

ANSYS 12 - Crank - Step 7

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 7: Verification & Validation

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 6.

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

Outline > Solution (F6) > Insert > Probe > Force Reaction

Under **Details of "Force Reaction"**, select **Fixed Support** as **Boundary Condition**. At the top menu, click Solve to Evaluate All Results.

The forces in the X and Z directions are essentially zero and the total Y Axis is 100.00 (lbf) as expected.

Refine Mesh

Let's repeat the solution on a finer mesh with more smaller element size. Repeat the mesh steps, but this time use element size of 0.05 instead of 0.125. This will create 10 divisions through the thickness of the crank instead of 4.

Obtain a new solution and compare both solution.

	Coarser Mesh	Finer Mesh
Max Deformation	0.0257in	0.0258in
Max Stress	27897psi	27978psi

The difference in maximum displacement at the tip of the crank handle is less than 0.4%.

The difference in maximum stress at the cut out area is less than 0.3%.

This means that our mesh is refine enough and the solution obtained using the original mesh is probably good enough.

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.

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

[Go to all ANSYS Learning Modules](#)