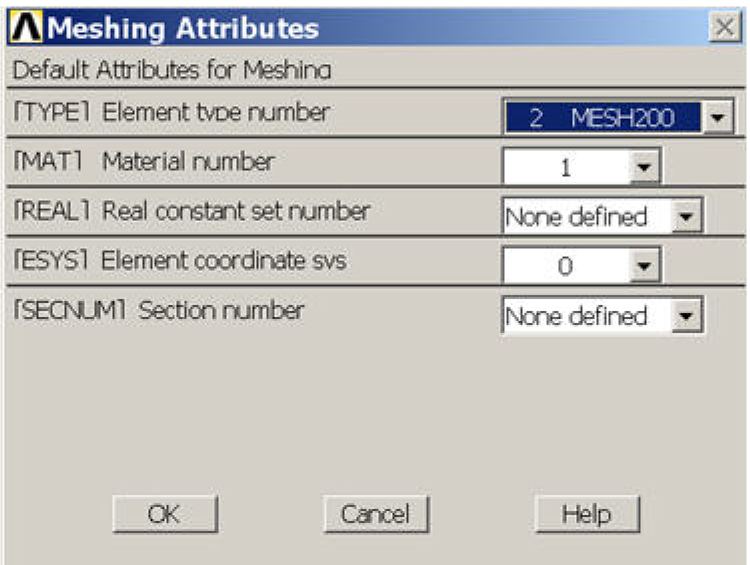
ANSYS 11 - Crank Step 5

Step 5: Mesh geometry

Bring up the *MeshTool*:

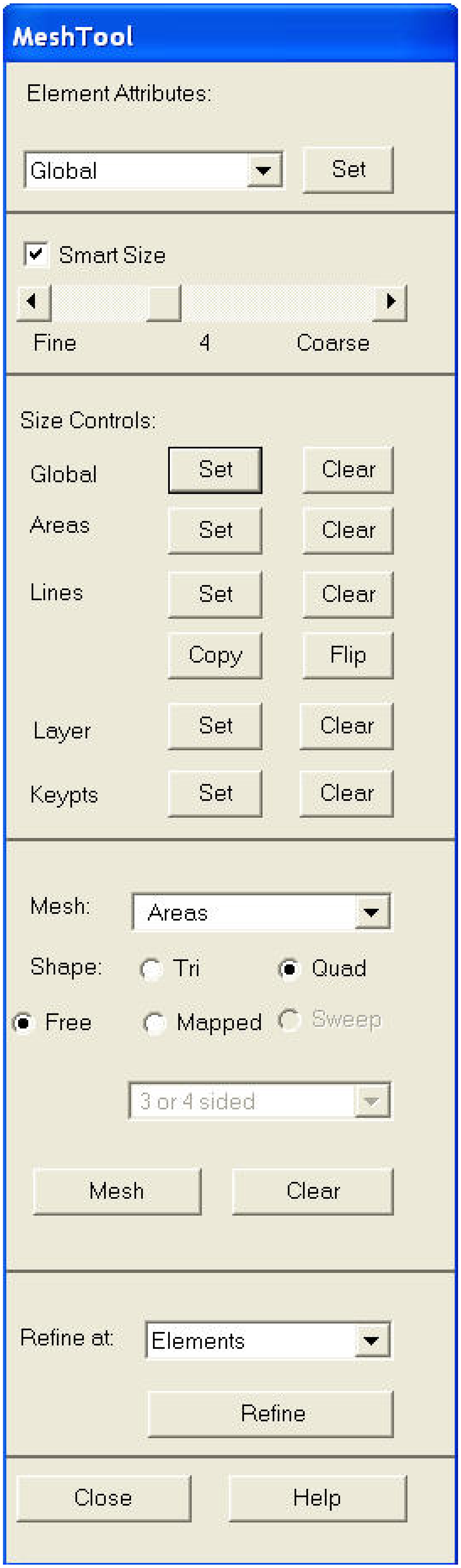
Main Menu > Preprocessor > Meshing > MeshTool

We'll first mesh the two front surfaces using *MESH200*. Click ***Set*** next to ***Global*** under ***Element Attributes***. Set the ***TYPE*** to ***MESH200*** and click ***OK***.

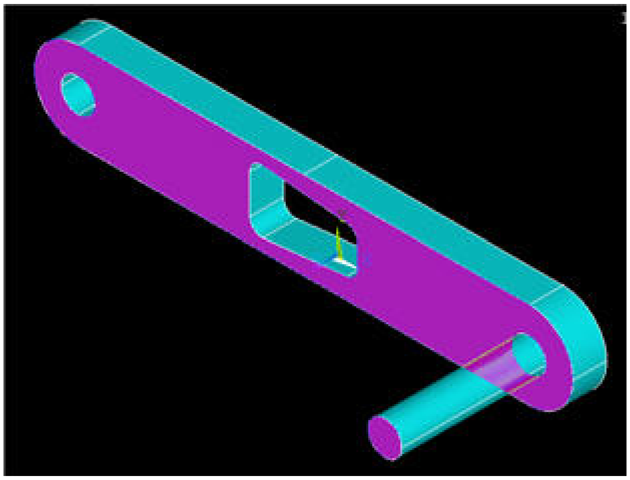


According to the ANSYS manual, "Smart element sizing (SmartSizing) is a meshing feature that creates initial element sizes for free meshing operations. SmartSizing gives the mesher a better chance of creating reasonably shaped elements during automatic mesh generation ... The SmartSizing algorithm first computes estimated element edge lengths for all lines in the areas or volumes being meshed. The edge lengths on these lines are then refined for curvature and proximity of features in the geometry." To turn on SmartSizing, check the box next to ***Smart Size***. Drag the slider to a size of 4 to get a finer mesh than the default.

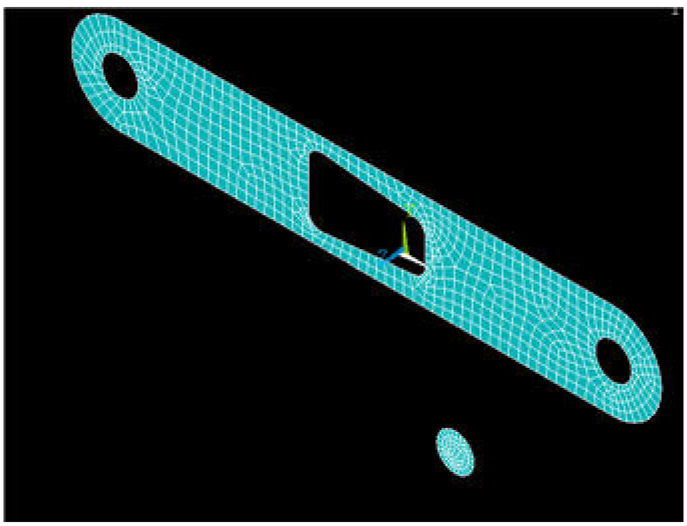
In order to have a little more control over what mesh ANSYS creates for us, we will set the *starting* element size for SmartSizing rather than use the default. Smartsizing will take this starting element size and modify/vary it over the geometry to account for curvature and corners. Under ***Size Controls,*** click the ***Set*** button next to ***Global***. Enter an ***element edge length*** of 0.12 and click ***OK***. The specified smart size of 4 and edge length of 0.12 are the result of an iterative process. You should experiment with different settings for these parameters to study the effect of the mesh on your solution, as discussed in Step 9. The goal is to obtain a solution that doesn't change as you refine the mesh.


MeshTool
Element Attributes:
Global
Set
Smart Size
Fine
4
Coarse
Size Controls:
Global
Set
Clear
Areas
Set
Clear
Lines
Set
Clear
Copy
Flip
Layer
Set
Clear
Keypts
Set
Clear
Mesh:
Areas
Shape:
Tri
Quad
Free
Mapped
Sweep
3 or 4 sided
Mesh
Clear
Refine at:
Elements
Refine
Close
Help

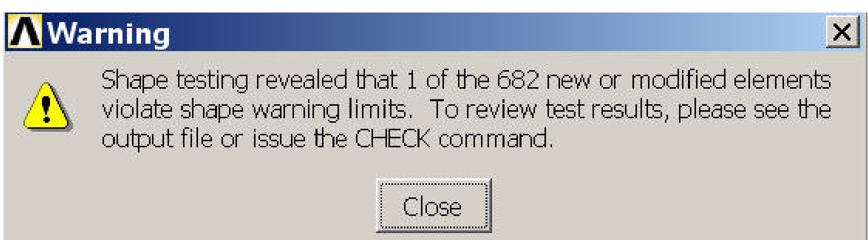
Select ***Areas*** to be meshed with a ***Quad*** shape using the ***Free*** mesher. Click ***Mesh***. Pick the front face of the crank and the pedal shaft.



Click ***OK***. You will now see:



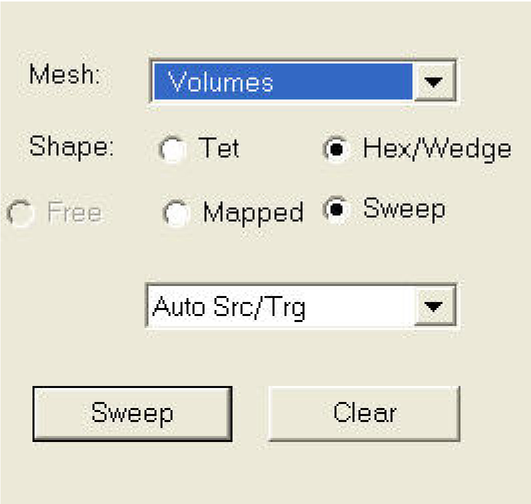
You'll get the following warning:



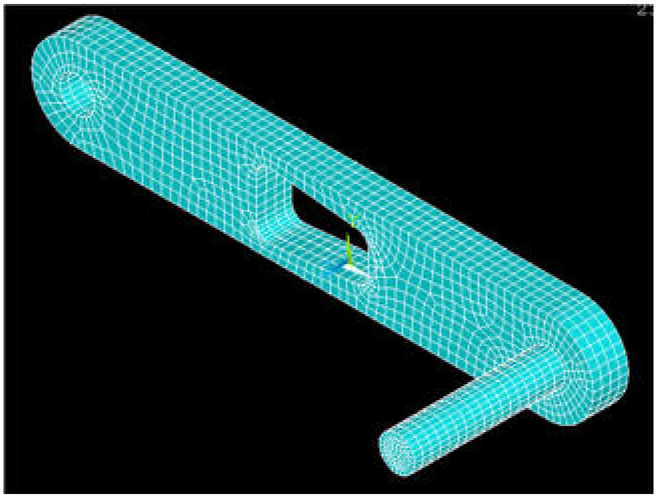
Elements that exceed shape warning limits can lead to degraded accuracy. Here it is a minor concern since only 1 element out of 682 is causing the warning. So it is reasonable to press on. In general, it is always a good idea to pay close attention to the warnings and understand their effect on your solution. As a veteran in these things, I can attest that ignoring warnings can come back to bite you in incovenient parts of the anatomy. Close the warning window.

> In the above, we chose the front faces of the crank arm and pedal shaft as the surface meshes for sweeping. However, we have found that for other crank geometries, when meshing using the MESH200 elements, it is a good idea to choose the two *back* faces of the crank arm and pedal shaft that are flush with each other (i.e. the negative-z faces). This ensures that the nodes around the circumference of the circle on the two parts will match up and may prevent problems in sweeping the volume elements.

Bring up the MeshTool again. Click ***Set*** next to ***Global*** under ***Element Attributes***. Set the ***TYPE*** to ***SOLID45*** and click ***OK***. We want four layers of mesh elements to span the thickness of the crank, so the desired element edge length in the sweep direction is (0.5 /4) = 0.125 in. Under ***Size Controls,*** click the ***Set*** button next to ***Global***. Enter an ***element edge length*** of 0.125 and click ***OK***. We will now sweep, i.e. extrude, the surface meshes created above across the corresponding volumes. Select ***Volumes*** to be meshed with a ***Hex*** shape along with the ***Sweep*** option as shown below. Make sure ***Auto Src /Trg*** is selected; this will automatically pick a source (Src) surface mesh and sweep/extrude it to a target (Trg) surface.



Click ***Sweep*** and ***Pick All*** to sweep-mesh both volumes. ANSYS will extend our previous surface meshes across the corresponding volumes.



ANSYS issues a warning that 5 out of 3986 elements violate shape warning limits. Since the number of "bad" elements is small, this is a minor concern and we'll press on. But keep in mind that what we'll obtain is a reasonable first-cut solution but it will not be the final word. For that, you'll have to show that the solution is independent of the mesh. Close the warning window and the Meshtool.

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