

ANSYS - Crank 6 Specify boundary conditions Content

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

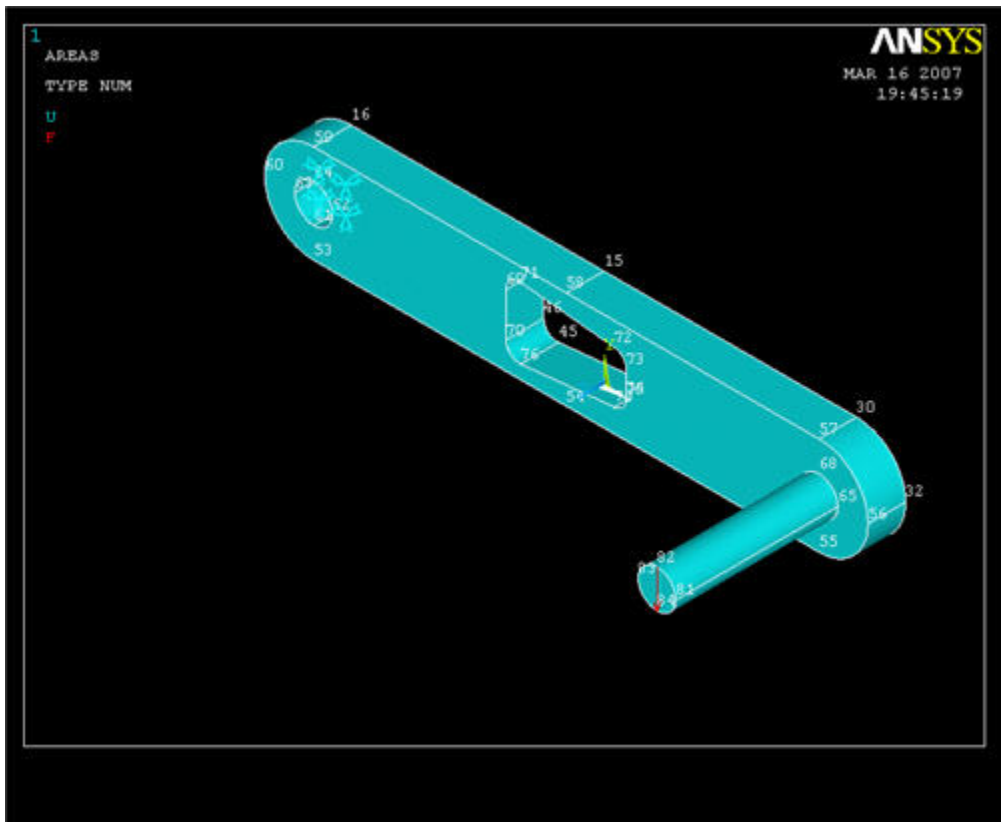
It will be helpful to see the areas we're constraining, so select Utility Menu > Plot > Areas. We can see that the hole consists of multiple areas (4, in fact). Hold down the left-click and you can see that there are 4 surfaces that make up the inside of the hole. Pick all 4 and click OK. Select All DOF and click OK. The displacement value can be left blank as it defaults to 0. You can now see the displacement is fixed in 3 directions at four places.

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:



Now let's see some results!

Save Your Work

Toolbar > SAVE_DB