ANSYS - Bike Crank

Author: Rajesh Bhaskaran, Cornell University

Problem Specification
1. Pre-Analysis & Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Numerical Solution
6. Numerical Results
7. Verification & Validation
Exercises
Comments

3D Finite-Element Analysis of a Bike Crank

This module is from our free online simulations course at edX.org (sign up here). The edX interface provides a better user experience and the content has been updated, so we recommend that you go through the module there rather than here. Also, you will be able to see answers to the questions embedded in the module there.

Created using ANSYS 14.5.

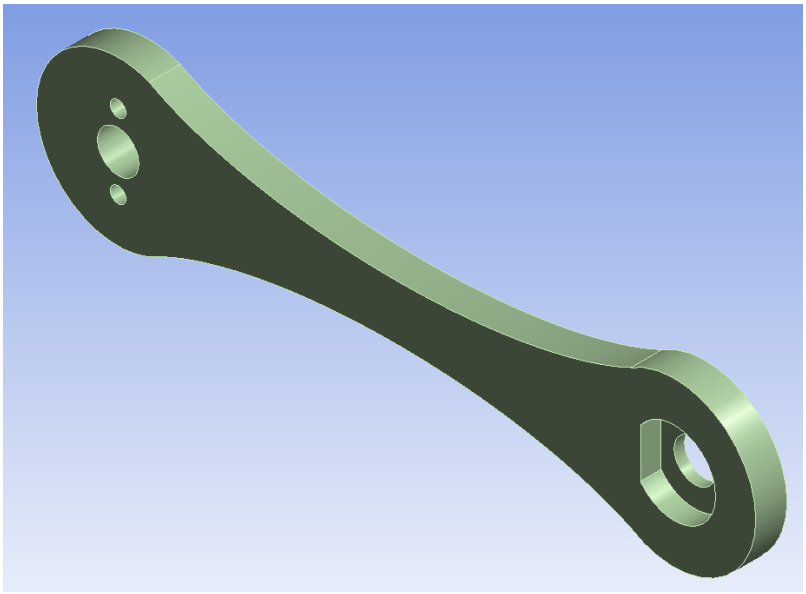
Learning Goals

In this tutorial, you will learn to:

- Determine the displacements and stresses in a bike crank using 3D FEA capabilities in ANSYS Mechanical
- Verify the finite-element results from ANSYS by refining the mesh and also comparing with hand calculations

Problem Specification

Consider the following bike crank model.

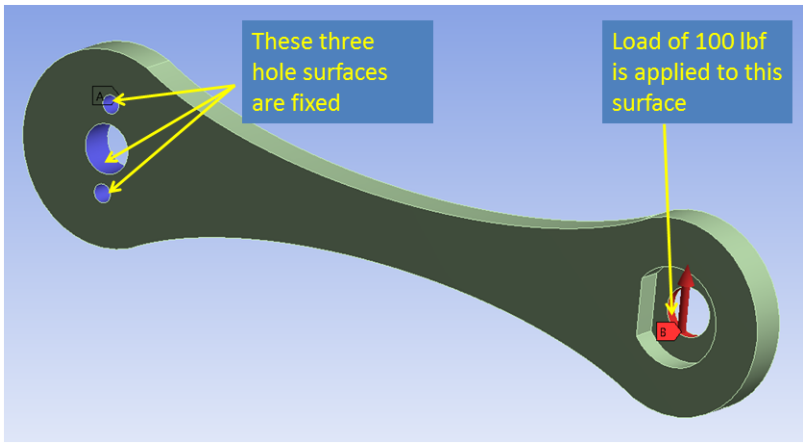


To orient ourselves, the following figure shows the location of a similar bike crank mounted on a bicycle.



Material properties: The bicycle crank's material is aluminum 6061-t6. The Young's modulus is 10,000 ksi, and the Poisson's Ratio is .33.

Boundary conditions: Apply a load of 100 lbf in the y-direction on the right hole surface and fix the 3 left hole surfaces as shown below. Note that this is an approximation of the actual loads and constraints on the bike crank.



Using ANSYS Mechanical, determine the following:

- Deformed shape and displacement field
- Stress distribution

Go to Step 1: Pre-Analysis & Start-Up

Go to all ANSYS Learning Modules