ANSYS AIM - Bike Crank Tutorial

Author(s): Andrew Dawd, 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
Comments

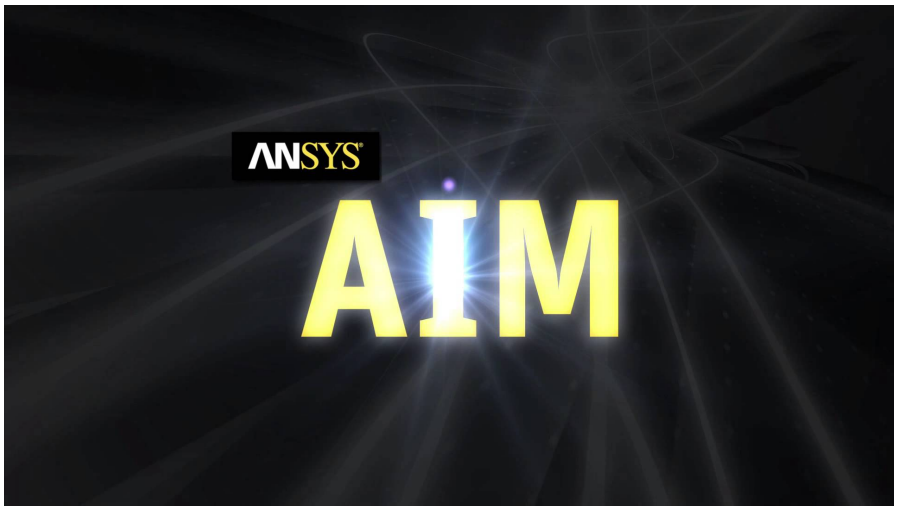
3D Finite Element Analysis of a Bike Crank Using ANSYS AIM

Created using ANSYS 16.2. There may be some differences in newer version (AIM 17.0)

Learning Goals

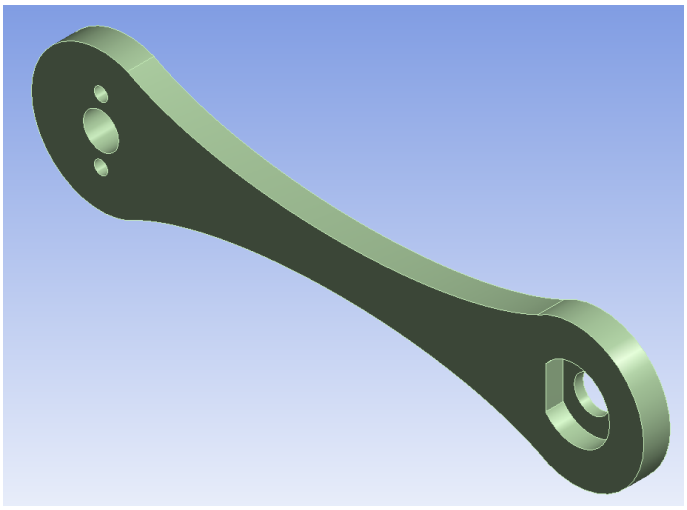
In this tutorial, you will learn to:

- Determine the displacements and stresses in a bike crank using 3D FEA capabilities in ANSYS AIM
- Verify the finite-element results from ANSYS by refining the mesh and also comparing with hand calculations
- Gain a better understanding of the new ANSYS AIM software and its comparison to Workbench.



Problem Specification

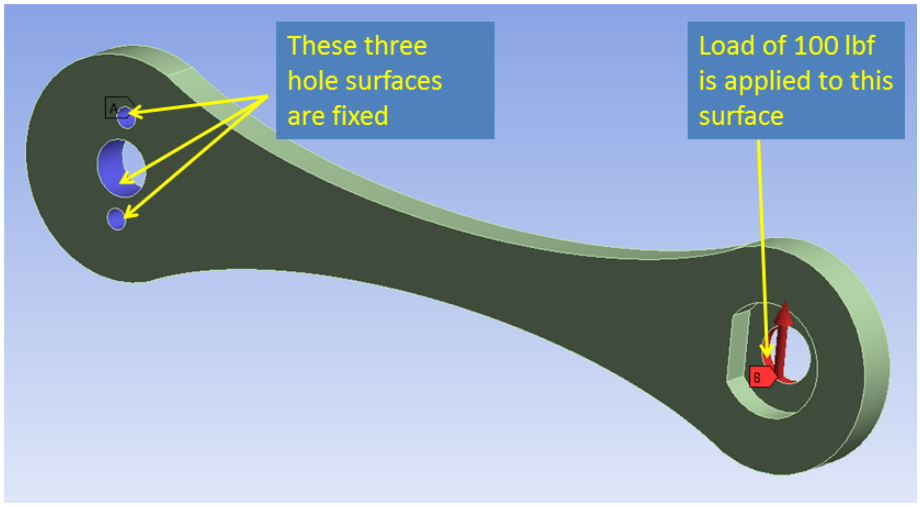
Consider the following bike crank model:



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 0.33 .

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 AIM, determine the following:

- Deformed shape and displacement field

- Stress distribution

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

Go to all (ANSYS or FLUENT) Learning Modules