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

∧ Scale Volumes	×
[VLSCALE] Scale Volumes	
RX,RY,RZ Scale factors -	1000 1000
- in the active coordinate system	
KINC Keypoint increment	
NOELEM Items to be scaled	Volumes and mesh ▼
IMOVE Existing volumes will be	Moved 💌
	Facural Found
OK Apply	Cancel Help

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.

Go to Step 5: Mesh geometry

See and rate the complete Learning Module

Go to all ANSYS Learning Modules