

ANSYS 11 - Crank Step 6

Problem Specification

1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
8. Postprocess the results
9. Validate the results

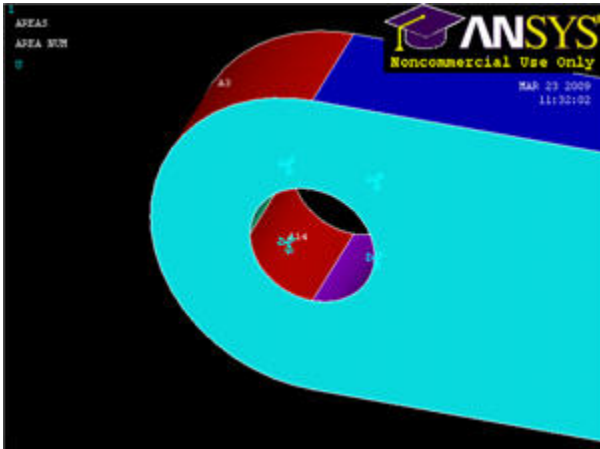
Step 6: Specify boundary conditions

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.



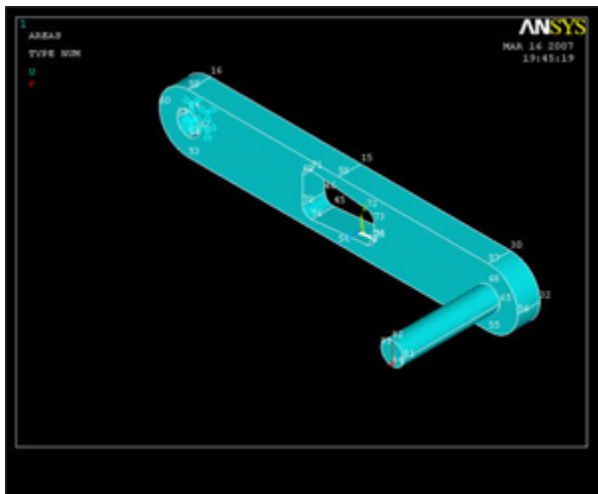
[Higher Resolution Image](#)

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](#)