MEE 631 CAD of Mechanical Systems

Tutorial: Importing CAD geometry in ANSYS APDL and Workbench

Tools Used: ANSYS 17.1 and SOLIDWORKS

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This tutorial cover importing of geometry through IGES, Parasolid files in ANSYS APDL.

Importing geometry file through direct integration of Solidworks and ANSYS Workbench and loading geometry through IGES, ACIS and Parasolid file through ANSYS Workbench

In order to import a CAD model from any modelling software into ANSYS it is necessary that proper supporting files are created so that appropriate surface and volumes are replicated in ANSYS GUI.

ANSYS APDL can read CAD model data through digital exchange of information supported by the various formats. Two of the mostly used and efficient formats will be discussed in this turorial:

- Parasolid
- Initial Graphics Exchange Specification (IGES)

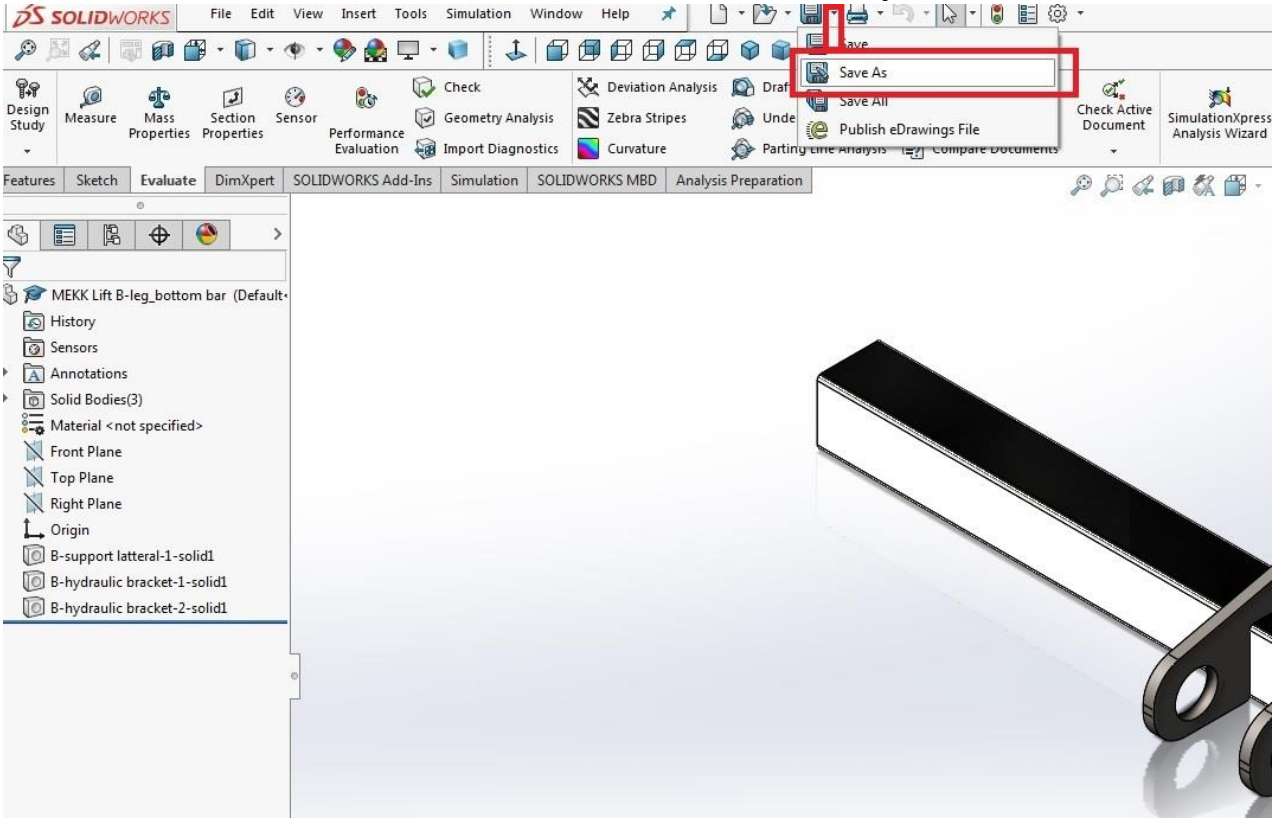
Parasolid files are of two types,

S. No	File Type	Extension
1	Neutral Binary Parasolid	.x_b
2	Text Parasolid	.x_t

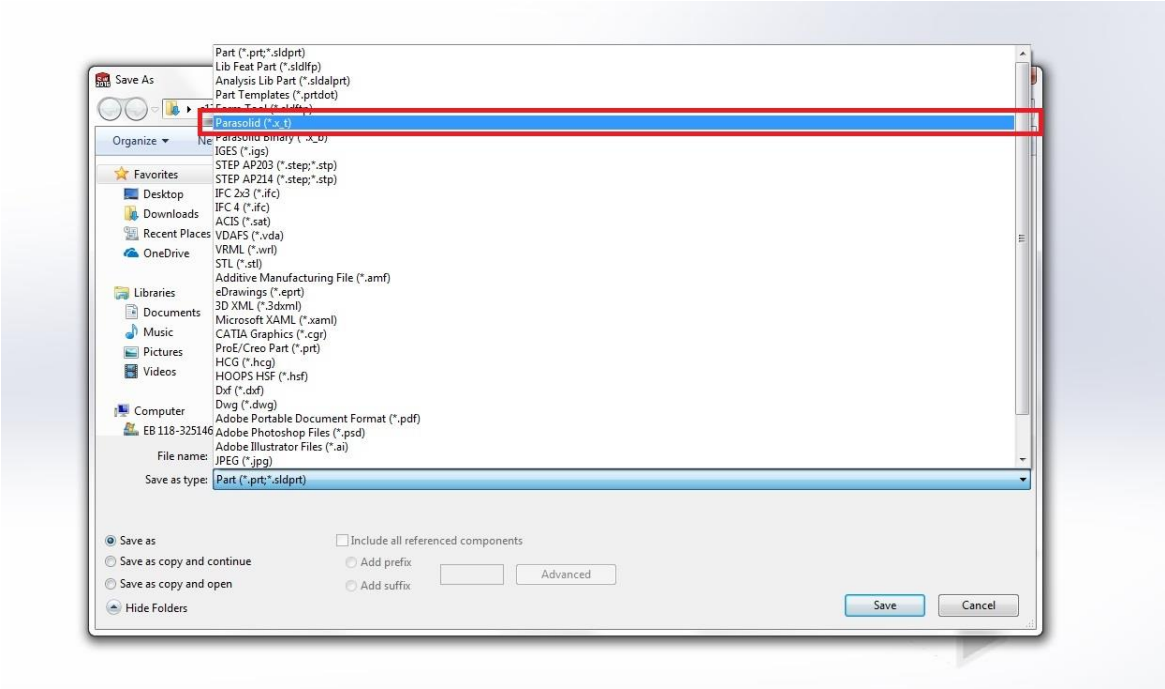
Importing through Parasolid file

In order to import a geometry through Solidworks follow the following steps:

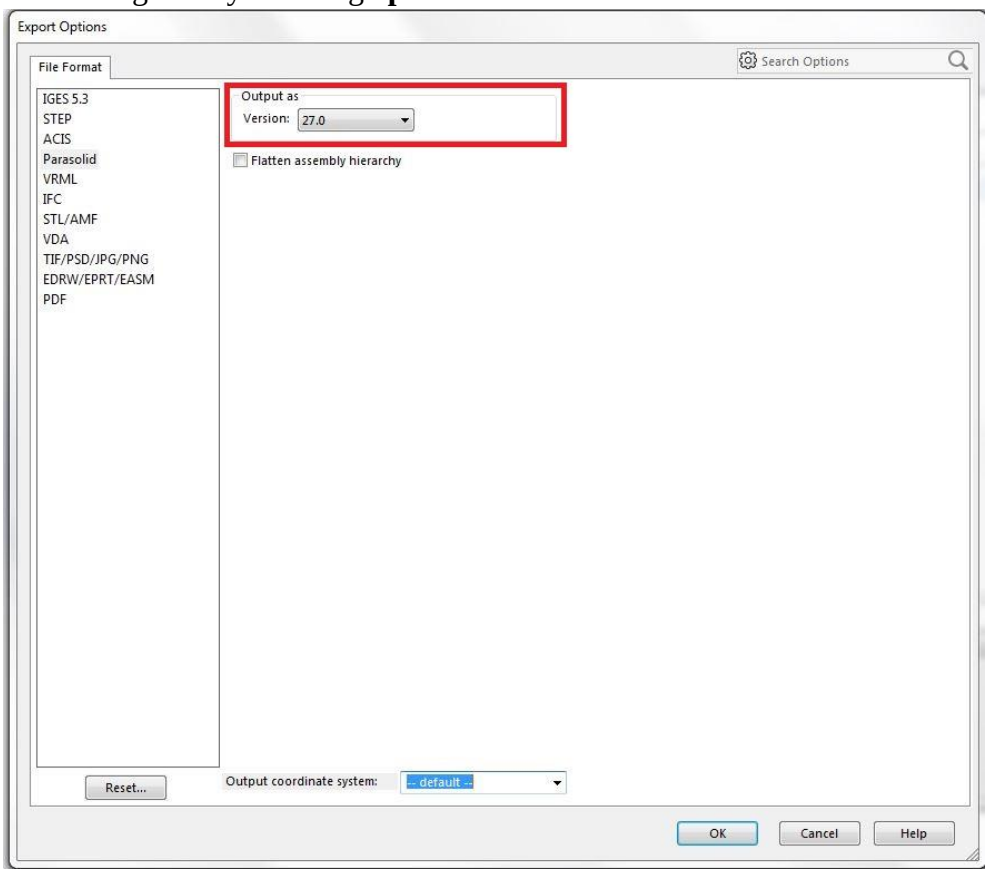
1) When the CAD model is finished select **"SAVE AS"** from the **Utility menu**



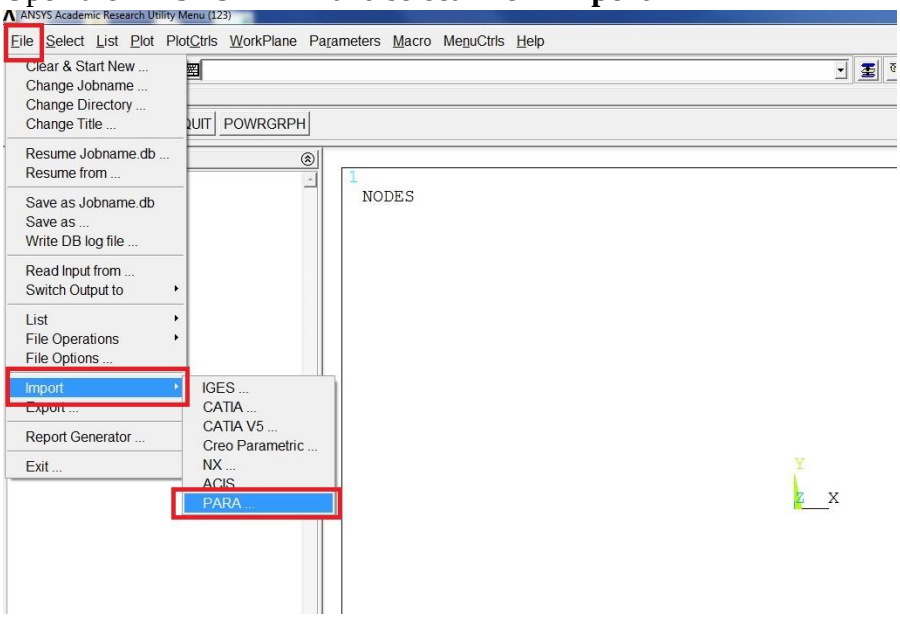
2) In the save as window select the file type as **“Parasolid (*.x_t)”**



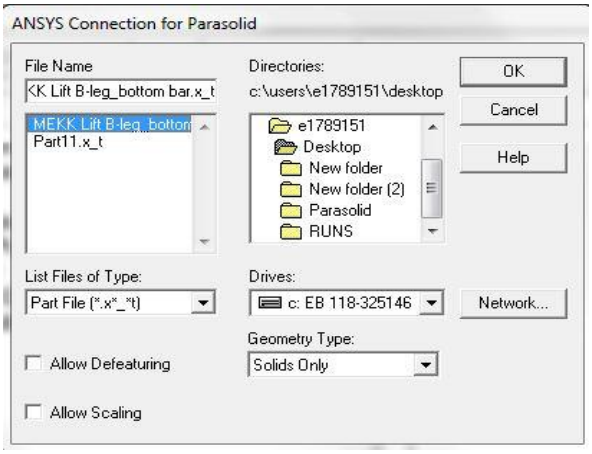
Before clicking the save key make a check that the output file is being saved with **Version 27** or higher. By selecting **options** from the save as window.



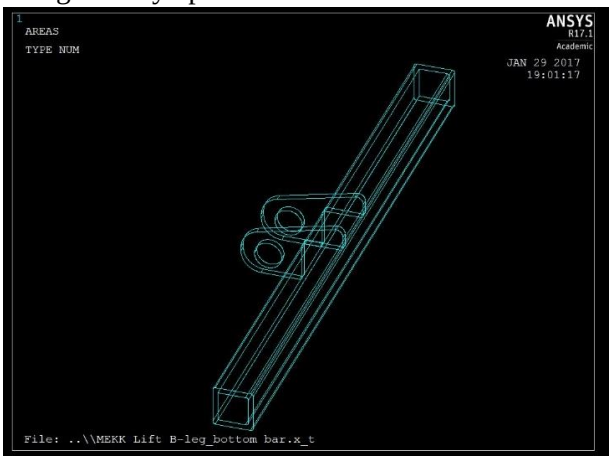
3) Open the ANSYS APDL and select **File > Import > PARA**



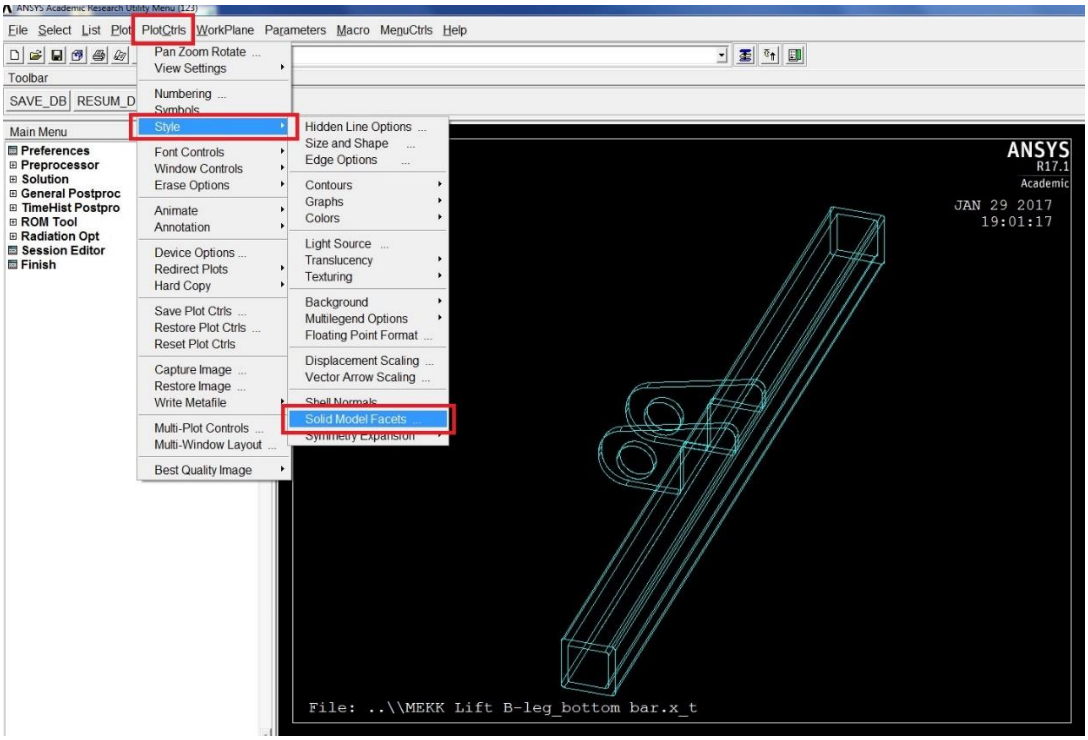
4) Select the file and click **OK**



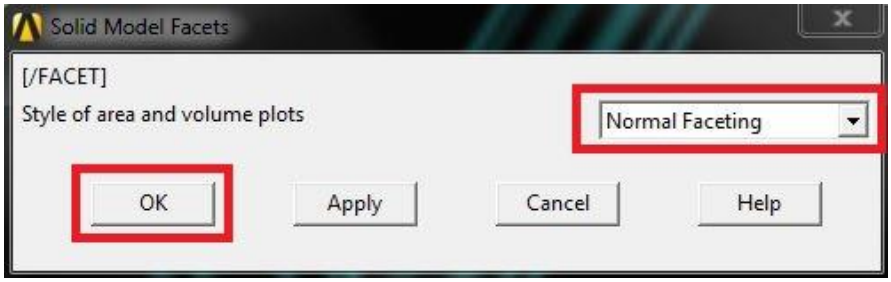
5) The geometry opens as a wireframe model



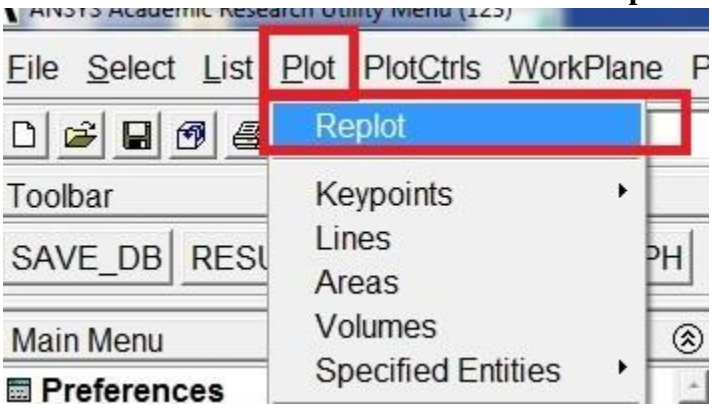
To obtain the solid model follow the steps:
PlotCtrls > Style > Solid Model Facets

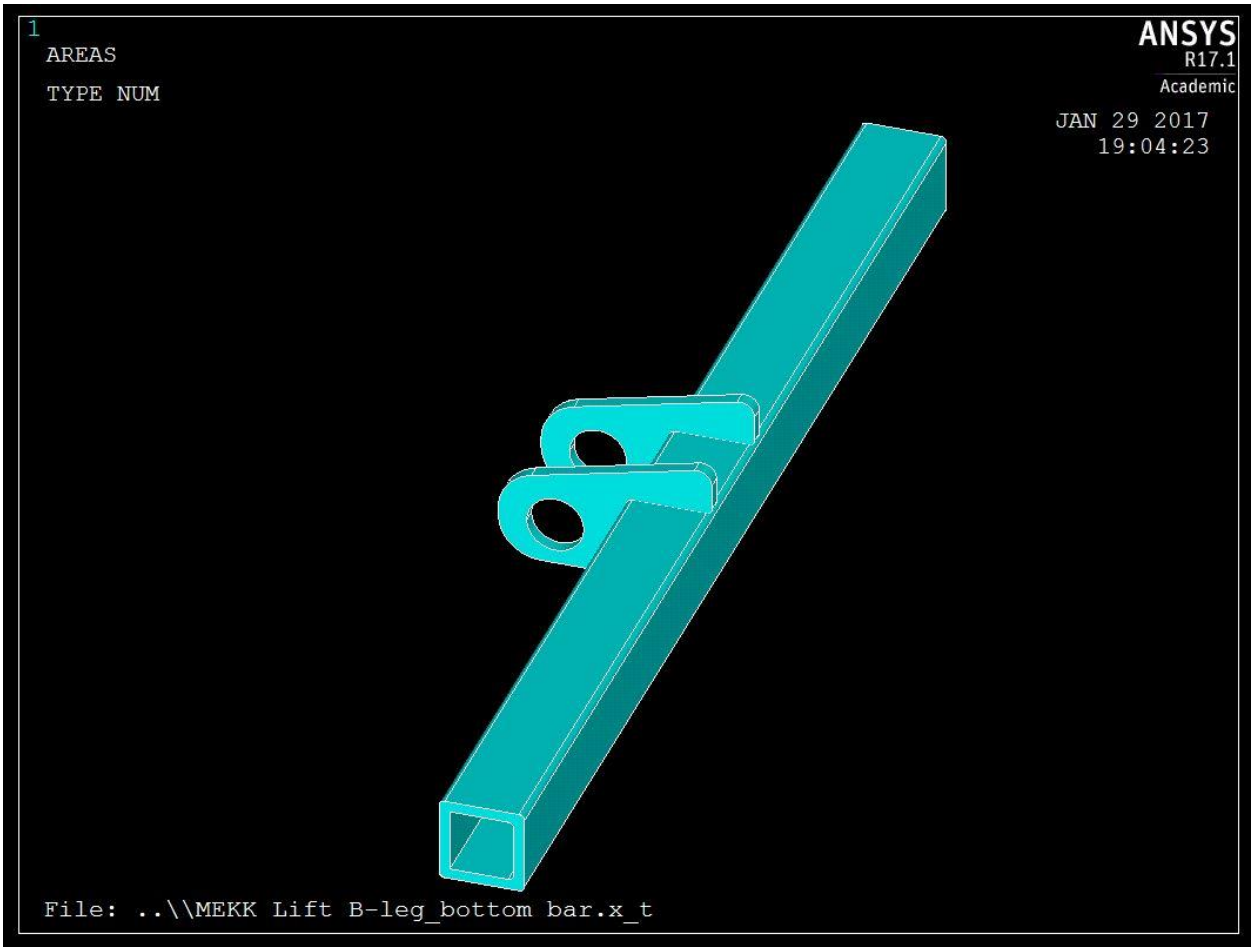


Select **"Normal Faceting"** in the dialog box and click **OK.**



6) In order to view the solid model select **Plot > Replot**




1
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TYPE NUM
ANSYS
R17.1
Academic
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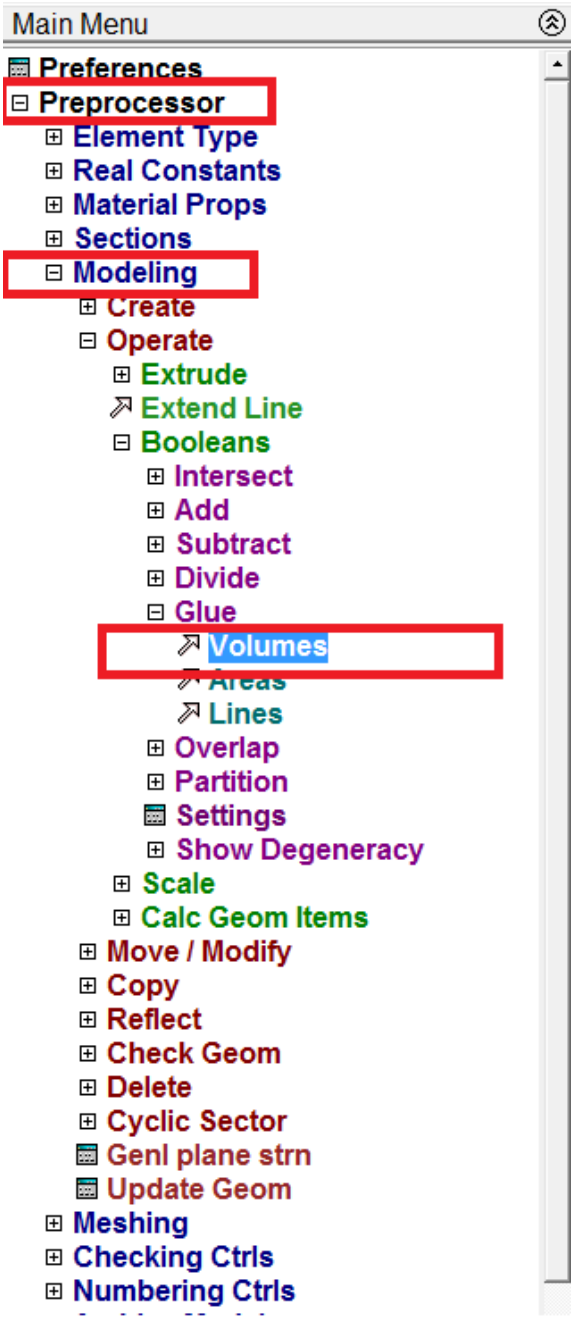
19:04:23
File: ..\\MEKK Lift B-leg_bottom bar.x_t

NOTE: Before proceeding to mesh the geometry make sure you **GLUE** all the volumes to avoid error.

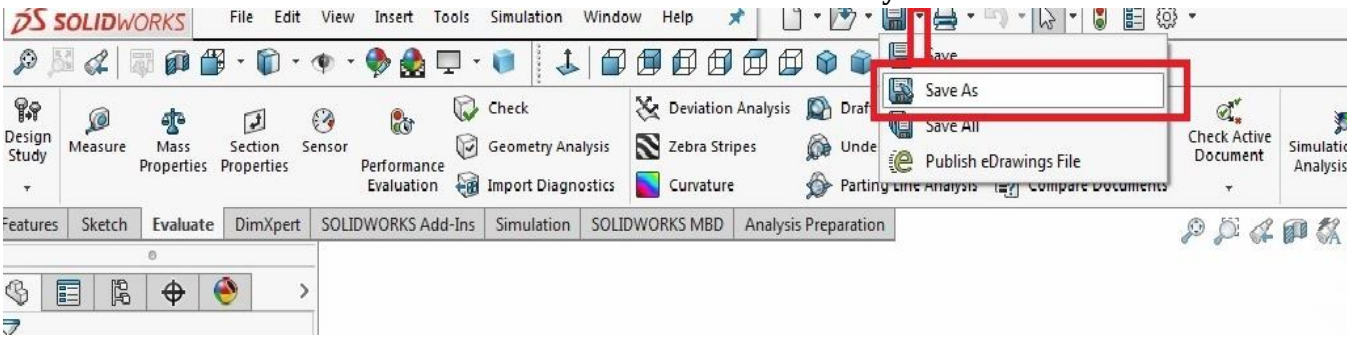
Preprocessor > Modelling > Operate > Booleans > Glue > Volumes > Pick all



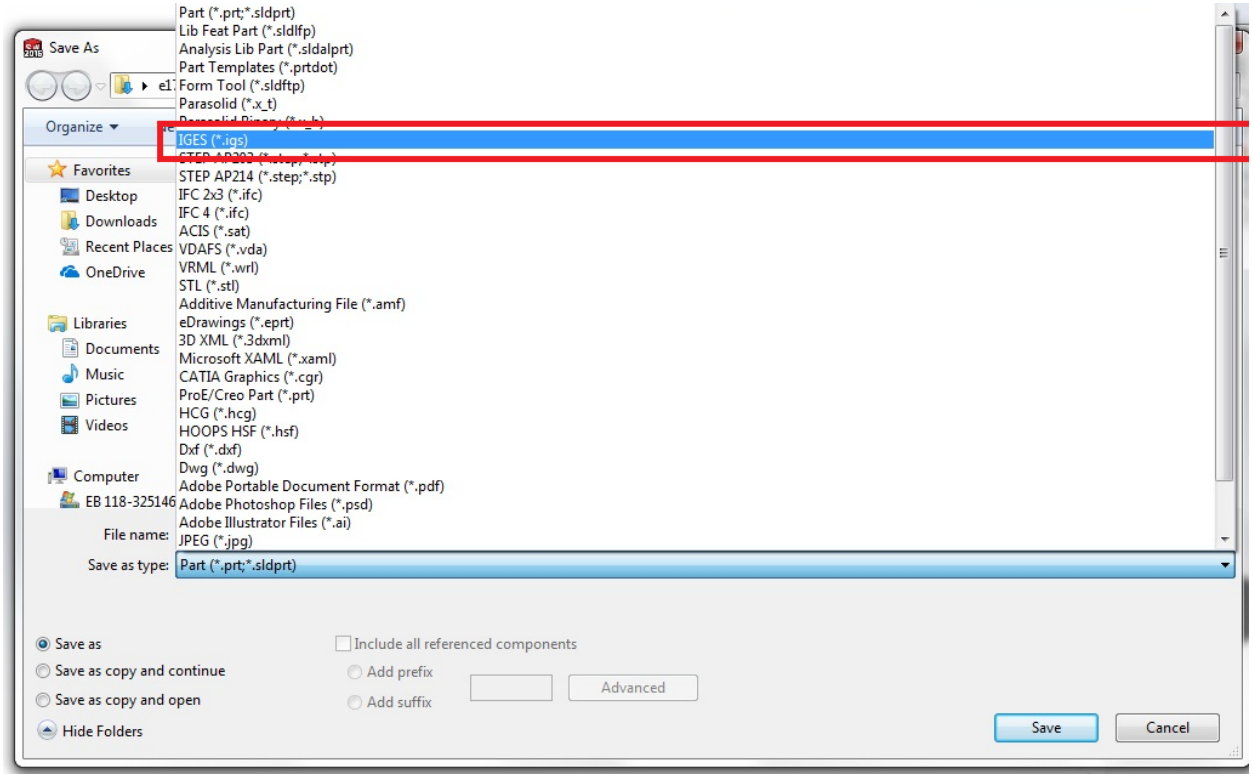
Initial Graphics Exchange Specification (IGES)

Opening geometry in ANSYS through IGES file. IGES files are followed by extension **"*.igs"**

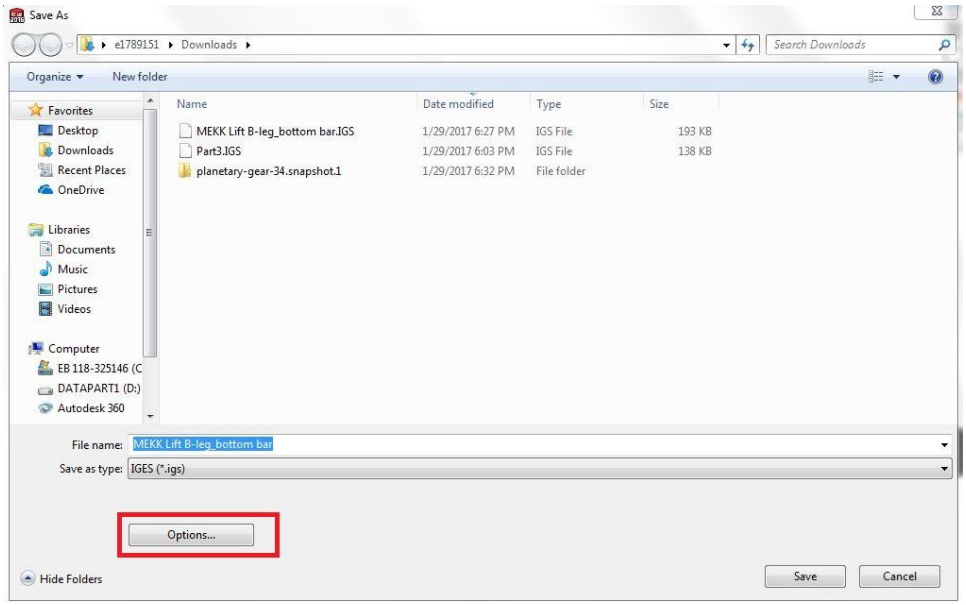
1) When the CAD model is finished select **"SAVE AS"** from the **Utility menu**



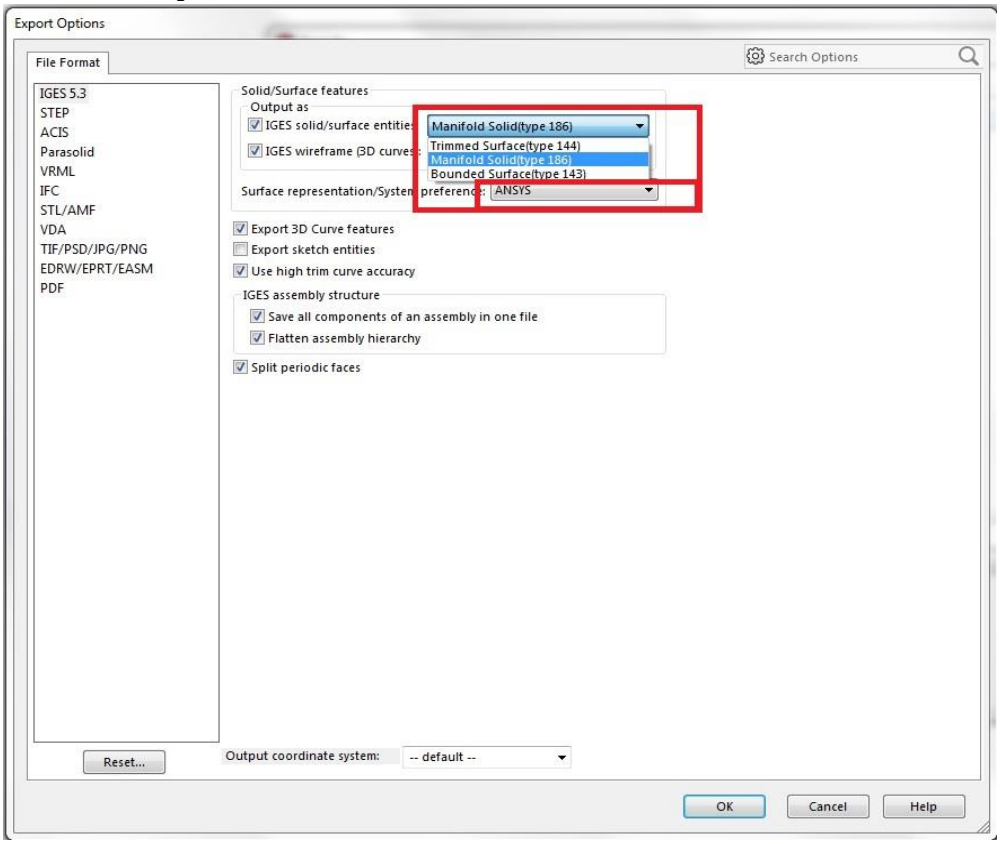
2) Select **IGES** as file type in Save as window



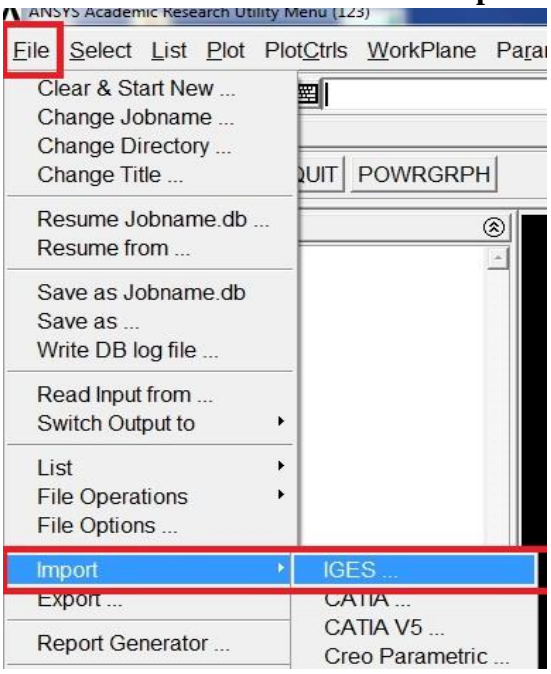
3) After selecting the file type select options in order to change appropriate settings for the file.



4) Depending on the type of geometry select the appropriate option for first dropdown menu (IGES Solid/Surface entities)
Important Note: the Surface representation/System preference should reflect **ANSYS** as the selected option. Then save the file.

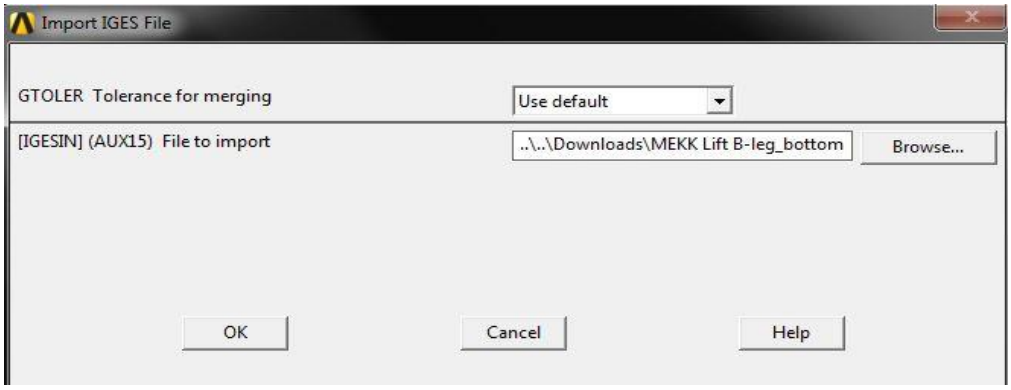


5) In ANSYS APDL Select **FILE > Import > IGES**

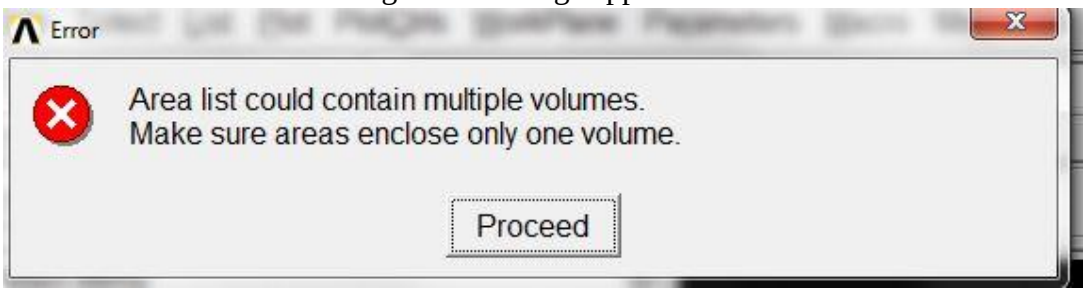


Make sure the following options are selected then click OK.

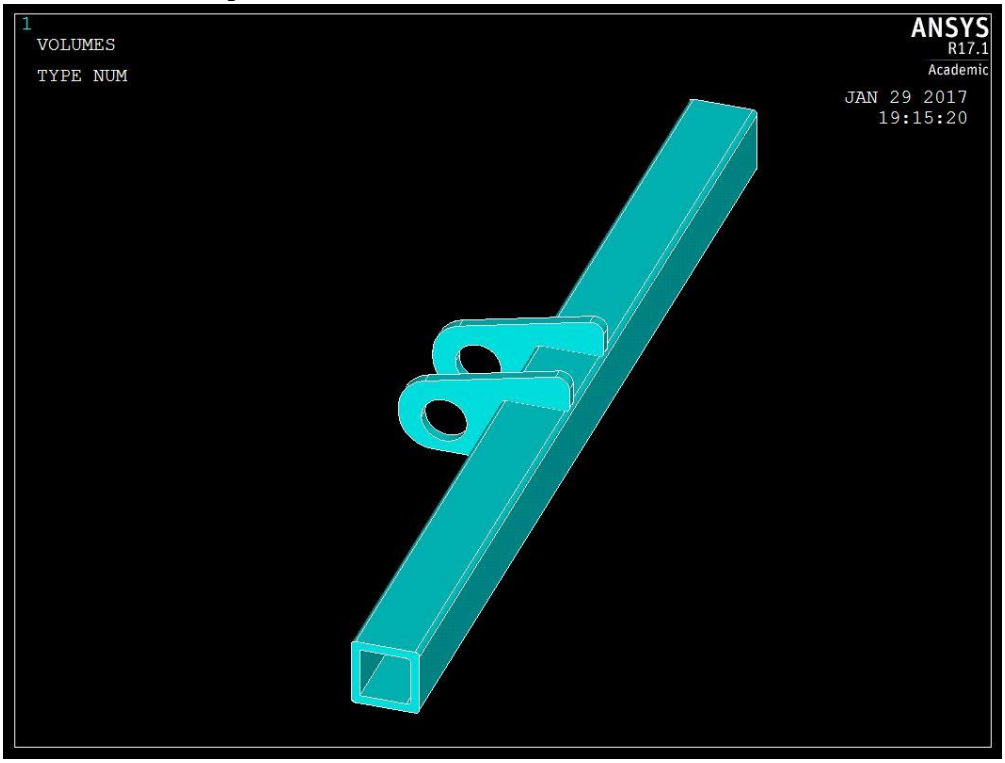
Use Default tolerance for merging.
Select the file name and click Ok to open.



Click **Proceed** if the following error message appears.

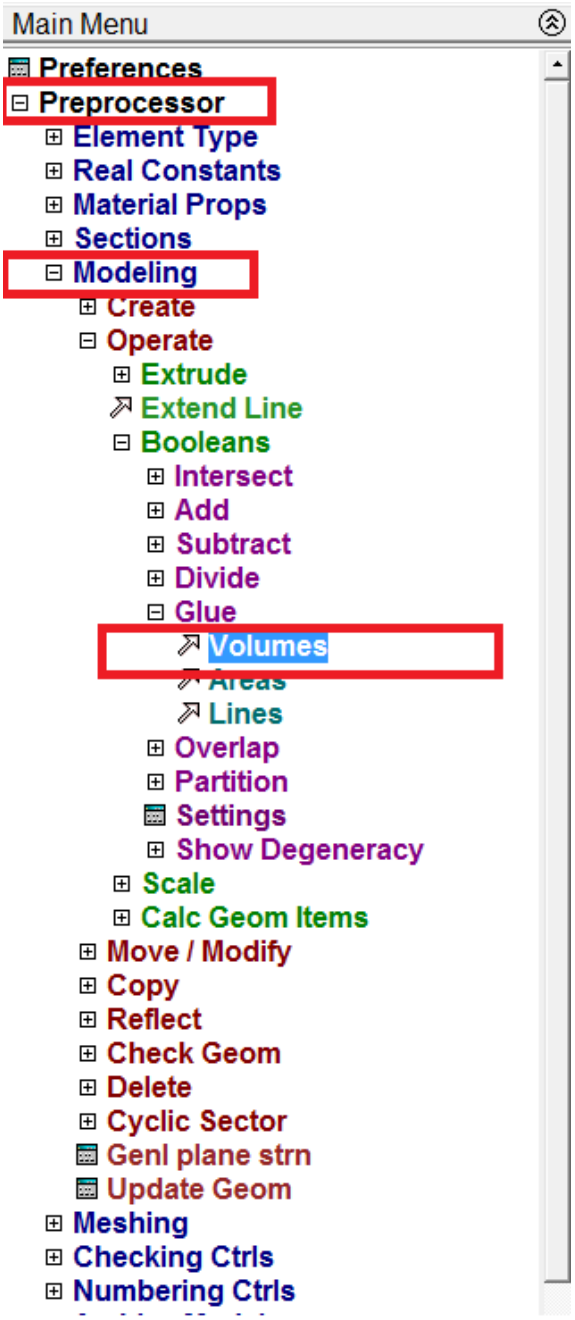


The desired file opens as a solid model



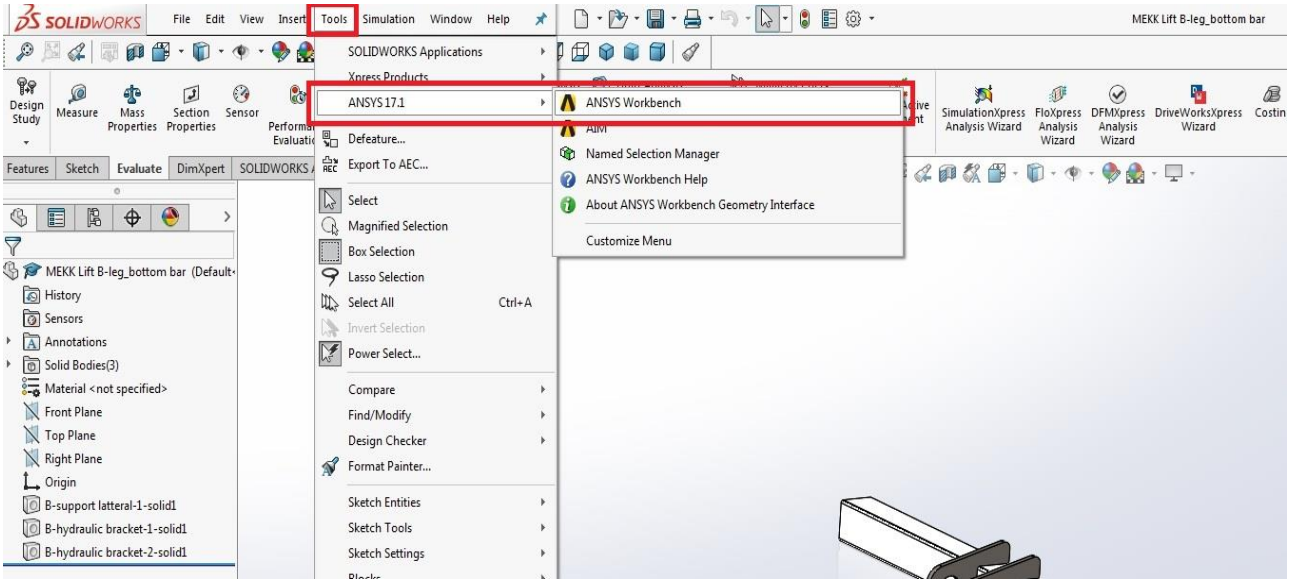
NOTE: Before proceeding to mesh the geometry make sure you **GLUE** all the volumes to avoid an error.

Preprocessor > Modelling > Operate > Booleans > Glue > Volumes > Pick all

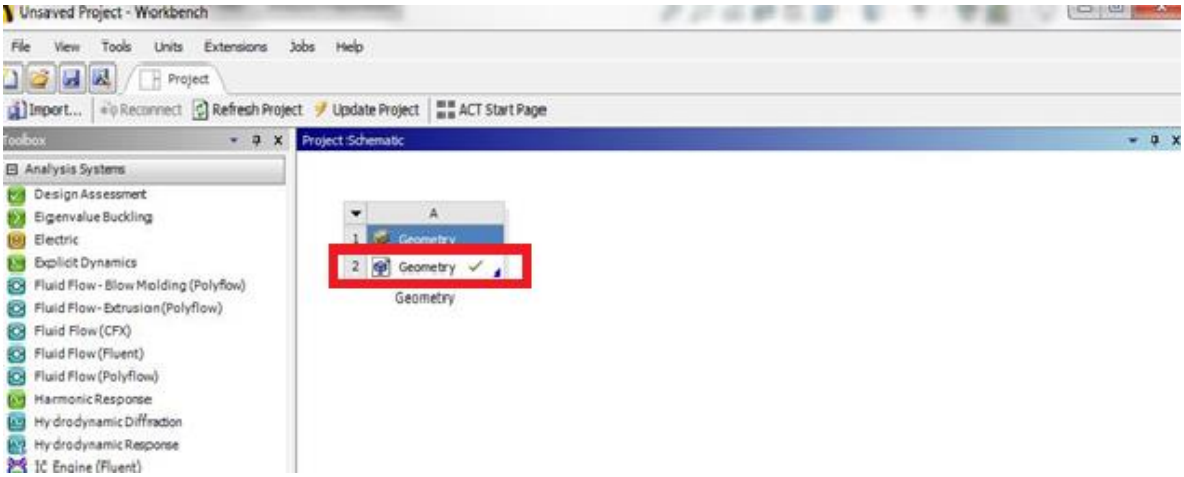


Importing file to ANSYS Workbench.

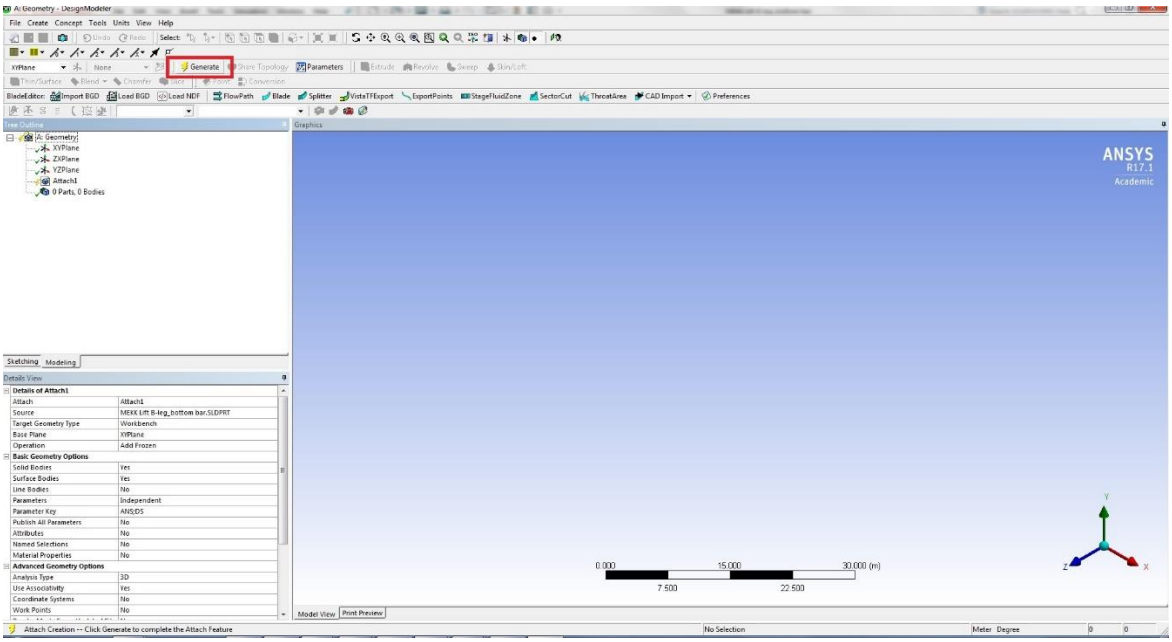
1) In Solidworks utility menu select **Tools > ANSYS 17.1> Workbench** in order to directly open a geometry file with the help of assisted Solidworks-ANSYS integration.



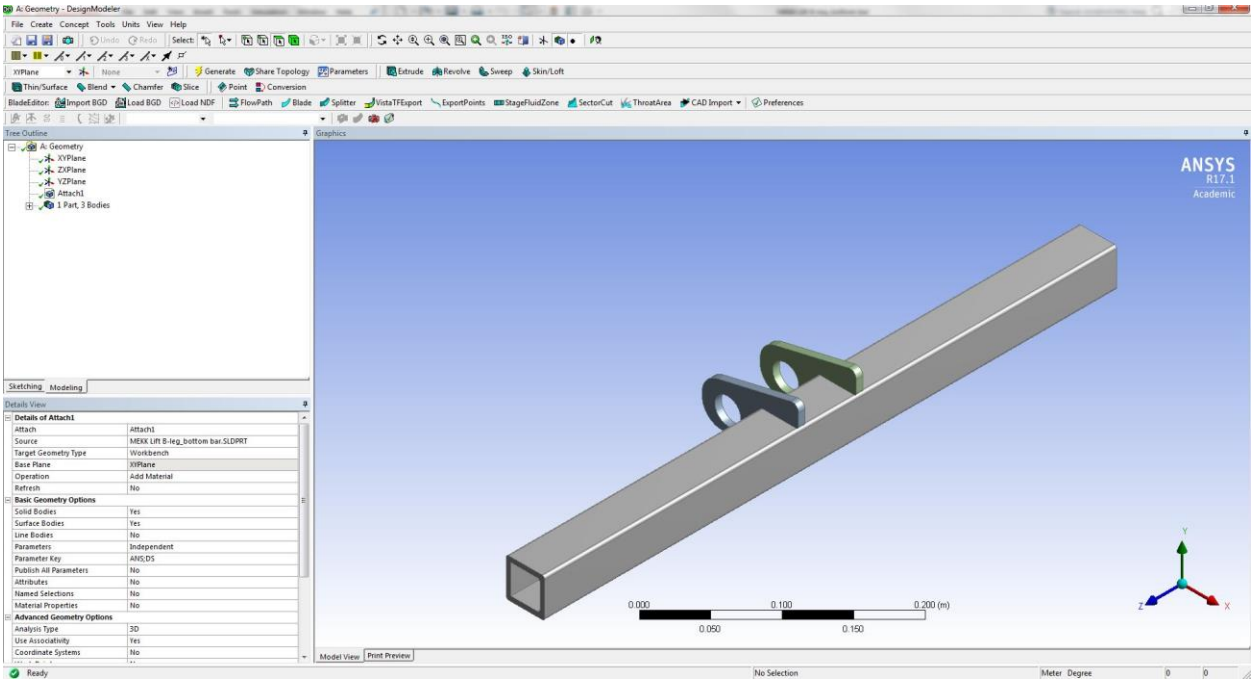
2) ANSYS Workbench opens automatically with a geometry block.



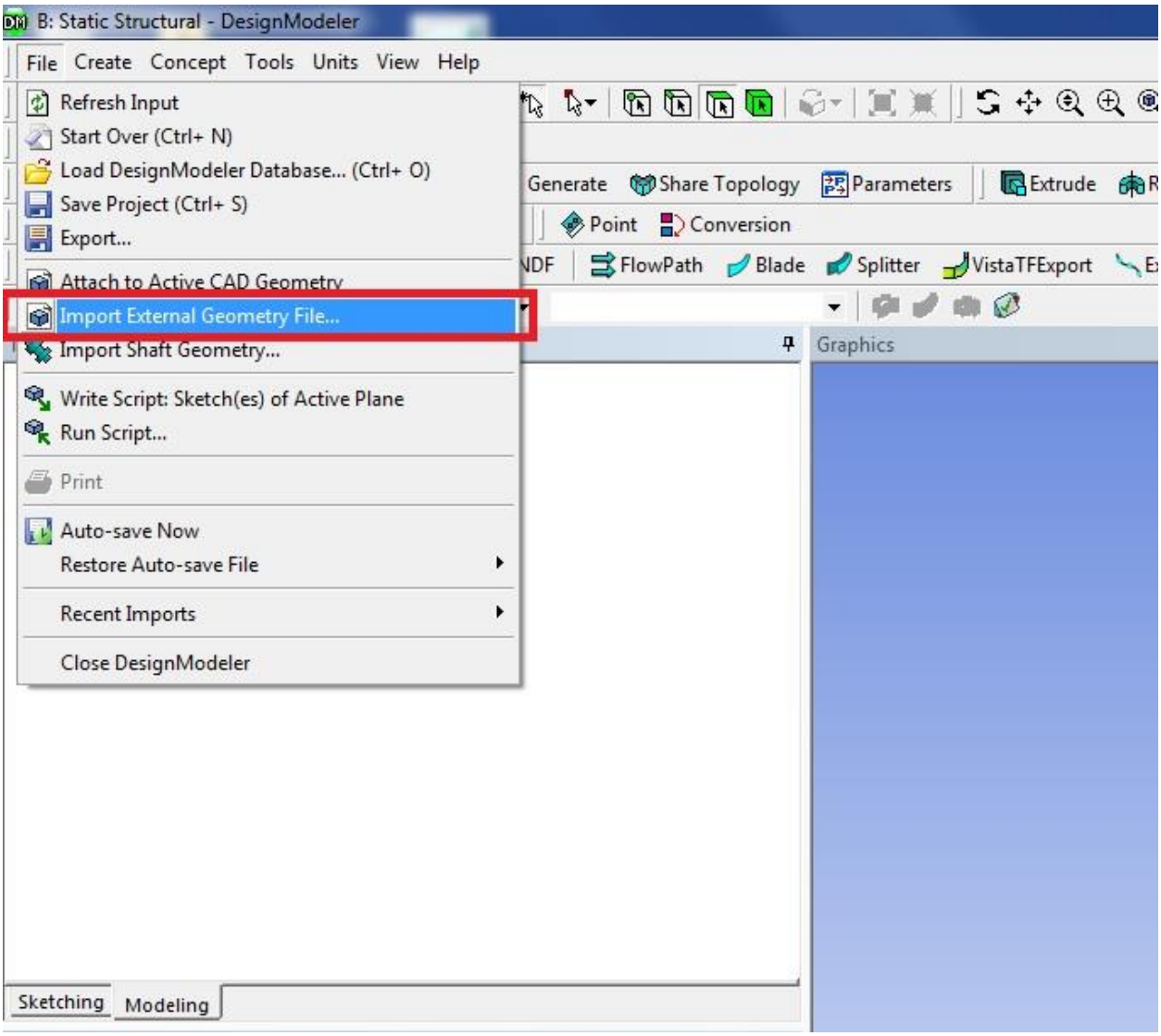
3) Double click the option -2 Geometry in order to open the design Modeler. Click generate to create the geometry.



The desired geometry opens in the design modeler and can be linked with any other analysis block in order to create mesh, apply loads and solve the loading condition to obtain results.



If the geometry is available in **IGES** file **(.igs)** or **ACIS** file **(.sat)** or **Parasolid (.x_t)** can be directly opened in ANSYS workbench design modeler by selecting **File > Input External Geometry File.**



Once the file loads up click **Generate** to get the model.