Bike Crank - Mesh

Author: Rajesh Bhaskaran, Cornell University

Mesh

The video below demonstrates the steps to mesh the geometry using Hexahedral elements which look like boxes. Hexahedral (or hex) elements yield higher accuracy compared with the default tetrahedral (or tet) elements for the same number of nodes. The "multizone" meshing method is used to get hex elements for this geometry.

Summary of steps in the above video:

1. Double click on Model in the project window.
2. Suppress pedal and Part9 under Geometry.
3. Highlight Mesh, Mesh Control > Sizing.
4. Choose the Select Body under Help button and highlight the Bike Crank > Apply.
5. Element Size: 0.2 in.
6. Update to view the more refined mesh.
7. Mesh Control > Method > click on body > Apply.
8. Method > Multizone.
9. Update and check presence under Mesh (you should see Body Sizing and MultiZone.

If you are running Ansys 15 or above, **don't worry if your mesh looks different**. If everything else was done correctly, you may expect a mesh not that regular, especially in the middle of the crank. For Ansys 17.2, expect to get around 1139 elements and 6400 nodes. Different versions of the mesher give different numbers of elements and nodes with similar settings. This is just the initial mesh. As part of verification, one needs to do a mesh convergence study.

We next refine the mesh locally in "areas of concern" using a "face sizing" as demonstrated in the following video.

Summary of steps in the above video:

1. Zoom into left hole using the right mouse button.
2. Click on Mesh Control > Sizing > Select Face > click on the inside of the three small holes.
3. Element Size : 0.1 in (half the size of the overall mesh).
4. Choose the Select Body under Help button and highlight the Bike Crank > Apply.
5. Update (will refine the mesh around the holes).

In the following video, we interrogate the mesh in the interior of the model by cutting through the mesh using "section planes".

One important metric of element quality is skewness. The video below shows how to check the skewness distribution and display elements that are highly skewed.

Go to Step 4: Physics Setup

Go to all ANSYS Learning Modules