotted in the *Graphics* window. ***Close*** the *MeshTool*.

The mesh statistics are reported in the *Output* window (usually hiding behind the *Graphics* window):

** AREA 3 MESHED WITH 105 QUADRILATERALS, 0 TRIANGLES **
** Meshing of area 3 completed ** 105 elements.

NUMBER OF AREAS MESHED = 1
MAXIMUM NODE NUMBER = 130
MAXIMUM ELEMENT NUMBER = 105

Save Your Work

Toolbar > SAVE_DB

Go to Step 6: Specify boundary conditions

See and rate the complete Learning Module

Go to all ANSYS Learning Modules