

Bike Crank (Part 2) - Mesh

Author: Sebastien Lachance-Barrett, 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](#)

Mesh

In this section, we will mesh each gauge with one shell element. We will also hard code commands to the gauges so that ANSYS ignores their stiffness i.e. the original stiffness matrix is not changed by the addition of the gauges.

ANSYS looks for a thickness input for each "surface body" corresponding to the gauges. We'll give the gauges a thickness. Keep in mind that this thickness does *not* affect the strain values for the gauges since the gauges are just riding along on the crank in our model. So their strain will be the same as the underlying strain on the crank face.

Summary of steps in the above video:

1. Open up Model in the Project Schematic
2. Expand Geometry and click on one of the Surface Bodies
3. The yellow highlights Thickness which must be filled in
4. Enter an arbitrarily small number: 0.01 in as thickness does not matter in our analysis

We now show you how to insert commands to the surface bodies so that the stiffness of the gauges are ignored.

Summary of steps in the above video:

1. Right click the first Surface Body > Insert > Commands
2. Type et,matid,181 (this surface body should be matched with type 181 of the element catalog)
3. Then type keyopt,matid,1,2 (specific command for element 181, have a stiffness of 0)
4. Copy paste the code to the two other surface bodies

For the surface bodies to be fixed to the crank, much like a strain gauge would be perfectly glued on the crank, we need to specify the contacts between these two bodies as being bonded. Here, we show you how to perform this step using automatic connections.

Summary of steps in the above video:

1. Right click Connections > Insert > Connection Group
2. Right click Connection Group > Create Automatic Connections

(Optional) If automatic connections do not work, then one must define the contacts manually. The following video will show you how to do this.

Summary of the above video:

- Right click Connections > Insert > Connection Group
- Right click Connection Group > Insert > Manual Contact Region
- Click the Contact Selection Field > Select Face > click on front face of bike crank > Apply
- Click the Target Selection Field > zoom into the strain gauge > click on strain gauge > Apply

Finally, we specify the surface elements to have a mesh size of one using edge sizing.

Summary of steps in the above video:

1. Click Mesh in the tree > Update
2. Zoom in on surface bodies (strain gauges)
3. Right click on Mesh > Sizing
4. Select Edges on tool bars > hold down Ctrl and select all strain gauge edges
5. In the Details Window, under Type > Number of Divisions

6. Behavior > Hard
7. Update Mesh

[Go to Step 4: Physics Setup](#)

[Go to all ANSYS Learning Modules](#)