ANSYS - Bike Crank Part 2 (old)

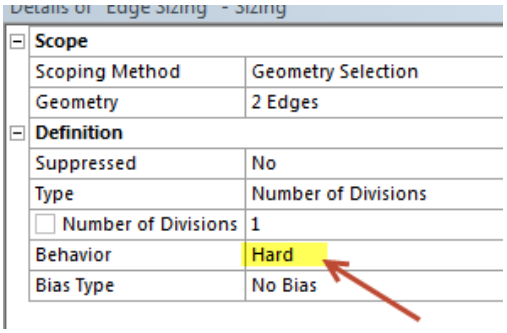
Unable to render {include} The included page could not be found.

This page covers follow-on exercises to the bike crank tutorial. In particular, it shows how to add a strain gage to the ANSYS model to predict the strain that would be measured by a gage mounted on the crank model.

Strain Gage Modeling

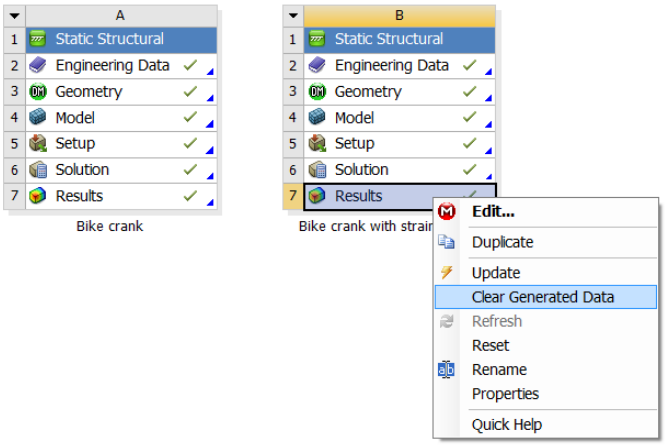
The following video shows how to add a surface to the crank model to find the average strain over the area covered by a strain gauge mounted on the crank. The video includes audio as do other videos on this page. Note that you have to make the following modifications to the procedure shown in the video:

- Using "Automatic Connections" to connect the crank arm and the gage doesn't always work. An alternate procedure to manually connect the two entities is shown in a later video below.
- When specifying the edge sizing for the gage, set Behavior=Hard as in the snapshot below. Otherwise, ANSYS might over-ride your edge setting and you will get more than one element.



An overview of the procedure followed in the video and explanation of the associated steps can be found in this powerpoint presentation from MAE 3272, Spring 2013.

Our ANSYS Mechanical 14.0 model with a strain gage mounted can be downloaded here for your review. Download the *.wbpz* file and double-click on it to load it in ANSYS Workbench. To save the file in the usual *.wbpj* format, select *File > Save as*. To reduce the file size, the results have not been saved. You'll have to clear all results by selecting *(right-click)Results > Clear Generated Data* in the project page. Then, re-solve in ANSYS Mechanical (the red M in the taskbar) to see the results.



Manually Connecting Crank Arm and Gage

The above video shows how to add a connection between the crank arm and the gage using ***Create Automatic Connections***. But this automated procedure doesn't always work. In that case, one has to connect the two entities manually using ***Insert > Manual Contact Region*** as shown in the video below.

Meshing with Hexahedrals using the Multi-Zone Method

The multi-zone method in ANSYS

- Gives hexahedral or box elements for certain geometries
- Chops up the domain into regions which can be "hex-meshed". These regions often have a shape that can be created though extrusion
- The default meshing method often gives a tetrahedral mesh which has lower accuracy than a hex mesh for the same number of nodes. The advantage of tetrahedrals is that you can mesh any general geometry while pure hex meshes can be created only for some geometries.

The steps for using the multi-zone meshing method for the bike crank geometry are shown in the video below.

Creating a New Coordinate System in ANSYS Mechanical

If your default coordinate system is not centered on one of the holes in the crank, or you wish to learn how to define new coordinate systems inside the mechanical window, please refer to the following video:

Comments

Unable to render {include} The included page could not be found.

Unknown macro: 'menulink'

First time using our Piazza discussion board? Please click on the following link for instructions on how to enroll in our class:

Piazza Sign-Up Tutorial

Don't worry, it is quick and easy! After enrollment, you will be able to access the discussion pages for all SimCafe tutorials. Rest assured that you can post anonymously if you wish.

Go to all ANSYS Learning Modules