ANSYS 11 - Crank

Unknown macro: {rate}

Unknown macro: 'alias'

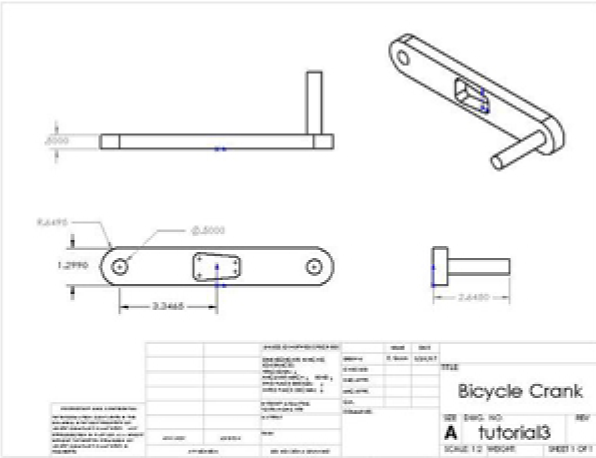
Author: Rajesh Bhaskaran, Cornell University

Note Title

The following ANSYS tutorial is under construction.

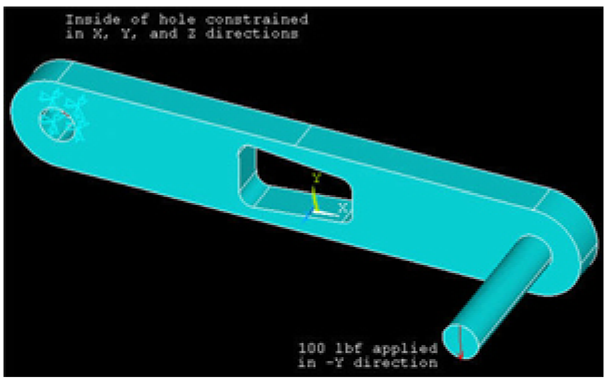
Problem Specification

A preeminent bicycle company is disappointed with the negative feedback they have received on their latest model, and they have pinpointed the problem to an outdated bicycle crank design that they assumed would still withstand typical loads. To protect their reputation, they have outsourced the task of analyzing the crank to you, providing you with the geometry of the bicycle crank and attached pedal shaft shown below. The dimensions are given in inches. The material they selected has an Young's modulus E=2.8x10^7 psi and Poisson ratio =0.3.



Higher Resolution Image

Using ANSYS, determine the mechanical response due to a load of 100 lbf applied vertically downward at the end of the pedal shaft as shown in the figure below. Assume that the crank is attached rigidly to a fixed shaft fitted into the hole near the left end of the crank. This means you can constrain the surface of the left hole in *X*, *Y* and *Z* directions as indicated below.



Calculate the deflection, strain and stress distributions in the crank/pedal shaft combination for this loading condition. Use the ANSYS results to evaluate the degree of stress concentration in the vicinity of the cut-out in the crank geometry.

Go to Step 1: Start-up and preliminary set-up

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Step 1: Start-up and preliminary set-up

Start ANSYS

Create a folder called `crank` at a convenient location. We'll use this folder to store files created during the ANSYS session.

Start > All 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 `crank` as your ***Job Name***. The job name 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 *crank.db* in your working directory.

For this tutorial, we'll use the default values for the other fields. Click ***Run***. This brings up the ANSYS interface. To make best use of screen real estate, move the windows around and resize them so that you approximate this screen arrangement. This way you can read instructions in the browser window and implement them in ANSYS. Note that this tutorial has been formatted to fit in a skinny browser window. If your monitor screen is small, you can use *Alt +Tab* keys to conveniently switch between the ANSYS and browser windows (this trick works in Microsoft Windows).

You can resize the text in the browser window to your taste and comfort:

In Internet Explorer, select **Menubar > View > Text Size**, then choose the appropriate font size.

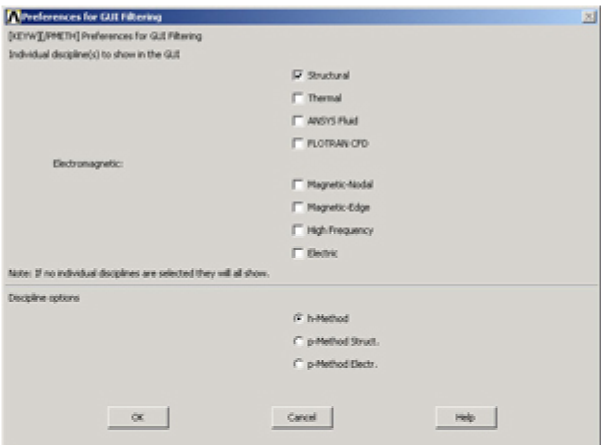
In Mozilla Firefox, select **Menubar > View > Zoom**.

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.

Go to Step 2: Specify element type and constants

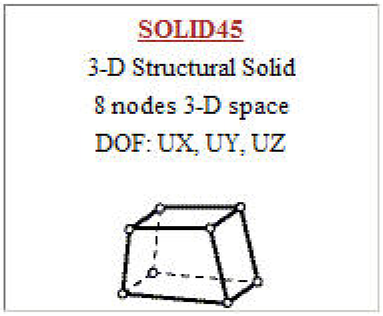
See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Step 2: Specify element type and constants

Specify Element Type

We next select the appropriate element type(s) for our problem from a large list of about 200 candidates. Consider this as equivalent to rifling through a sizable toolchest, picking out one or more tools and placing them on a table for later use (in step 5, in our case). To see which element types are appropriate for this problem, bring up the pictorial summary of element types: **Utility menu > Help > Help Topics**. Search for "pictorial summary" and double-click on the search result titled ***3.2 Pictorial Summary***. Click on the link to ***SOLID Elements***. These are the element types you can use to mesh a solid volume. Check out the ***Solid45*** element type. It is a brick-shaped element of the type referred to as "hexahedral" or "hex". It has a node at each corner and each node has three degrees of freedom: displacement in the *x*, *y* and *z* directions.

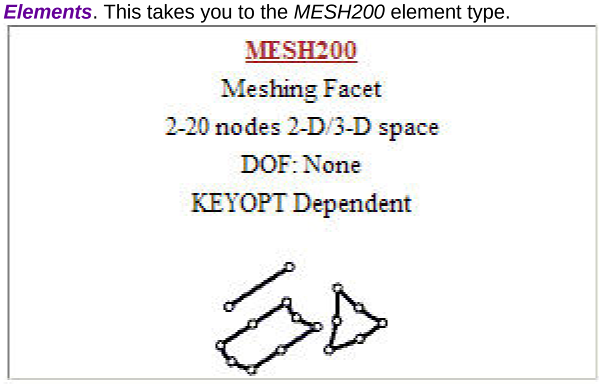


Click on the ***SOLID45*** link in the pictorial summary. This takes you to the help page for this element. Read through the juicy information at the beginning of this help page. Note that there are no real constants to be defined in our case. Click on ***SOLID45*** in the statement "See SOLID45 in the Theory Reference for ANSYS and ANSYS Workbench for more details about this element" which appears near the top of the help page. Click on ***Equation 12-188***. This shows you the shape function for the element i.e. the equation used to determine the displacement at a general point within the element from the displacement values at the 8 nodes.

We will create our volume mesh in two steps:

1. Mesh the front surfaces of the crank as well as the pedal shaft.
2. Extrude these surface meshes to get the corresponding volume meshes.

This is analogous to creating a sketch and then extruding while making a solid in a CAD package. Since we cannot mesh *surfaces* with *SOLID45*, we need an additional element type called *MESH200*. Go back to the pictorial summary of element types. Scroll to the top of the page and click on the link to ***MESH Elements***. This takes you to the *MESH200* element type.



Click on the ***MESH200*** link in the pictorial summary to see the help page for this element. You see the following information:

"MESH200 is a "mesh-only" element, contributing nothing to the solution. This element can be used for the following types of operations:

- Multistep meshing operations, such as extrusion, that require a lower dimensionality mesh be used for the creation of a higher dimensionality mesh"

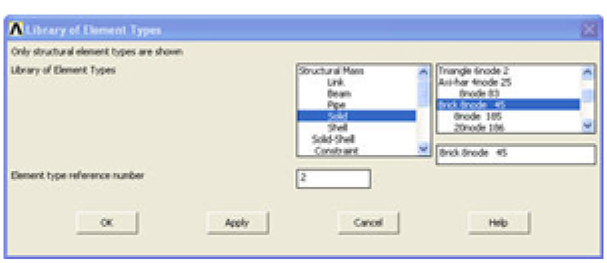
In our case, meshing the two front surfaces with *MESH200* elements can be thought as going to these surfaces and marking out points and lines with a pen to show ANSYS where to put the corresponding *SOLID45* nodes and element faces. The *SOLID45* nodes and elements are actually placed at these pre-marked locations in the extrusion step. In effect, *MESH200* provides greater control over the mesh without actually contributing to the solution.

Referring to Figure 200.1 in the *MESH200* help page, we see that this element type comes in 12 different flavors. For our purposes, we will be using the ***3-D quadrilateral with 4 nodes*** option since this corresponds to an element face of *SOLID45*. The help page indicates that this option is selected by setting ***KEYOPT(1) = 6***. Note that there are no real constants to be defined for *MESH200*.

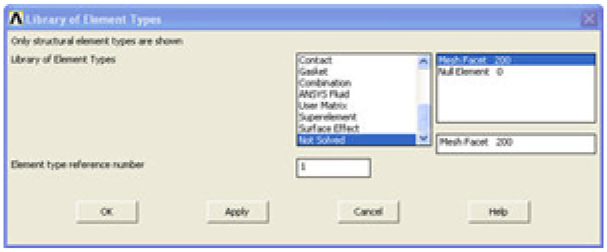
Minimize the ANSYS *Help* window. Select

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

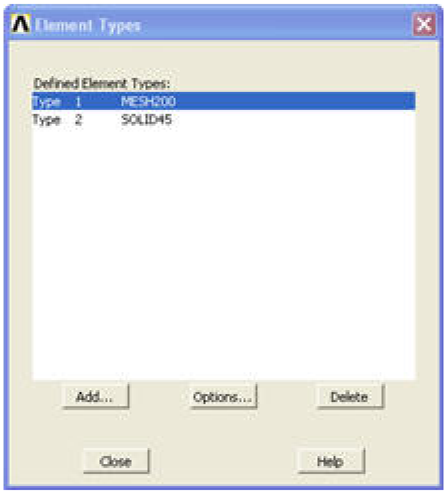
Pick ***Structural Mass Solid*** in the left field and ***Brick 8node 45*** in the right field. Click ***Apply*** to select this element.



Scroll down the left field, pick ***Not Solved*** in this field and then ***Mesh Facet 200*** in the right field. Click ***OK*** to select this element.



The *Element Types* window should list two types of elements: ***MESH200*** and ***SOLID45***.



In order to set the *3-D quadrilateral with 4 nodes* option for *MESH200*, click on ***MESH200*** and then ***Options ...*** in the above menu. Then, select ***QUAD 4-NODE*** next to ***Element shape and # of nodes K1***. Click ***OK***.

Close the *Element Types* menu.

Specify Element Constants

There are no real constants to be set for either element type as noted above.

Save your work

Toolbar > SAVE_DB

Go to Step 3: Specify material properties

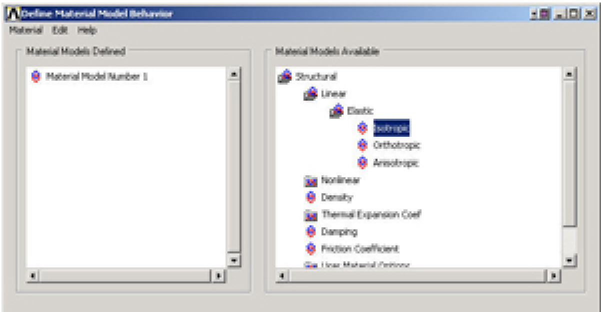
See and rate the complete Learning Module

Go to all ANSYS Learning Modules

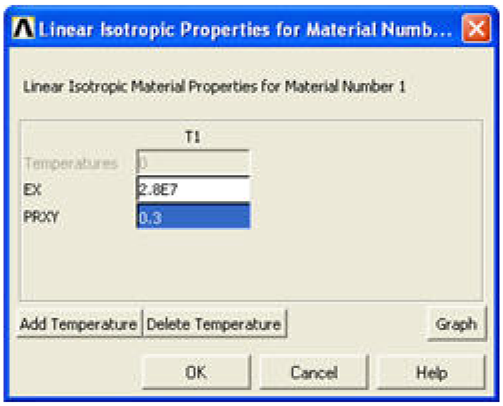
Step 3: Specify material properties

Main Menu > Preprocessor >Material Props > Material Models

In the *Define Material Model Behavior* menu, double-click on ***Structural***, ***Linear***, ***Elastic***, and ***Isotropic***.



Enter `2.8E7` for Young's modulus ***EX***, `0.3` for Poisson's Ratio ***PRXY***. Click ***OK***.



To double-check the material property values, double-click on ***Linear Isotropic*** under ***Material Model Number 1*** in the *Define Material Model Behavior* menu. This will show you the current values for *EX* and *PRXY*. ***Cancel*** the *Linear Isotropic Properties* window.

This completes the specification of ***Material Model Number 1***. When we mesh the geometry later on, we'll use the reference no. 1 to assign this material model. Close the *Define Material Model Behavior* menu.

Save your work

Toolbar > SAVE_DB

Go to Step 4: Specify geometry

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Step 4: Specify geometry

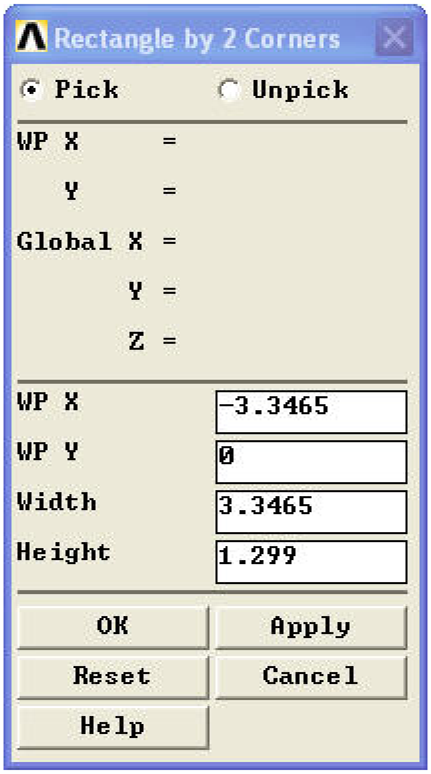
Note that you can import geometry from a CAD package such as Pro/Engineer or SolidWorks into ANSYS by following these instructions.

Since the geometry excluding the cutout region is symmetric with respect to the vertical centerline, we will model half of the crank and then mirror the other half to complete the crank body. Then we will create the cutout from a set of keypoints.

Create a Rectangular Area

Main Menu > Preprocessor > Modeling > Create > Areas > Rectangle > By 2 Corners

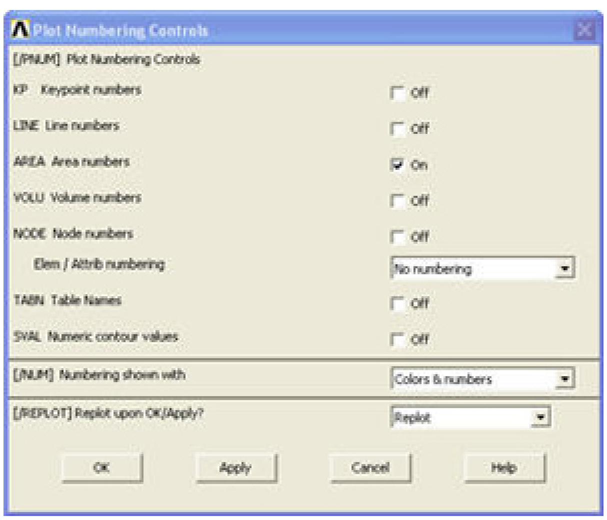
Enter the values as shown below. Click ***OK***.



It may be helpful to turn on area numbering to identify the different areas you create.

Utility menu > PlotCtrls > Numbering ...

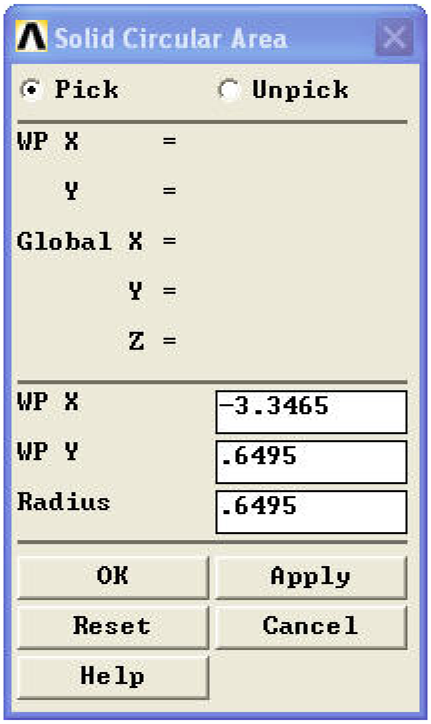
Check the box next to ***AREA Area numbers*** to turn on area numbering. Click ***OK***.



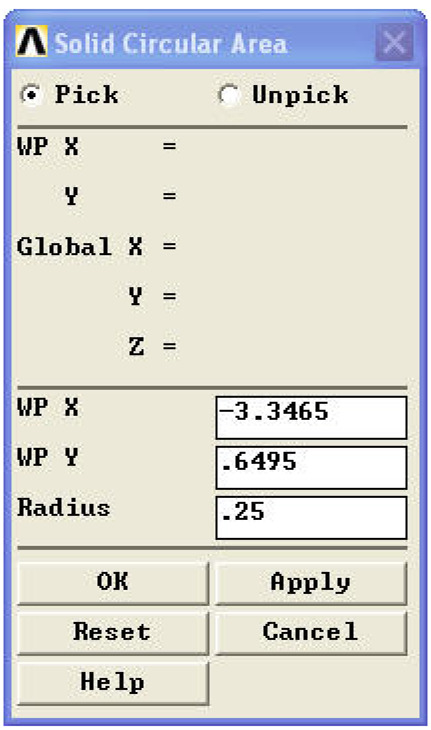
Create Circular Areas

Main Menu > Preprocessor > Modeling > Create > Areas > Circle > Solid Circle

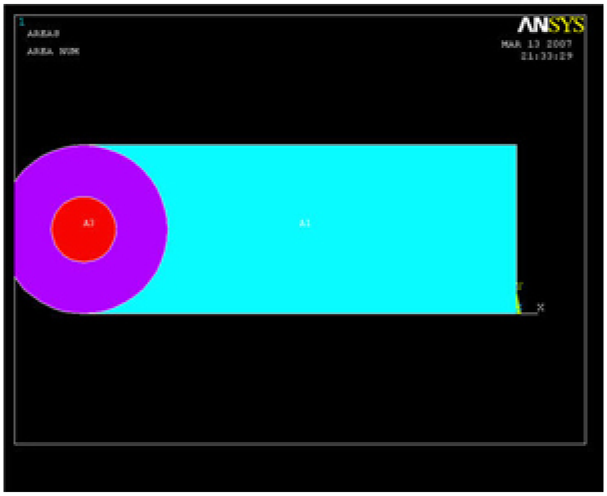
Enter the values as shown below. Click ***Apply***. This creates the rounded end of the crank.



Enter the new set of values shown below. Click ***OK***. This creates the area for a hole.



Your window should look something like the picture below. You can click **Utility Menu > Plot > Replot** or click on the ***Fit View***  button on the right toolbar to refresh the view.

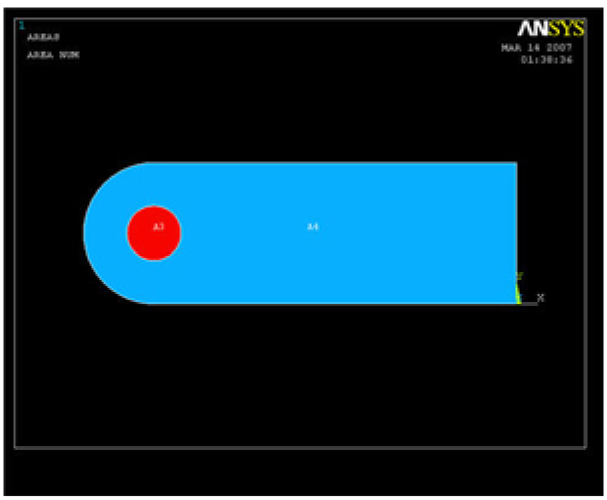


To correct any mistakes, you must click **Main Menu > Preprocessor > Modeling > Delete > Areas Only** and then pick each area you want to remove. The mouse pointer will show an up arrow for picking areas and a down arrow for un-picking areas. Right-click to switch between pick and unpick mode. When you have made all your selections, click ***OK***. Click **Utility Menu > Plot > Replot** to refresh the view.

Add Areas

Main Menu > Preprocessor > Modeling > Operate > Booleans > Add > Areas

Pick the rectangular and large circular areas. Click OK. (This is where the area numbering may come in handy) The result should look like the image below.

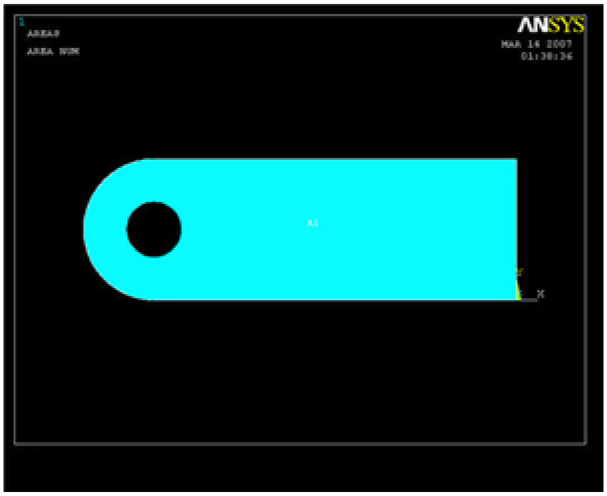


Subtract Hole Area

Now we create the hole by subtracting the round area from the rest of the crank.

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

First pick the body of the crank and click ***OK***. Then pick the hole, and click OK again. The result is shown below.



Reflecting the Area

To create the other half of the crank, we will reflect the current area about the Y-Z plane.

Main Menu > Preprocessor > Modeling > Reflect > Areas

Click on ***Pick All***. The Y-Z plane is selected by default, so click ***OK***. All that's left now is to add the two halves of the crank together.

Main Menu > Preprocessor > Modeling > Operate > Booleans > Add > Areas

Click on ***Pick All***.

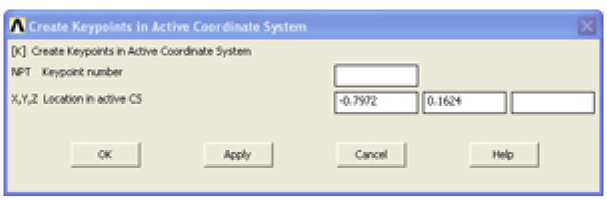
Higher Resolution Image

Creating Keypoints for the Cut-out Region

Since the material to be removed in the middle of the crank is an irregular shape, we will define some keypoints in order to create and subtract this area.

Main Menu > Preprocessor > Modeling > Create > Keypoints > In Active CS

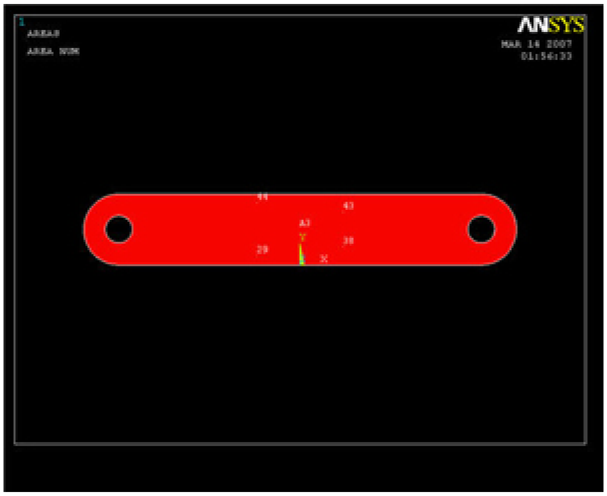
Enter the values shown below and click ***Apply***. Leave the keypoint number blank to let ANSYS automatically assign an ID number. Alternatively, you may specify your own number (as long as that keypoint isn't already taken). To see a list of existing keypoints, go to **Utility Menu > List > Keypoint > Coordinates Only**. The Z location is left blank because it is 0 by default.



Points to add:

```
(-0.7972, 0.1642)
(0.7972, 0.3248)
(0.7972, 0.9744)
(-0.7972, 1.1368)
```

The result:



Creating Lines and Fillets from Keypoints

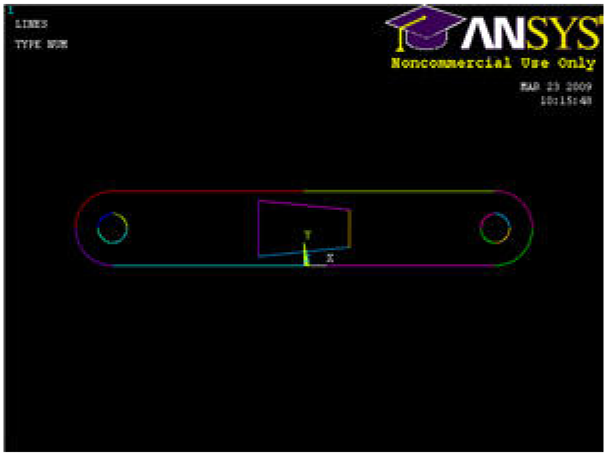
Main Menu > Preprocessor > Modeling > Create > Lines > Lines > Straight Line

Select pairs of points by clicking on beginning and end keypoints. You will notice that after clicking on the first point, ANSYS will predict where you want the line to be drawn to. Select four lines to form a quadrilateral at the center of the crank, then click ***OK***.

Don't panic if all the lines disappear. In the current view, only areas are displayed. Switch to line view by:

Utility Menu > Plot > Lines

The result:



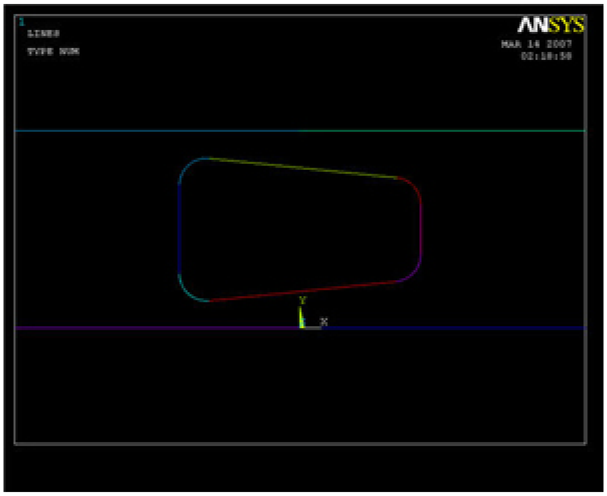
Higher Resolution Image

Next, we want to fillet the corners, as specified in the drawing.

> **You can zoom in and out by using the mouse wheel or clicking on the appropriate buttons on the right toolbar (magnifying glass with + or -).**

Main Menu > Preprocessor > Modeling > Create > Lines > Line Fillet

Pick two lines that meet at a corner where you want to put a fillet, then click ***OK***. Enter a ***Fillet radius*** of 0.177, and click ***Apply***. Repeat for the other three corners of the quadrilateral. Compare results with image below.



Finishing the Crank Face

All that's left now is to create a new area from the filleted quadrilateral region, and then subtract it from the rest of the crank face.

Main Menu > Preprocessor > Modeling > Create > Areas > Arbitrary > By Lines

In the Pick window, select ***Loop***. Click on any of the line segments that we have just created and the entire cutout region should be selected. Click ***OK***. Switch back to area view by going to

Utility Menu > Plot > Areas

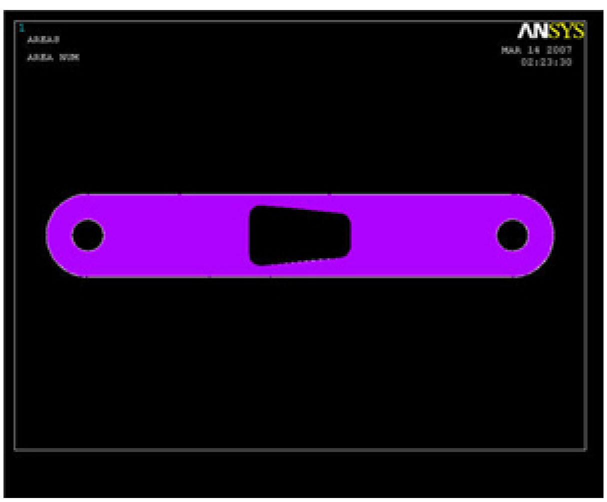
Subtract out the new area from the rest of the crank by the same procedure as before.

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

Select the rest of the crank face, then ***OK***.

It will be helpful to hold down the left mouse-button while picking an area, as an area changes color when it is selected. Move the pointer until the desired area is highlighted, then release the button. Finally, select the new cut-out area, then press OK again.

The result:



Creating the Volume

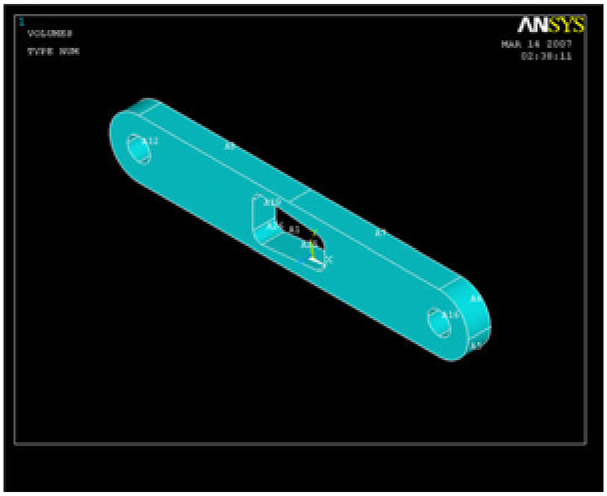
We will now make the face 3-D by extruding it by a given offset distance, similar to modeling in CAD.

Main Menu > Preprocessor > Modeling > Operate > Extrude > Areas > By XYZ Offset

Click ***Pick All***. In the following window, change the DZ offset to `0.5`. Click ***OK***. To see your finished work, go to

Utility Menu > Plot > Volumes

Then click on the isometric view button  on the right toolbar.

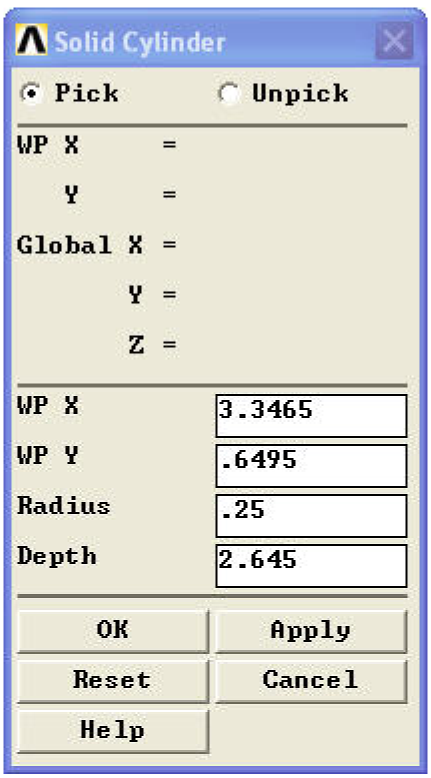


Higher Resolution Image

Creating the Pedal Shaft

Main Menu > Preprocessor > Modeling > Create > Volumes > Cylinder > Solid Cylinder

Enter the following values and press ***OK***.



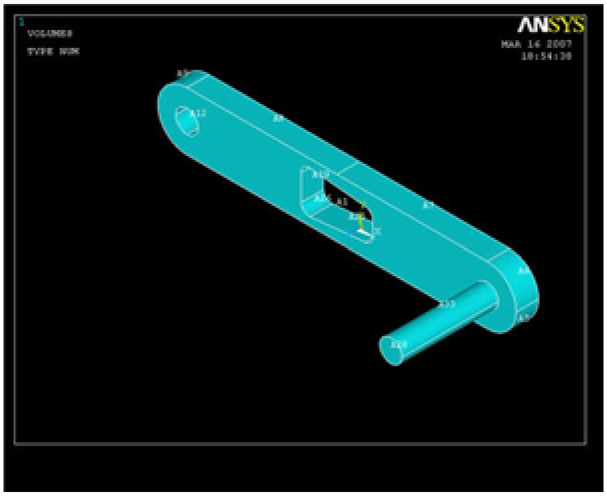
We must now glue the shaft to the crank. The reason for using "glue" instead of performing a boolean add on the volumes is to maintain two discrete parts. This provides more flexibility in modeling, as it can allow for different materials and meshes. Note: If glue is not used, the two pieces will be independent of each other and the solution will be incorrect.

Main Menu > Preprocessor > Modeling > Operate > Booleans > Glue > Volumes

Click ***Pick All*** to glue our two volumes together. Note that there are no visual indicators of whether or not the volumes have been glued. You should check the Command Window and look for the "GLUE VOLUMES" command.

```
GLUE VOLUMES
     INPUT VOLUMES =      1     2
     INPUT VOLUMES WILL BE DELETED IF POSSIBLE
     OUTPUT VOLUMES =      1     3
```

Your complete crank model should now look like this:



Higher Resolution Image

Save Your Work

Toolbar > SAVE_DB

Go to Step 5: Mesh geometry

See and rate the complete Learning Module

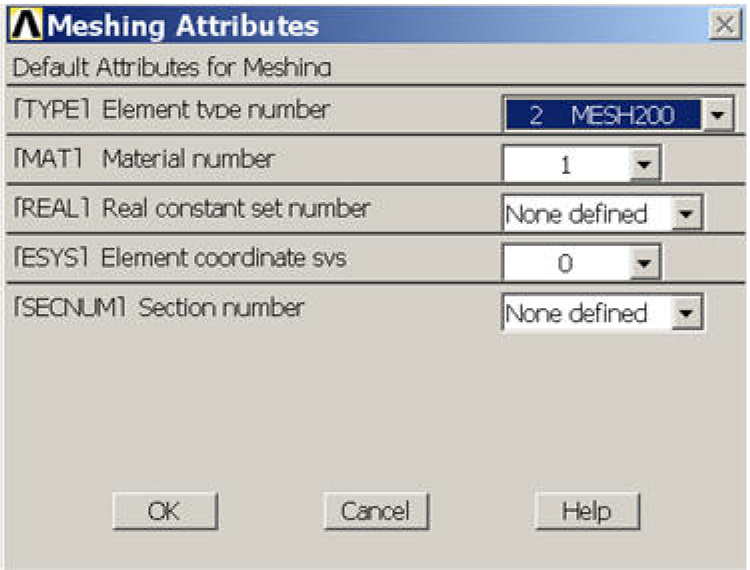
Go to all ANSYS Learning Modules

Step 5: Mesh geometry

Bring up the *MeshTool*:

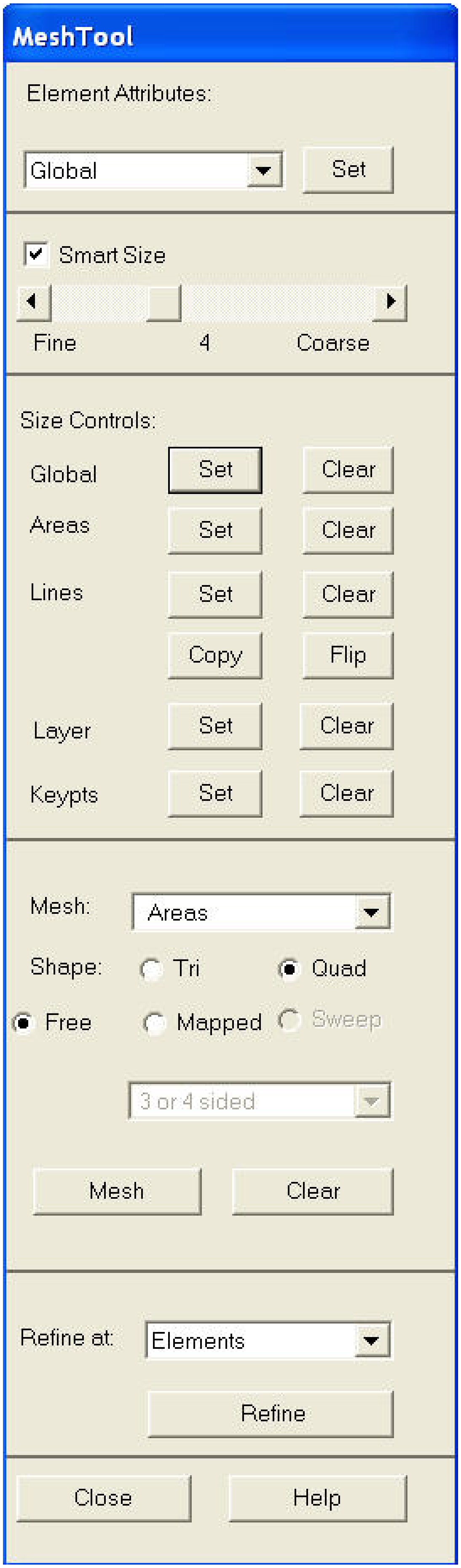
Main Menu > Preprocessor > Meshing > MeshTool

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***.

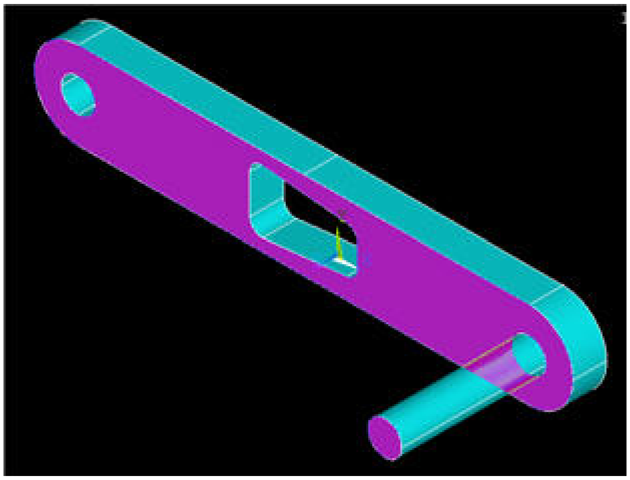


According to the ANSYS manual, "Smart element sizing (SmartSizing) is a meshing feature that creates initial element sizes for free meshing operations. SmartSizing gives the mesher a better chance of creating reasonably shaped elements during automatic mesh generation ... The SmartSizing algorithm first 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.

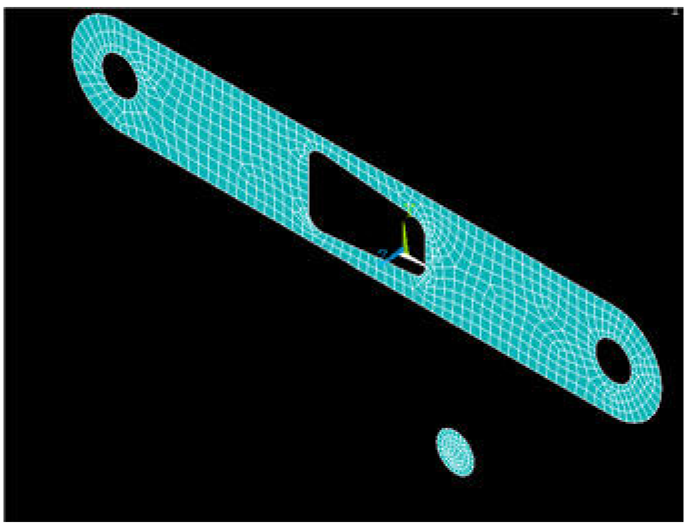
In order to have a little more control over what mesh ANSYS creates for us, we will set the *starting* element size for SmartSizing rather than use the default. Smartsizing will take this starting element size and modify/vary it over the geometry to account for curvature and corners. Under ***Size Controls,*** 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

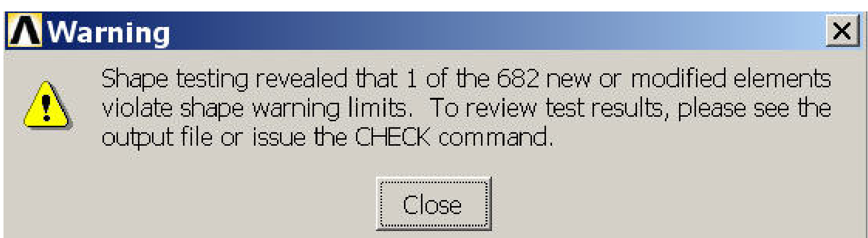
Select ***Areas*** to be meshed with a ***Quad*** shape using the ***Free*** mesher. Click ***Mesh***. Pick the front face of the crank and the pedal shaft.



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



You'll get the following warning:

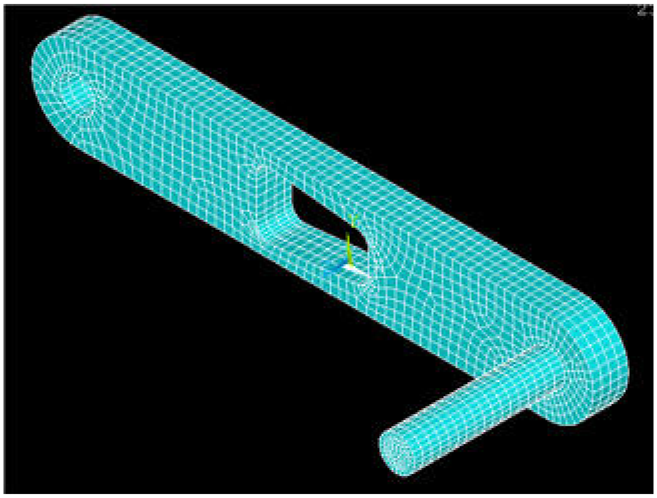


Elements that exceed shape warning limits can lead to degraded accuracy. Here it is a minor concern since only 1 element out of 682 is causing the warning. So it is reasonable to press on. In general, it is always a good idea to pay close attention to the warnings and understand their effect on your 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

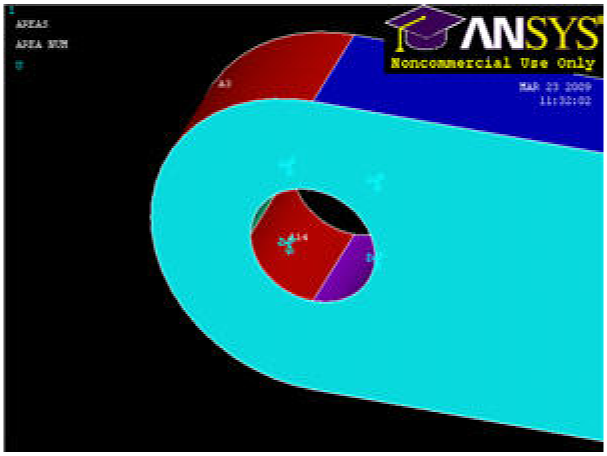
Step 6: Specify boundary conditions

We have two loading conditions to specify. First we must fix the hole where the crank would attach to the bicycle. Then we apply our loading condition of 100 lb on the end of the shaft.

Fixed End

Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Areas

It will be helpful to see the areas we're constraining, so select **Utility Menu > Plot > Areas**. We can see that the hole consists of multiple areas (4, in fact). Hold down the left-click and you can see that there are 4 surfaces that make up the inside of the hole. Pick all 4 and click ***OK***. Select ***All DOF*** and click ***OK***. The displacement value can be left blank as it defaults to 0. Symbols appear at the centers of each constrained area indicating that the area is constrained in three directions.



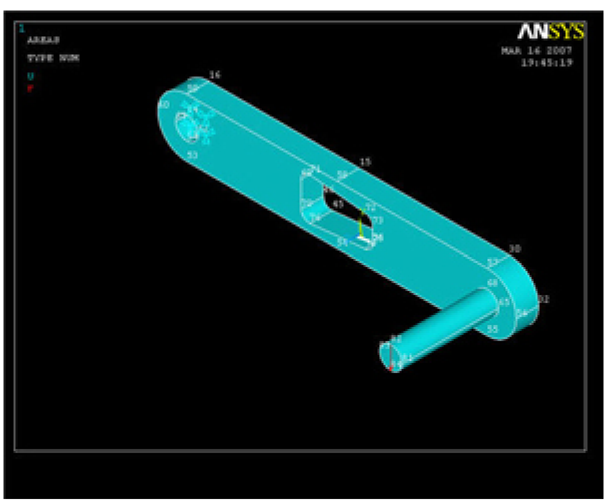
Higher Resolution Image

Force on Shaft

Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Force/Moment > On Keypoints

Select Utility Menu > PlotCtrls > Numbering ... and turn On ***Keypoint Numbers***. Click ***OK***. Notice that there is conveniently a keypoint at the tip of the shaft, and pick this point to apply the force. Click ***OK***. From the orientation of our axes, we want a constant force in the ***FY*** direction with a value of `-100`. Click ***OK***.

What the model looks like now:



Higher Resolution Image

Now let's see some results!

Save Your Work

Toolbar > SAVE_DB

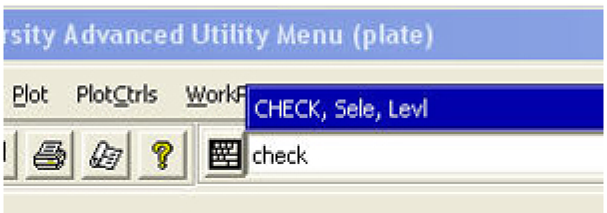
Go to Step 7: Solve!

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Step 7: Solve!

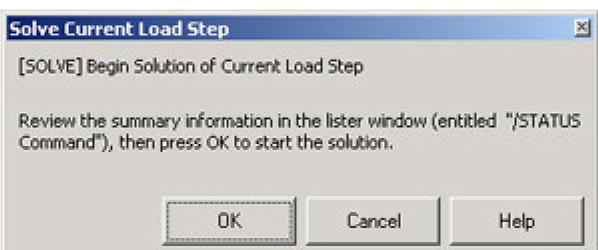
Before we start the solution, we should check our model for errors. Enter `check` in the *Input* window and press Enter.



All warnings and errors found will be displayed in the Output Window. There are no errors but you will see the warnings regarding element shapes that we encountered before. So we're finally ready to kick back and let ANSYS do some of the work: assembling the local and global stiffness matrices and inverting the global system to determine the displacements at the nodes.

Main Menu > Solution > Solve > Current LS

Click ***OK*** in *Solve Current Load Step* menu.



ANSYS should cheerfully report "Solution is done!"

Verify that ANSYS has created a file called *crank.rst* in your working directory. This file contains the results of the (previous) *solve*.

Go to Step 8: Postprocess the results

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

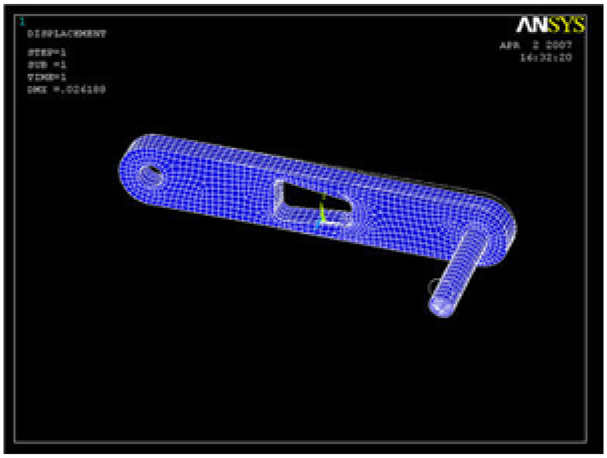
Step 8: Postprocess the Results

Plot Deformed Shape

Main Menu > General Postproc > Plot Results > Deformed Shape

Select ***Def + undef edge*** and click ***OK***.

This plots the deformed and undeformed shapes in the *Graphics* window. The maximum deformation *DMX* is 0.026148 inches as reported in the *Graphics* window. We should check that our results make sense. It appears that the boundary conditions have been satisfied as the tip of the shaft moves downward and the hole at the other end of the crank is held in place.



Animate the deformation

Utility Menu > PlotCtrls > Animate > Deformed Shape...

Select ***Def + undeformed*** and click ***OK***. Select ***Forward Only*** in the *Animation Controller*. This is also a good way to check the boundary conditions have been applied correctly. Close the Animation Controller.

Plot Nodal Solution of von Mises Stress

For a quick refresher on von Mises stress, click Help. Search for von mises and click on the result ***2.4 Combined Stresses and Strains*** (this is lower down among the search results). Read through section 2.4.2.

Main Menu > General Postproc > Plot results > Contour Plot > Nodal Solu

Select ***Nodal Solution > Stress > von Mises stress*** and click ***OK***. To change the range of stresses displayed, go to
Utility Menu > PlotCtrls > Style > Contours > Uniform Contours ...

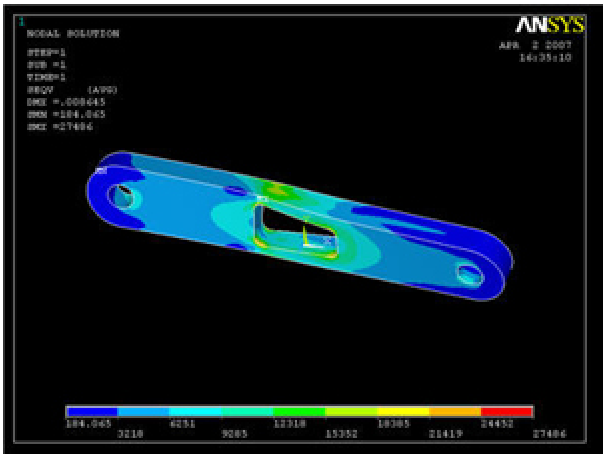
and select ***User specified***. Specify a range of minimum 0 and maximum 25000. We can now see more color variation in the model, and easily pick out the red areas.

When you plot the "Nodal Solution", ANSYS obtains a continuous distribution as follows:
1. It determines the average at each node of the values of all elements connected to the node.
2. Within each element, it linearly interpolates the average nodal values obtained in the previous step.
This procedure is in effect a smoothing operation.

The stress concentration located at the tip of the shaft can be ignored as the force is applied as a point load. Let's look at the results just for the crank by deselecting the elements within the pedal shaft volume. Go to

Utility menu > Select > Entities ...

Select ***Volumes***, ***By Num/Pick***, ***From Full*** and click ***Apply***. Pick the crank volume and click ***OK***. After we've selected a volume, we must next select all the elements in this volume. In the Select Entities window, select ***Elements***, ***Attached to***, ***Volumes*** and click ***Apply***. Click ***Replot*** to display the new selection. Notice the deformation is exaggerated, revealing that deformation is primarily caused by torsion.



Higher Resolution Image

To select the whole model again, go to **Utility Menu > Select > Everything**.

Comparing the $_{xx}$ Stress with von Mises Stress

To verify that the bending stress in the crank is relatively insignificant, we can compare the element $_{xx}$ solution with the elemental von Mises solution.

General Postproc > Plot Results > Contour Plot > Element Solu

Click on Stress, then X-Component of stress , then Apply.

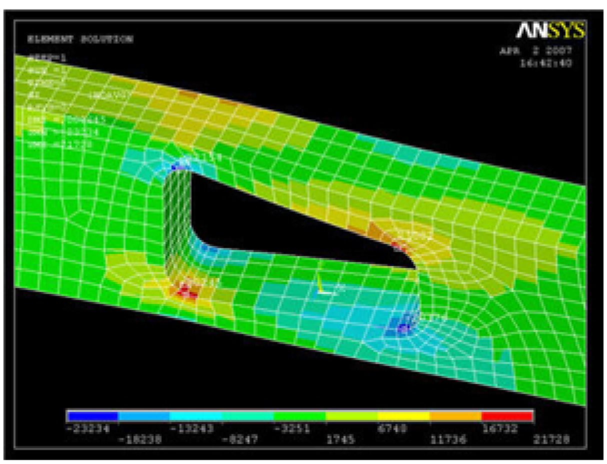
If grey areas are appearing in your contour plots, you should go to **Utility Menu > PlotCtrls > Style > Contours > Uniform Contours ...**, select ***Auto calculated***, and click ***OK***.

Notice that the top-left and bottom-right corners of the cutout area are now blue, and that the scale has been readjusted to show that blue is now a large negative stress value. If this were a case of pure bending, we would expect the top of the crank to be in tension, not compression!

To find out information about specific points on the model, go to

General Postproc > Query Results > Subgrid Solu

Select ***Stress***, ***X-direction SX***, and click ***OK***. The picking window will appear, and you can click on any point in the model. Click ***OK*** when finished.



Compare the stress values with the von Mises stress. (Click on von Mises stress, then ***OK***)

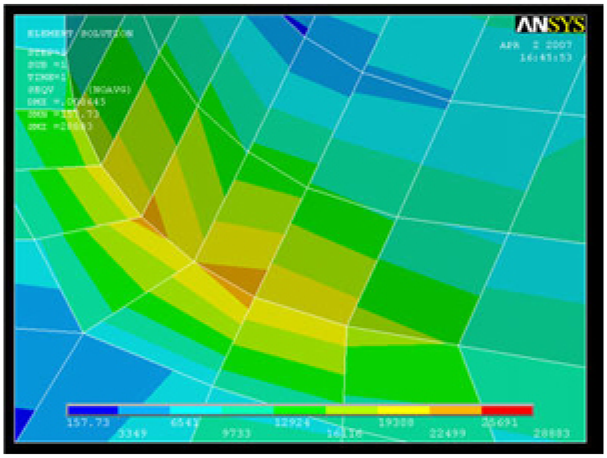
Investigate the Stress Concentration

Let's zoom in on the red area. Use the mouse wheel to zoom in and out in the view area. Some other viewing functions: Holding down the Ctrl key and the left mouse button allows you to pan the view, while holding the Ctrl key and the right mouse button allows you to rotate the view. Hold down the right mouse button and draw a rectangle to zoom in on a specific region.

Recall that the nodal solution shows smoothed stress values. Let's compare the nodal solution with the element, i.e. non-smoothed, solution.

General Postproc > Plot Results > Contour Plot > Element Solu

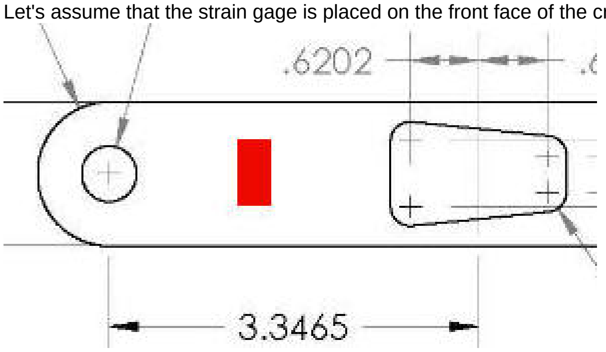
Click on ***Stress***, then ***von Mises Stress***, then the ***OK*** button. In the vicinity of the cut-out corners, there are fairly significant discontinuities in the von Mises stress across adjacent elements. This suggests that we need to refine the mesh at least in this region. This is done in the next step.



Calculate Average Strain in Specified Area

In a perfect world, we would be able to validate the ANSYS results by comparing them with strain gage measurements at selected locations. We unfortunately don't have strain gage measurements for this particular geometry but will anyway show you the process by which you can calculate the average strain over an area where a strain gage would be placed. This will help prepare you to compare your ANSYS results with strain gage measurements for a different geometry for which you may have experimental data.

Let's assume that the strain gage is placed on the front face of the crank (*z*=0.5") roughly halfway between the left hole and the cutout as shown below.



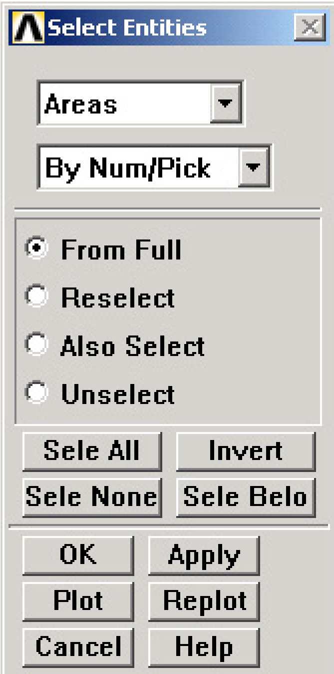
In the coordinate system in our model, let's say this area is given by -1.77" x -1.57" and 0.45" y 0.85". To find the average strain in this area, we will select all nodes that lie in this area and then list the strain values at these nodes. You can copy these values over to Excel or MATLAB to find the average value.

ANSYS provides extensive capabilities, referred to as "select logic", for selecting a subset of the full model using various criteria. We'll use select logic to select the nodes on the front face of the crank. We'll first select the area corresponding to this face.

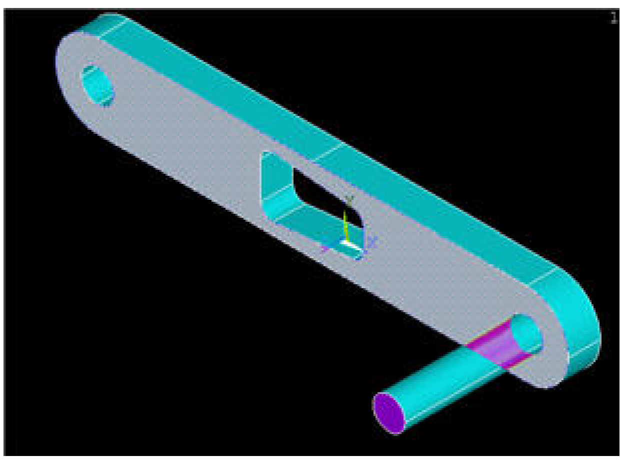
Utility Menu > Plot > Areas

Utility Menu > Select > Entities

Select ***Areas*** from the pull-down menu at the top. Make sure ***By Num/Pick*** is selected below that. Click ***Apply***.



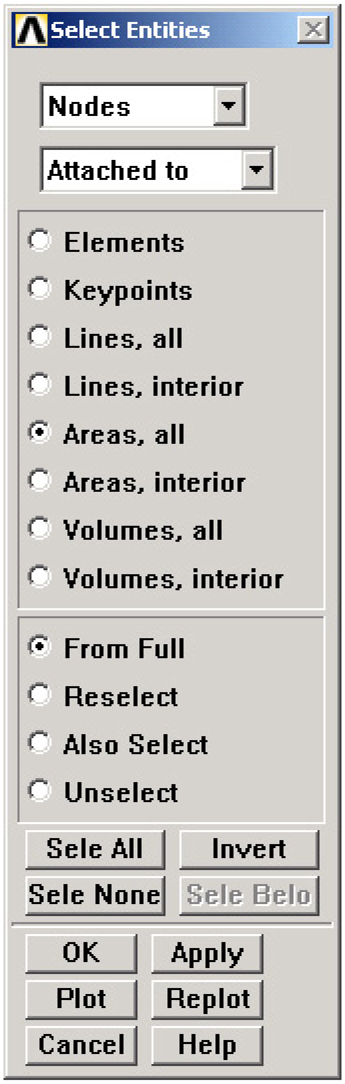
Hold down the left mouse button until the front face is picked. Click ***OK*** in the pick menu.



Only the area corresponding to this face is selected currently. Verify this by clicking ***Replot*** in the *Select Entities* menu (this replots areas).

One key thing to remember about ANSYS' "select logic" is that the various entity types (areas, volumes, nodes, elements, etc) are selected independently. So all nodes are still "selected", not just the ones that are located on the front face of the crank. Verify this: **Utility Menu > Plot > Nodes**.

We next select the nodes attached to the previously selected area. In the *Select Entities* menu, select ***Nodes*** from the pull-down menu at the top and ***Attac hed*** to below that. Select ***Areas, All*** below that. Click ***Apply***.



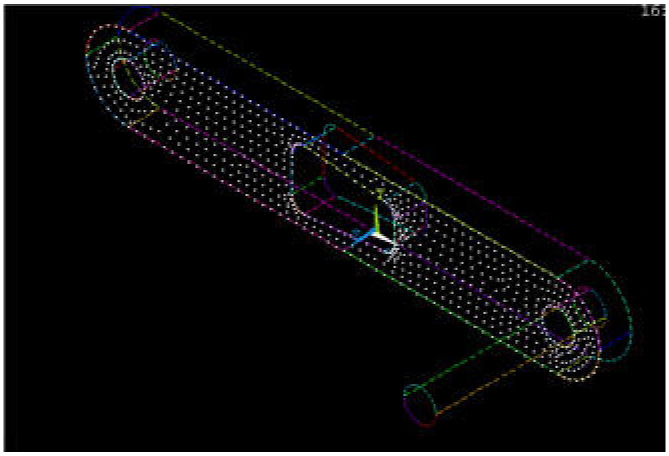
Check that only nodes attached to the front face are currently selected by clicking ***Replot*** in the *Select Entities* menu (this replots nodes).

We can get a better idea of where these nodes are located by plotting nodes as well as lines.

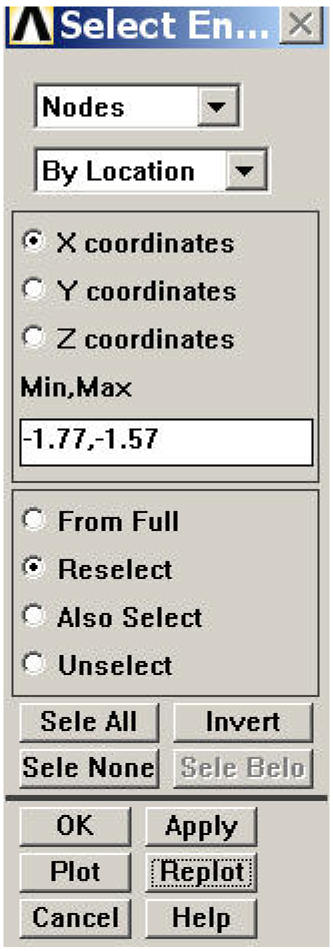
Utility Menu > PlotCtrls > Multi-Plot Controls ... **> OK**

Select ***Lines*** and ***Nodes*** and click ***OK***.

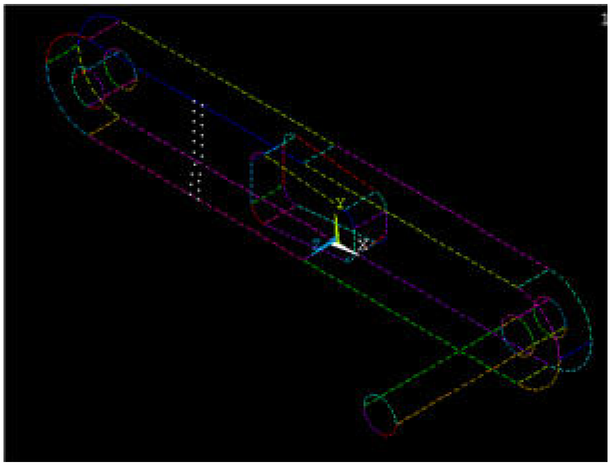
Utility Menu > Plot > Multi-Plots



From these currently selected nodes, we next select nodes that satisfy the following criterion: -1.77" x -1.57". In the *Select Entities* menu, retain ***Nodes*** at the top. Select ***By Location*** and ***X coordinates*** below that. Enter ***Min,Max*** values as per the snapshot below. Since we want the nodes to be selected from the current set rather than the full set, choose the ***Reselect*** radio button. Click ***Apply*** and then ***Replot***.

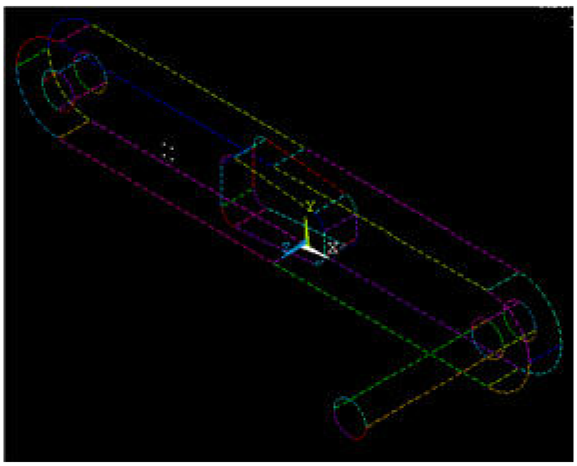


You should see that only the nodes that are in the desired *x*-coordinate range are selected.



Save your work:**Toolbar > SAVE_DB**

Next, from the current set of nodes, select nodes that satisfy the following criterion: 0.45" y 0.85" by appropriately modifying the previous select action step. The snapshot below shows what I get: four nodes remain in the selected set. If you mess up, resume from your *.db* file.



Now all actions on nodes will be performed only on the four nodes that are currently selected. For instance, to list the strain values at these nodes, choose

Main Menu > General Postproc > List results > Nodal Solution > Elastic Strain > X-Component of elastic strain

You can save these values to a text file using **File > Save as**. You can then read in the values from the text file into Excel or MATLAB for further processing such as finding the average.

Once you are done with the select operations, exit the *Select Entities* menu and choose

Utility Menu > Select > Everything

Utility Menu > Plot > Nodes

Save your work:**Toolbar > SAVE_DB**

Go to Step 9: Validate the results
See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Step 9: Validate the results

It is ***very important*** that you take the time to check the validity of your solution. This section leads you through some of the steps you can take to validate your solution.

Simple Checks

Does the deformed shape look reasonable and agree with the applied boundary conditions? We checked this in step 8.

Do the reactions at the supports balance the applied forces for static equilibrium? To check this, select

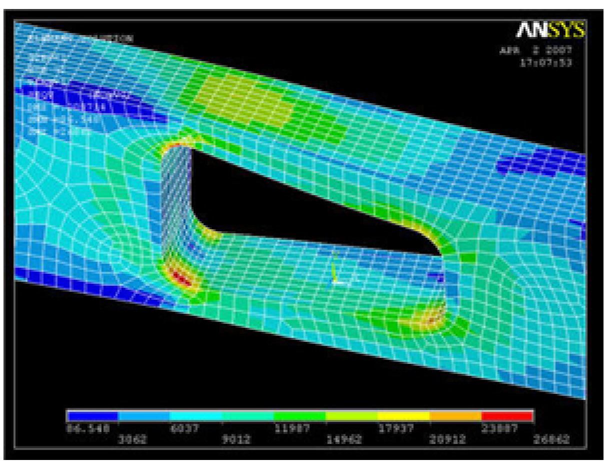
Main Menu > General Postproc > List Results > Reaction Solu

Select ***All struc forc F*** for Item to be listed and click ***OK***. The forces in the X and Z directions are essentially zero and the total Y-reaction is 100.00 (lbf) as expected.

Refine Mesh

Let's repeat the solution on a finer mesh with more divisions in the z-direction. Repeat the mesh steps for the MESH200 element, but this time use smart size 3 and element size of 0.08. Repeat the mesh steps for the SOLID45 element and set the element edge length to 0.05 instead of 0.125. This will create 10 divisions through the thickness of the crank instead of 4. When warned that the picked volumes are already meshed, check ***Yes*** and click ***OK*** to remesh.

Obtain a new solution and plot the elemental solution of the von Mises stress:



Higher Resolution Image

	Coarser Mesh	Finer Mesh
DMX	0.026148in	0.026651 in
SMX	25308psi	27942psi

The maximum displacement at the tip of shaft is 1.9% greater and the maximum stress is 10% greater. This indicates that the solution we have obtained is still dependent on the mesh. We would need to further refine the mesh. Do keep in mind that one would have to make more detailed comparisons between the solutions on the two meshes before we can make a definitive statement about the mesh independence of our results.

Exit ANSYS

Utility Menu > File > Exit

Select ***Save Everything*** and click ***OK***.

See and rate the complete Learning Module

Go to all ANSYS Learning Modules