

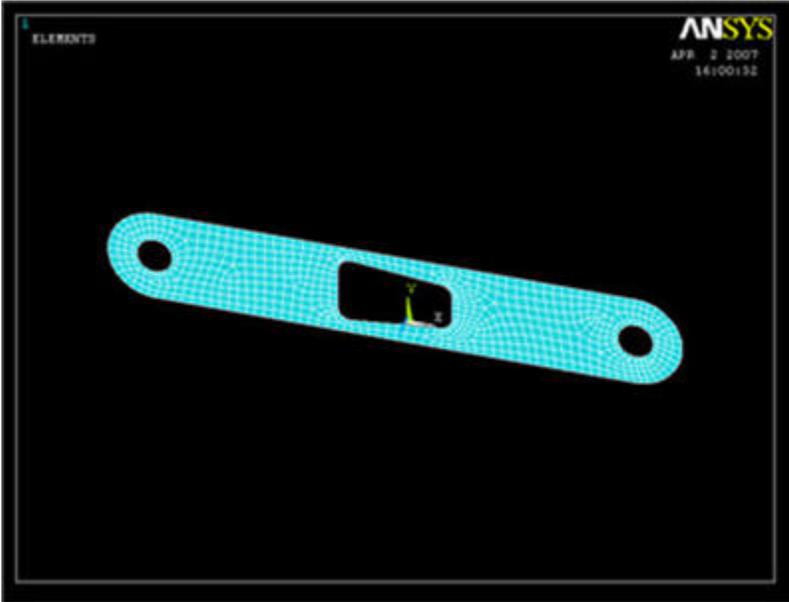
ANSYS - Crank 5 Mesh Geometry Content

Step 5: Mesh geometry

Bring up the *MeshTool*:

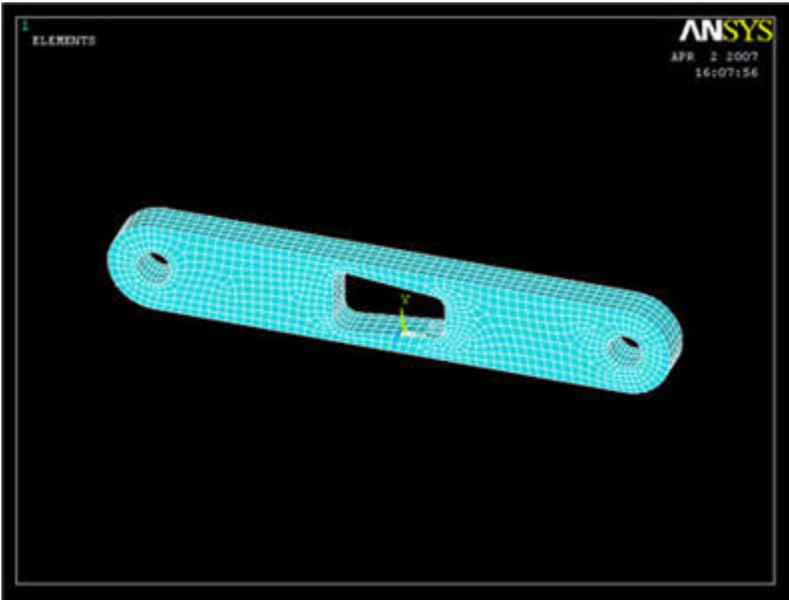
Main Menu > Preprocessor > Meshing > MeshTool

Click the Set button for next to Global Element Attributes. Make sure that the TYPE is MESH200 and click OK. Check the box next to Smart Size and drag the slider to a size of 4. Click the Set button under Global Size Controls. Enter an element edge length of 0.12 and click OK. Select Areas to be meshed with a Quad shape using the Free mesher. Click Mesh. Pick the front face of the crank and click OK. You will now see:



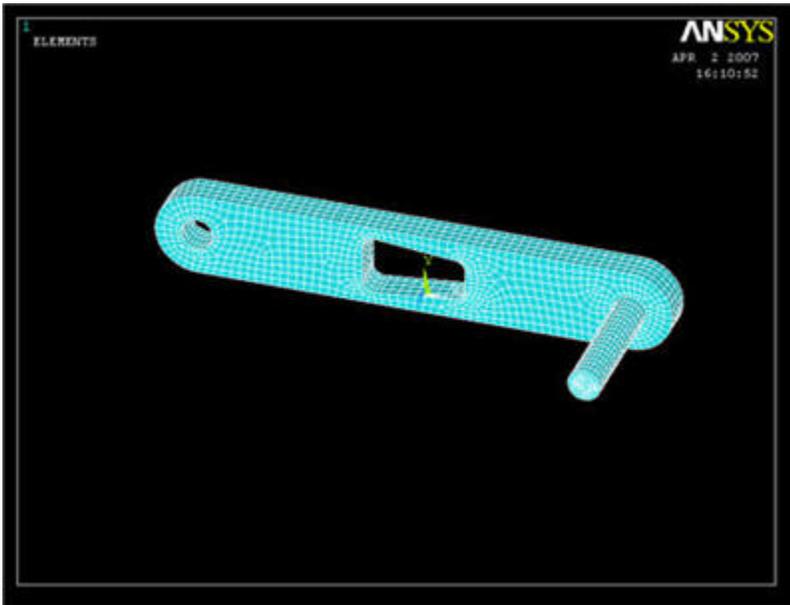
The specified smart size 4 and edge length of 0.12in are the result of an iterative process. ANSYS uses a sizing algorithm that creates smaller mesh elements in areas of potentially high stress concentration (e.g. curved regions) to improve solution accuracy. Selecting a smaller (finer) smart size will decrease element size close to the cutout curves. Specifying an edge length provides a uniform size for the rest of the area being meshed. In this case, since the material above and below the cutout act as beams, we want at least two mesh elements in the vertical direction. Feel free to experiment with the mesh tool parameters.

Bring up the MeshTool again, and now Set the mesh type to SOLID45. Under Global Size Controls, click Set. We want four layers of mesh elements to span the thickness of the volume, so we will enter a SIZE Element edge length of $(0.5 \text{ in} / 4) = 0.125$. Click OK. Since we had created a MESH200 on the face of the crank, we can now simply sweep that mesh across the volume. Choose Volumes with a Hex Shape/Wedge to be meshed. Make sure Auto Src /Trg is selected and click Sweep. Now all we have to do is pick the crank volume, and ANSYS will extend our previous mesh across the volume.



You can always see the rest of your model by selecting Utility Menu > Plot > Volumes.

Now repeat the procedure with the crank shaft. Remember that the shaft element size must be the same as the crank so that the shaft elements and crank elements in the hole line up. (You can ignore any errors that may pop up regarding element size. ANSYS is complaining because the shape of some of the quadrilateral elements in high stress areas have a non-ideal element shape) Your final meshed model should look like the following. We're almost ready to solve the problem.



Save Your Work

Toolbar > SAVE_DB