

ANSYS 12 - Crank - Step 3

Author: Rajesh Bhaskaran & Yong Sheng Khoo, Cornell University

Problem Specification

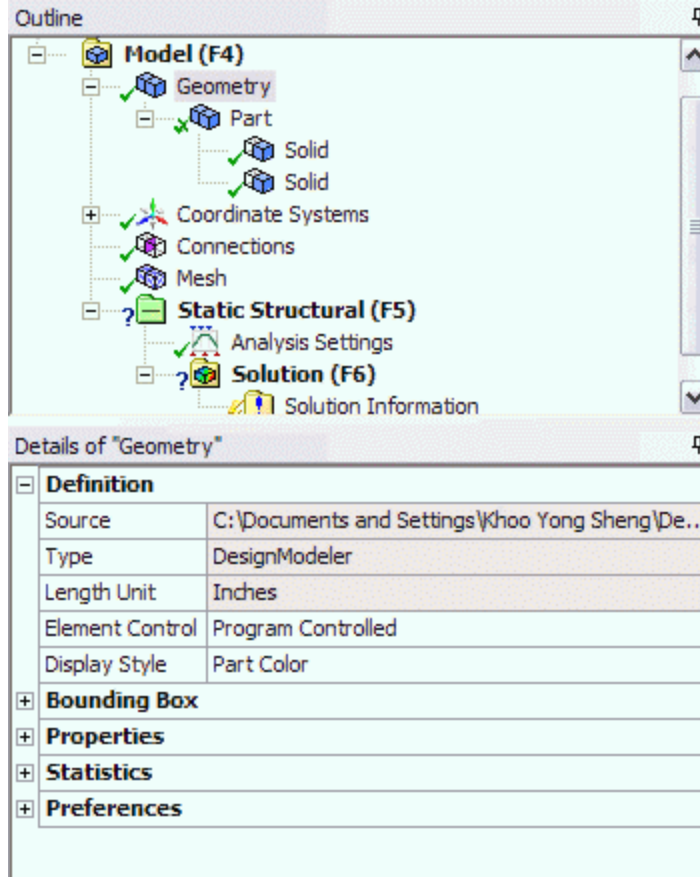
1. Pre-Analysis & Start-Up
2. Geometry
3. Mesh
4. Setup (Physics)
5. Solution
6. Results
7. Verification & Validation

Step 3: Mesh

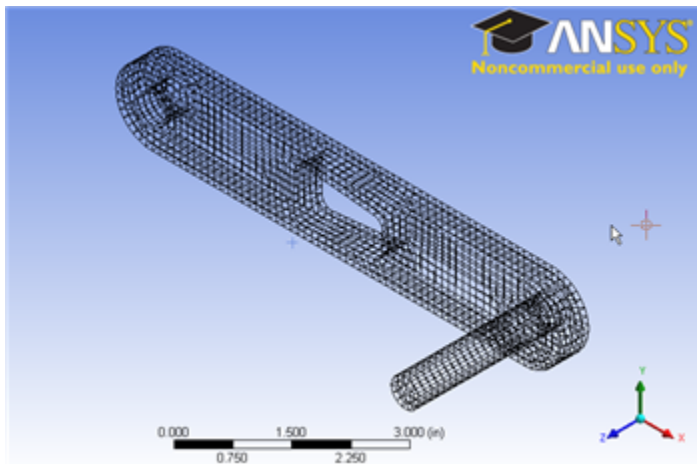
In the **Crank** cell, right click on **Model**, and click **Edit...**

A new window Mechanical Modeler will open. We will start with the default mesh generation method. Under **Outline**, right click on **Mesh** and click **Generate Mesh**. This will give us mesh that ANSYS mesher thinks appropriate for our geometry. We see that ANSYS is smart enough to create a mesh that sweep across the cross sectional area of the crank. This is the type of mesh we are seeking for our geometry. However, sizing wise, the mesh is too coarse. We will have to increase the number of elements.

Under **Outline**, click on **Mesh**. Next under **Details of "Mesh"**, expand the **Sizing** tab and we will specify the element size that we desire. Element size of 0.125 in seems to be a good start. Next to **Element Size**, enter 0.125. Finally, click on **Generate Mesh** again.



This is the mesh that appear in the Graphics window on the right.



[Higher Resolution Image](#)

You should have 18205 nodes and 3582 elements for your mesh. Click on **Statistics** under **Details of "Mesh"** to see that.

For future reference, the default mesh might not produce the desirable mesh that we want. To specify our own mesh control, right click on **Mesh** and select **Insert**. This way, we can insert meshing type and control that we desire. To learn more about different meshing methods, please refer to the **Mechanical Help** at the top menu.

[Go to Step 4: Setup \(Physics\)](#)

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

[Go to all ANSYS Learning Modules](#)