ANSYS 11 - Crank - CAD Import Instructions

You can import geometry from a CAD package such as Pro/Engineer or SolidWorks into ANSYS by following the instructions below.

Import Geometry

In your CAD package, save your geometry in *Parasolid* format. A *Parasolid* file commonly has the extension *.x_t* in its name.

In ANSYS, select

File > Import > Para

Browse to your parasolid file and click *OK*

Change display mode from wireframe to solid facets: **Utility Menu > Plotctrls > Style > Solid Model Facets**

Select *Normal Faceting* and click *OK.*

Re-plot volumes in order to see normal faceting: **Utility Menu > Plot > Volumes**
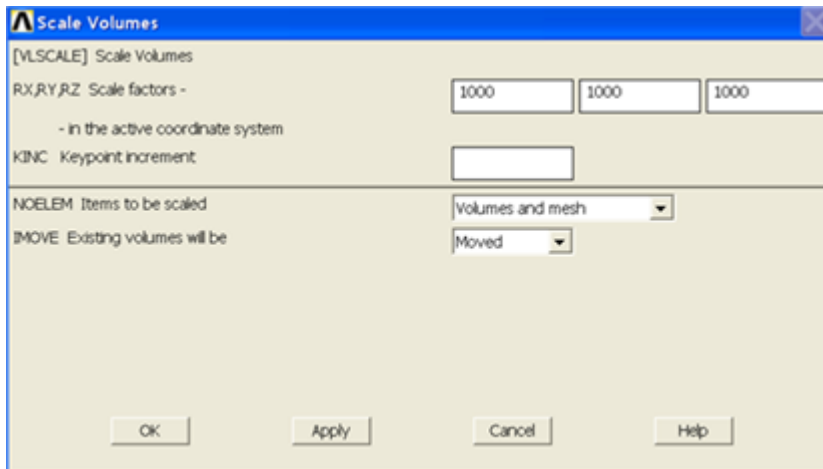
Rescale Geometry

When importing, ANSYS always scales the geometry to meters. To scale back to, say, *mm* from *meters*, use:

Main Menu> Preprocessor> Modeling> Operate> Scale> Volumes

Select *Pick All*.

For *RX, RY, RZ Scale factors*: enter *1000* for each

For *IMOVE Existing volumes will be:* select *Moved* and click *OK*

Check Geometry

Turn on keypoint numbers: **Utility Menu > PlotCtrls > Numbering**

Turn on *Keypoint numbers* and click *OK*.

List keypoint coordinates: **Utility Menu > List > Keypoint > Coordinates Only**

Spot-check X,Y,Z locations of selected keypoints against CAD drawing. Once you are satisfied that the keypoint locations are correct, you can turn off keypoint numbers: **Utility Menu > PlotCtrls > Numbering**

Turn off *Keypoint numbers* and click *OK*.

Glue Volumes

You need to glue volumes so that common surfaces (i.e. areas) are shared between the crank and the pedal shaft. Otherwise, ANSYS considers these to be two unconnected, independent parts. (I often forget this "glue" step and it comes back to bite me in sensitive spots of the anatomy).

Main Menu > Preprocessor > Modeling > Operate > Booleans > Glue > Volumes

Click *Pick All* to glue our two volumes together. Note that there are no visual indicators of whether or not the volumes have been glued. You should check the Command Window and look for the "GLUE VOLUMES" command.

See and rate the complete Learning Module

Go to all ANSYS Learning Modules