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

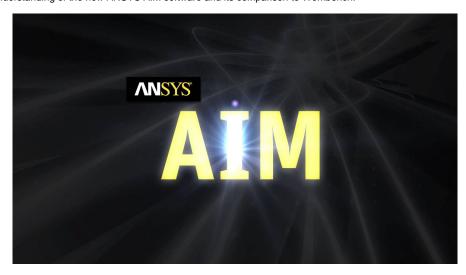
3D Finite Element Analysis of a Bike Crank Using ANSYS AIM

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

Learning Goals

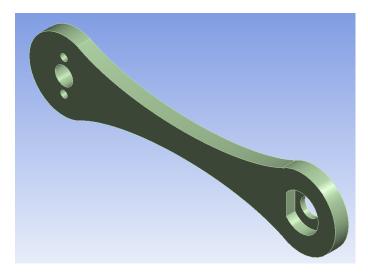
In this tutorial, you will learn to:

- 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:

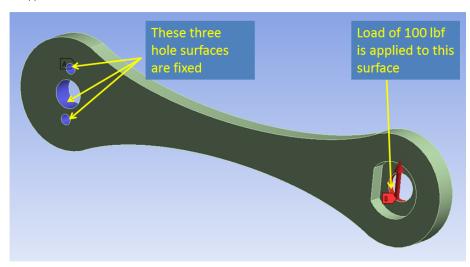


To orient ourselves, the following figure shows the location of a similar bike crank mounted on a bicycle.



 $\textbf{Material properties} : The \ bicycle \ crank's \ material \ is \ aluminum \ 6061-t6. \ The \ Young's \ modulus \ is \ \underline{10,000 \ ksi}, \ and \ the \ Poisson's \ Ratio \ is \ \underline{0.33} \ .$

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