# Importing ANSYS meshes into the finite element Matlab Toolbox

This tutorial on meshing is extracted from the ANSYS tutorial (plate with a hole) developed by Dr. R. Bhaskaran and given on the Swanson lab web site at this link.

Problem Specification

Consider the square plate of uniform thickness with a circular hole with dimensions shown in the figure below. In this tutorial, we only consider how to create the geometry, generate the finite element mesh and assign markers on the different boundary segments (which will be useful in the MATLAB programs for assigning boundary conditions).
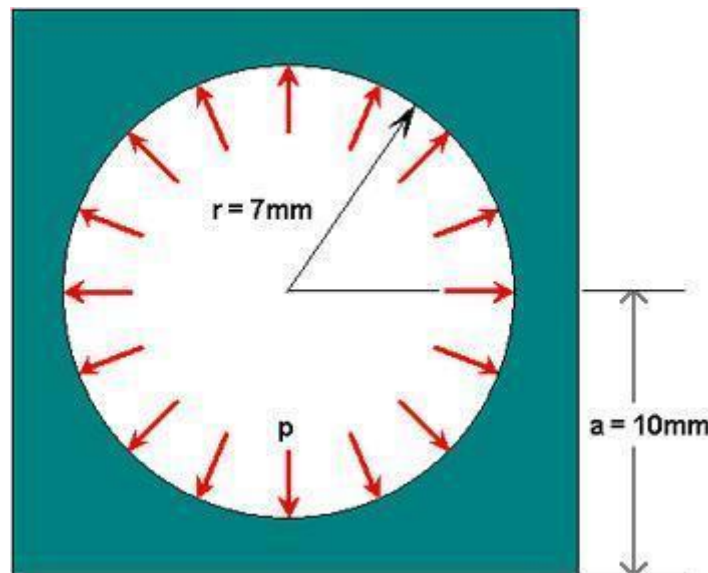
Step 1: Start-up and preliminary set-up

Create a folder

Create a folder called *plate* at a convenient location. We'll use this folder to store files created during the ANSYS session. There are two files provided with this tutorial: **ansys_ch.mac** and **start120.ans.** Here the number 120 is the version of the ANSYS. If you use older version, e.g. 100, you may change the file name to start100.ans.

Put these two files into the folder *plate* you just created.

Start ANSYS

Start > Programs > ANSYS 12.0 > Mechanical APDL Product Launcher

In the window that comes up, enter the location of the folder you just created as your *Working directory* by browsing to it. All files generated during the ANSYS run will be stored in this directory.

Specify *plate* as your *Initial jobname*. The jobname is the prefix used for all files generated during the ANSYS session. For example, when you perform a save operation in ANSYS, it'll store your work in a file called *plate.db* in your working directory.

For this tutorial, we'll use the default values for the other fields. Remember to choose the *Simulation Environment* to ANSYS. Click on *Run.* This brings up the ANSYS interface. Now in *ANSYS Toolbar*, you should see a button: **ANSYS_CH** as follows:
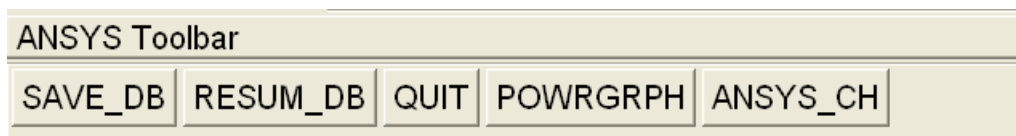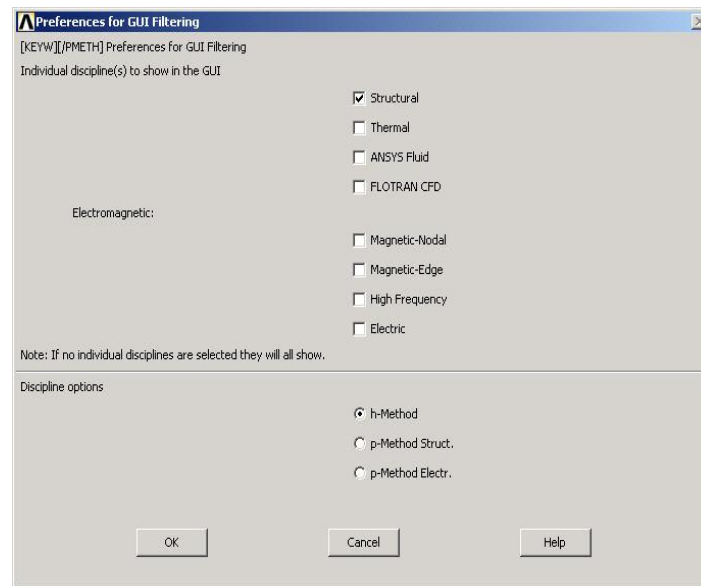

Later, we will use this button to export the mesh generated by ANSYS to four files which are compatible with our MatLab code.

Set Preferences

As before, we'll more or less work our way down the *Main Menu*.

Main Menu > Preferences

In the *Preferences for GUI Filtering* dialog box, click on the box next to *Structural* so that a tick mark appears in the box. Click *OK*.

Recall that this is an optional step that customizes the graphical user interface so that only menu options valid for structural problems are made available during the ANSYS session.
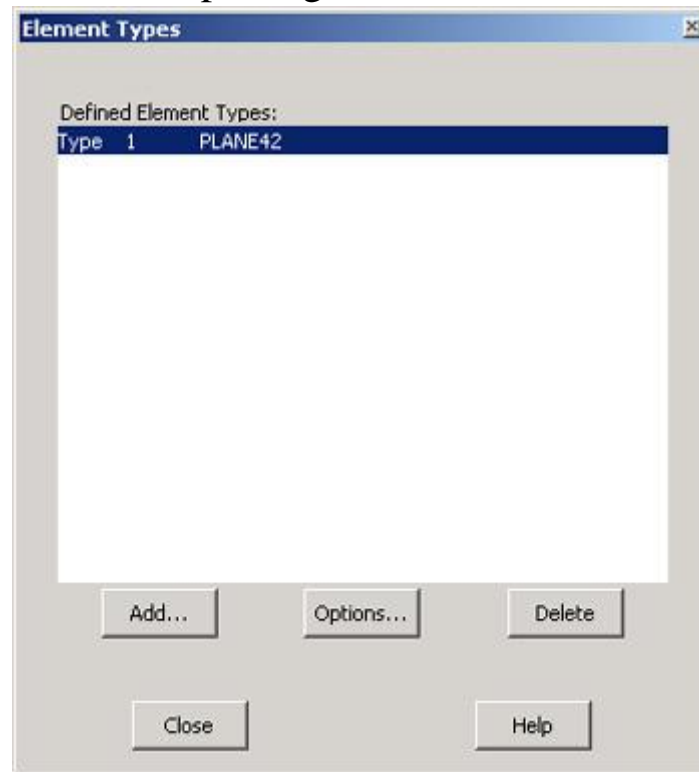
Step 2: Specify element type

Specify Element Type

Main Menu > Preprocessor> Element Type > Add/Edit/Delete > Add...

Pick *Structural Solid* in the left field and *Quad 4 node 42* in the right field. Click *OK* to select this element.
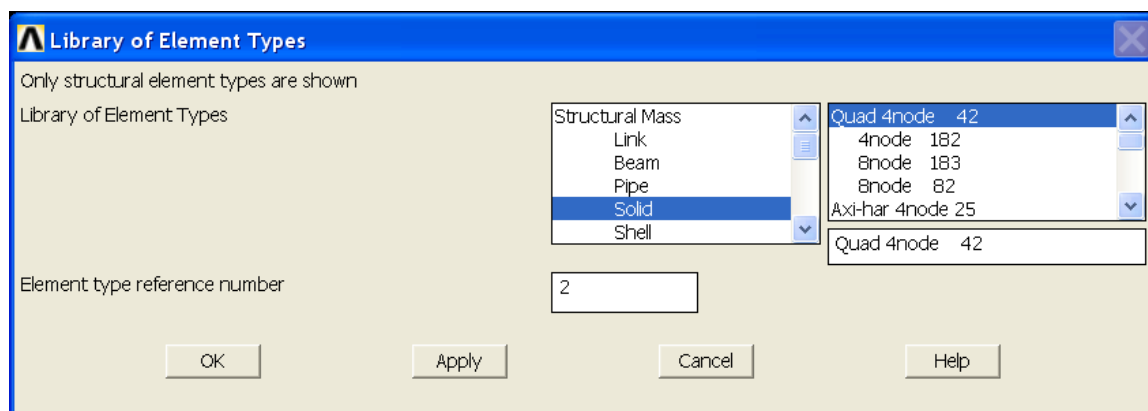
You'll now see the *Element Types* menu with *PLANE42* as the only defined element type.

You can look at the online help pages to learn about the properties of this element.

Utility Menu > Help > Help Topics

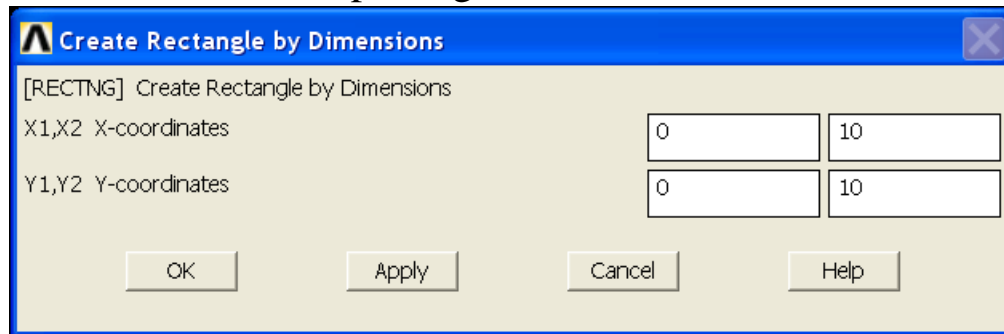*Close* the *Element Types* menu.

Step 3: Specify geometry

The geometry, material properties and loading are assumed all symmetric with respect to the horizontal and vertical centerlines and thus we need to model only a quarter of the plate. We will take the origin of the coordinate system to be at the center of the hole and model only the top right quadrant. We'll create the geometry by creating a square area of side *a* and subtracting the circular sector of radius *r* from it.

Create the Square

Main Menu > Preprocessor > Modeling >Create > Areas >Rectangle > By Dimensions

X1 and *X2* are the x-coordinates of the left and right edges of the square, respectively. Enter `0` for *X1*, `10` or *X2*.

Y1 and *Y2* are the y-coordinates of the bottom and top edges of the square, respectively. Enter `0` for *Y1*, `10` for *Y2*.

Create Rectangle by Dimensions

[RECTNG] Create Rectangle by Dimensions

X1,X2 X-coordinates | 0 | 10 |

Y1,Y2 Y-coordinates | 0 | 10 |

OK   Apply   Cancel   Help

Click *OK*. You should see a square appear in the graphics window.
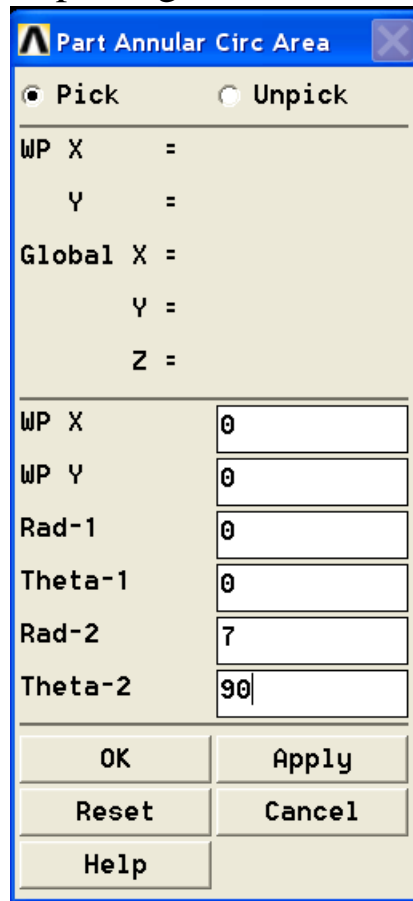
Create the Circular Sector

Main Menu > Preprocessor > Modeling > Create > Areas > Circle > Partial Annulus

WP X and *WP Y* are the x- and y-coordinates of the center of the circular arc. So enter 0 for both *WP X* and *WP Y*. (*WP* refers to the *Working Plane* which by default coincides with the global Cartesian coordinate system. We won't have to worry about the working plane in this friendly example.)

Rad-1 is the radius of the inner circular arc. We want to create a solid rather than an annular arc. Enter 0 for *Rad-1* to create a solid arc.

Rad-2 is the (outer) radius of the arc. Since we had defined the hole radius as parameter *r* earlier, enter 7 for *Rad-2*.

Theta-1 and *Theta-2* are the starting and ending angles of the arc, respectively. These angles need to be specified in degrees. Enter 0 for *Theta-1* and 90 for *Theta-2*. Click *OK*.

This will create and draw the circular sector. You'll see a white line denoting the circular sector.

Subtract Circular Sector from Square

**Main Menu > Preprocessor >Modeling > Operate > Booleans > Subtract > Areas**

In the *Input* window, ANSYS tells you to "pick or enter base areas from which to subtract". So we pick the square area as follows: Hold down the left mouse button, move the cursor over the areas until the square is selected (it will change color) and release the left mouse button. Click *OK*.
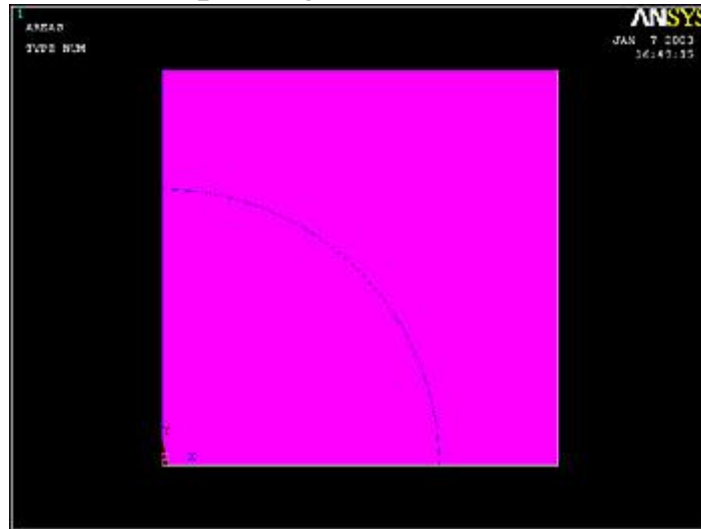
In the *Input* window, ANSYS now tells you to "pick or enter areas to be subtracted". So select the circular sector by holding down and releasing the left mouse button. Click ***OK***.
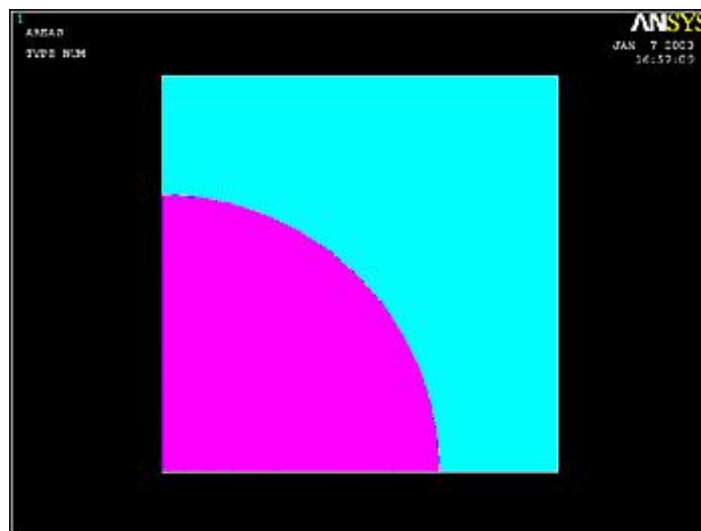
If you did this correctly, you will see that the circular sector has been subtracted out from the square area.
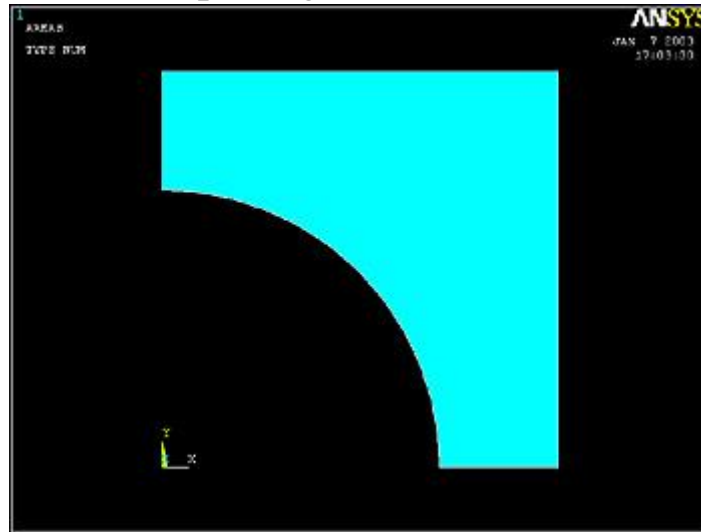
> You can also select areas during the Boolean subtract operation by simply clicking on them but it becomes difficult to select areas (and other components) in this fashion in more complicated geometries. That's why I made you use the "holding-down-the-mouse-and-releasing" technique.
>
> If you picked an area incorrectly, you can unpick it by clicking the right mouse button and selecting the area. The cursor changes to a downward arrow during an unpick operation. Right-click to return to pick mode.

Step 4: Mesh geometry

Bring up the *MeshTool*:

**Main Menu > Preprocessor > 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*.
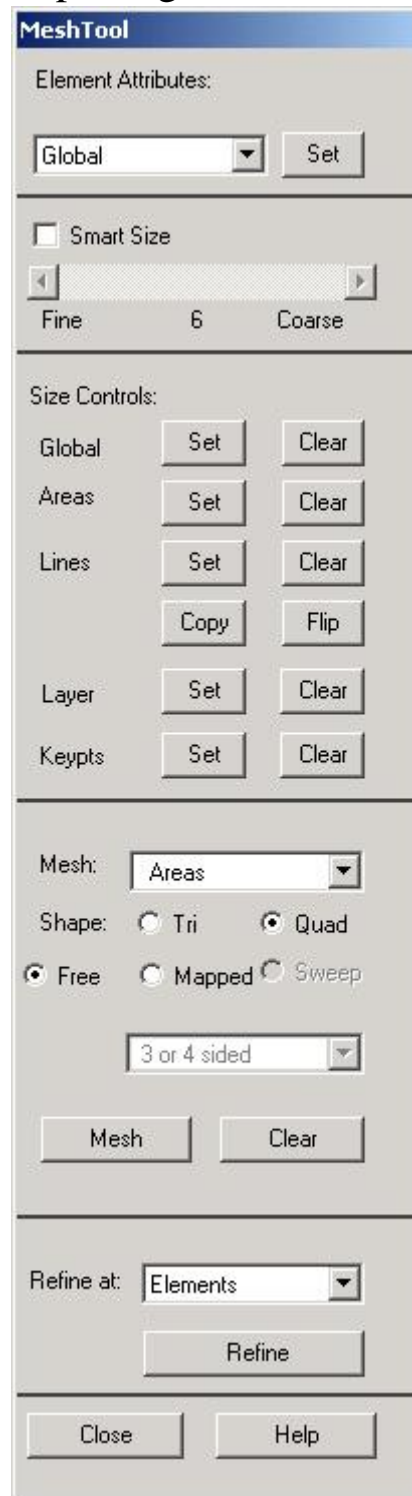
This brings up the *Meshing Attributes* menu. You will see that the correct element type and material number are already selected since we have only one of each. Recall that no real constants need to be defined for *PLANE42* element type with the plane stress key option.
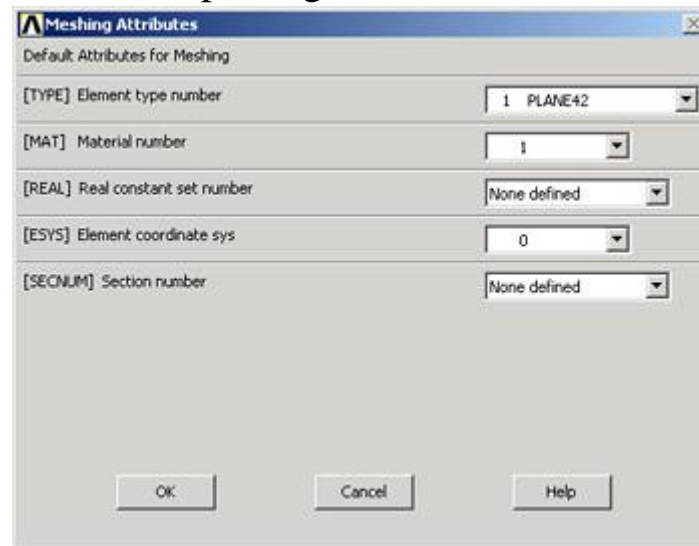
Click **OK**. ANSYS now knows what element type and material type to use for the mesh.

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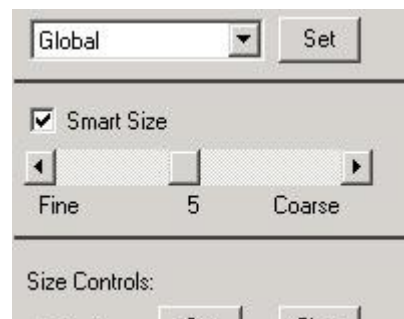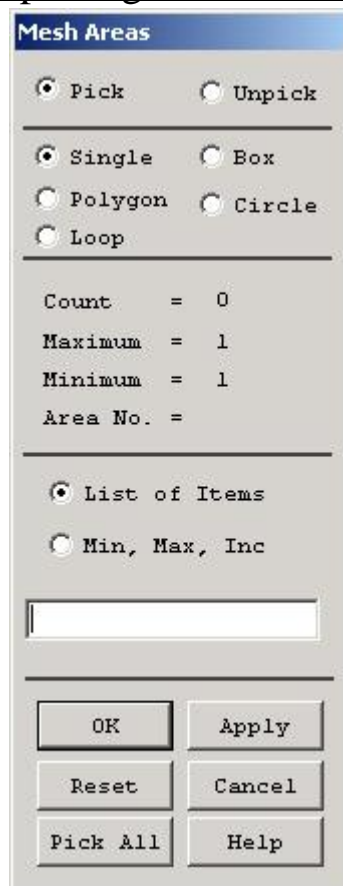
The only input necessary for the *SmartSize* option is the overall element size level for meshing. The element size level determines the fineness of the mesh. Its value is controlled by the slider shown in the above picture. Change the setting for the overall element size level to *5* by moving the slider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

Mesh Areas

- ⦿ Pick   ○ Unpick

- ⦿ Single   ○ Box
- ○ Polygon ○ Circle
- ○ Loop

Count   =   0
Maximum =   1
Minimum =   1
Area No. =

- ⦿ List of Items
- ○ Min, Max, Inc

OK	Apply
Reset	Cancel
Pick All	Help

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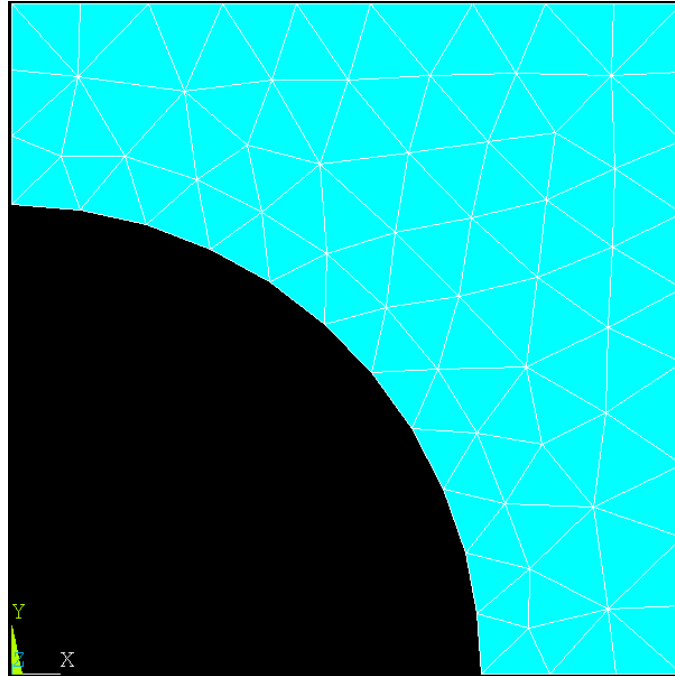
The mesh statistics are reported in the *Output* window (usually hiding behind the *Graphics* window):

** Meshing of area 3 completed ** 105 elements.
NUMBER OF AREAS MESHED   =   1
MAXIMUM NODE NUMBER   =   71
MAXIMUM ELEMENT NUMBER   =   105

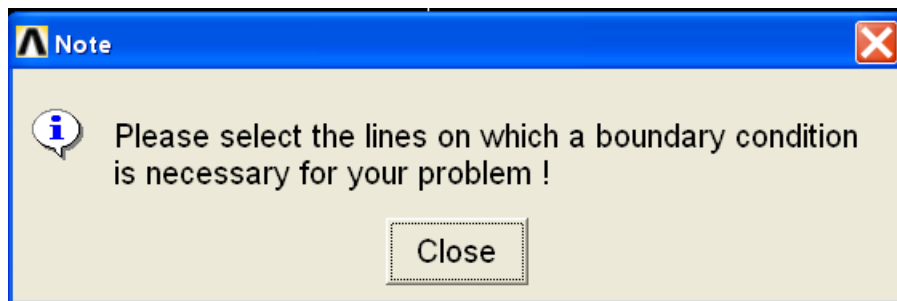You can refine the mesh by choose **refine** option
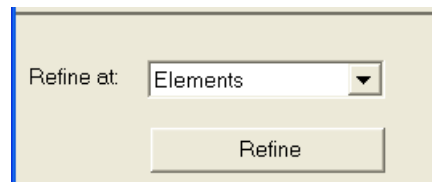
There are different refine options. You can choose refine at element level and select which element you want to refine.
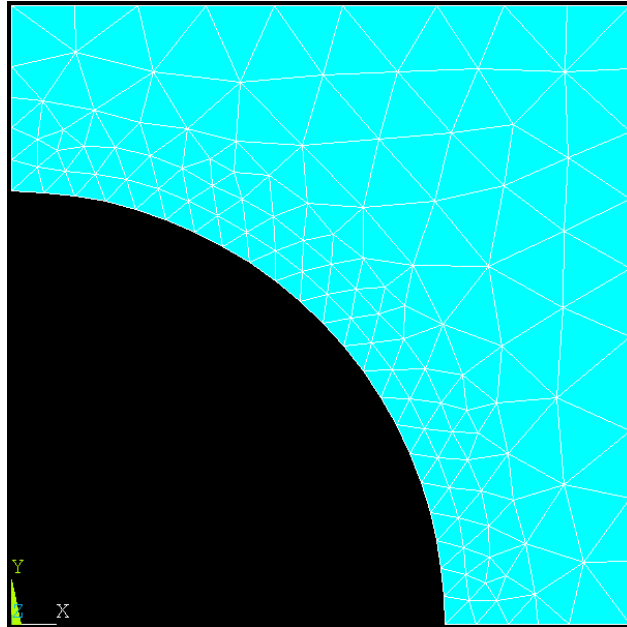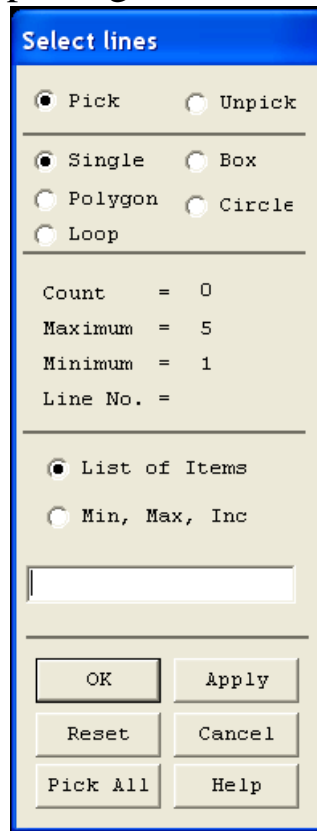
Step 5: Generate grid files

Now clicking on the 'ANSYS_CH' button on the ANSYS Toolbar, a window will pop out
to ask you to select boundaries,

You can select all the boundaries on the geometry or you can just choose the boundaries
which have physical boundary conditions for your problem.

Select lines

○ Pick   ○ Unpick

● Single   ○ Box
○ Polygon   ○ Circle
○ Loop

Count   =   0
Maximum   =   5
Minimum   =   1
Line No. =

● List of Items

○ Min, Max, Inc

OK	Apply
Reset	Cancel
Pick All	Help

Now in your folder, you will see the following 4 files:

ansys.elem, ansys.header, ansys.node, ansys.boundary.

Put these four files in you Matlab working directory and use the provided Matlab program (run Main.m) to check your mesh with boundary indicators.

2D Grid Plot