

ANSYS 12 - Crank - Step 4

Author: Rajesh Bhaskaran & Yong Sheng Khoo, Cornell University

Problem Specification

1. Pre-Analysis & Start-Up
2. Geometry
3. Mesh
4. Setup (Physics)
5. Solution
6. Results
7. Verification & Validation

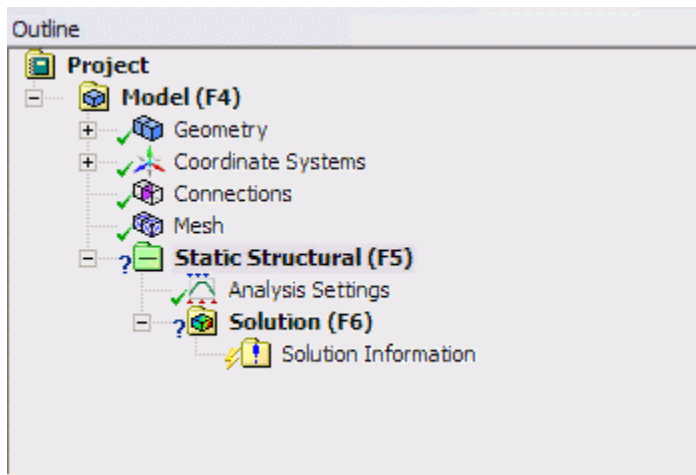
Step 4: Setup (Physics)

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 Support

Under **Outline**, right click on **Static Structural (F5)**, move the mouse cursor to **Insert** and select **Fixed Support**.

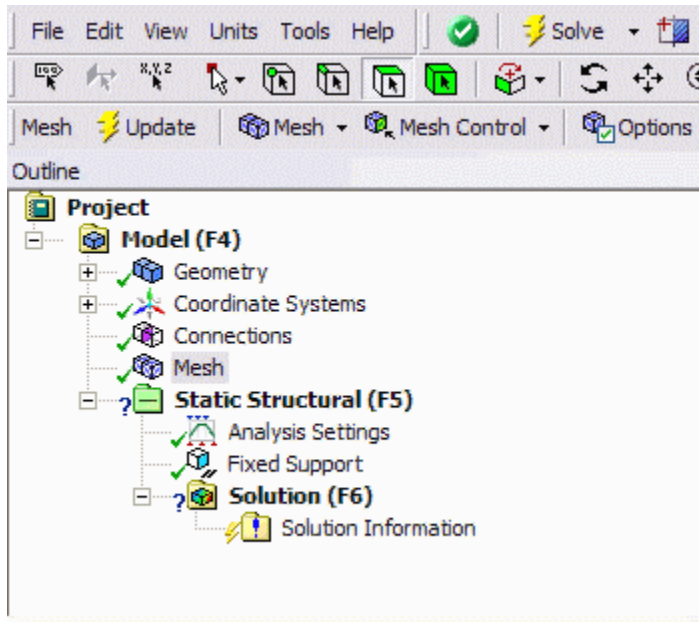
Now in the **Graphics** area, select the cylindrical hole and click **Apply** next to the **Geometry** under **Details of "Fixed Support"**. You will see the fixed support boundary condition shown in the Graphics window.



Force on Shaft

Now we will apply 100 lb force at the end of the shaft. This time, I will show you another way of inserting boundary condition.

Under **Outline**, left click on **Static Structural (F5)**. Third menu at the top will show different type of boundary conditions. Left click on **Loads** and select **Force**. In the **GraphicsDetails of "Force"** window, select the circular area at the end of the shaft. Under , click **Apply** next to **Geometry**. The selected area in the Graphics window will be applied with this boundary condition. Next we will enter the value of 100 lb force. Next to **Define By**, change **Vector** to **Components**. Enter a value of -100 next to **Y Component**. In the Graphics window, you will see the direction of force applied.



Congratulations! We are done setting up the boundary conditions.

[Go to Step 5: Solution](#)

[See and rate the complete Learning Module](#)

[Go to all ANSYS Learning Modules](#)