# ANSYS - Plate with a Hole - Step 5

**Problem Specification** 

- 1. Start-up and preliminary set-up
- 2. Specify element type and constants
- 3. Specify material properties
- 4. Specify geometry
- 5. Mesh geometry
- 6. Specify boundary conditions
- 7. Solve
- 8. Postprocess the results
- 9. Validate the results

Problem Set 1

## Step 5: Mesh geometry

Bring up the MeshTool:

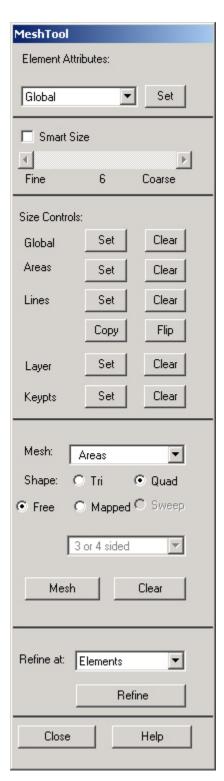
#### Main Menu > Preprocessor > Meshing > MeshTool

The MeshTool is used to control and generate the mesh.

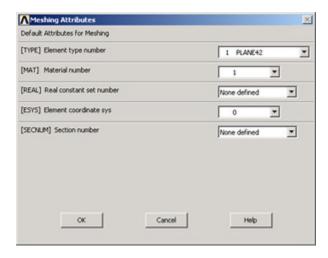
### **Set Meshing Parameters**

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.



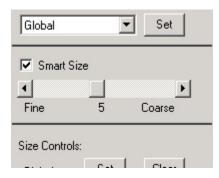
This brings up the *Meshing Attributes* menu. You will see that the correct element type and material number are already selected since we have only one of each. Recall that no real constants need to be defined for *PLANE42* element type with the plane stress keyoption.



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.

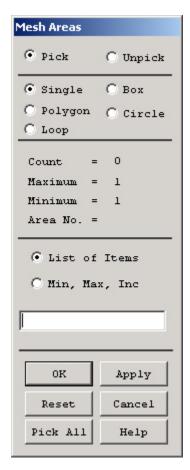


The only input necessary for the *SmartSize* option is the overall element size level for meshing. The element size level determines the fineness of the mesh. Its value is controlled by the slider shown in the above picture. Change the setting for the overall element size level to 5 by moving the slider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