

# 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

**i** 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.

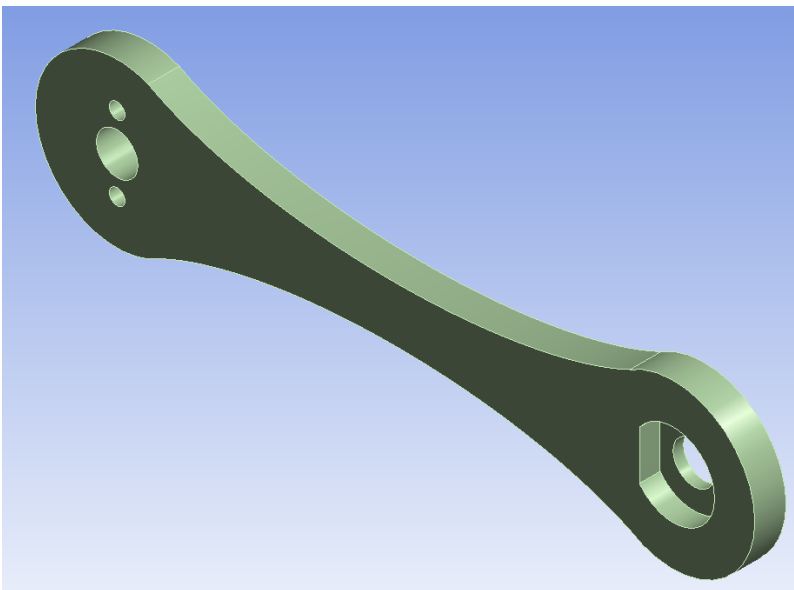
### Learning Goals

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.

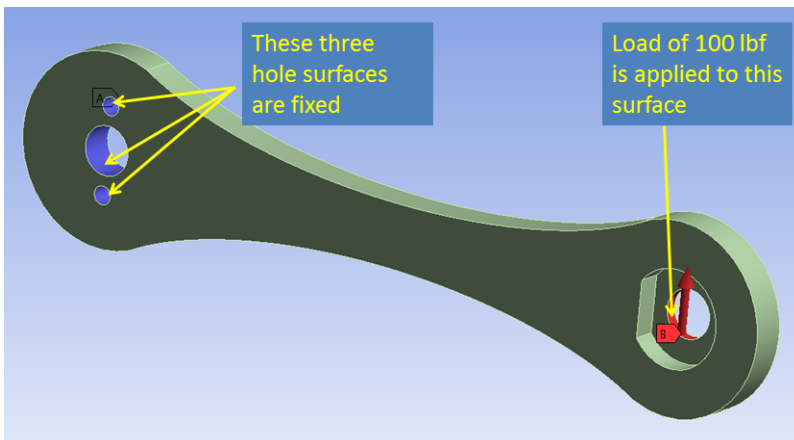


To orient ourselves, the following figure shows the location of a similar bike crank mounted on a bicycle.



**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](#)