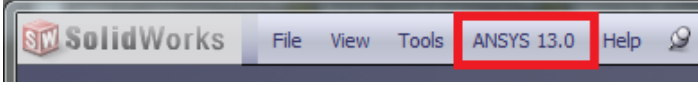


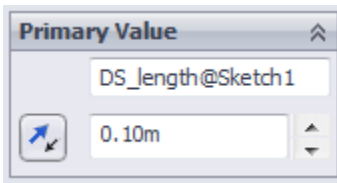
Plate With a Hole Optimization - Parameter Setup in Solidworks

Defining Parameters in Solidworks to be used in ANSYS Workbench

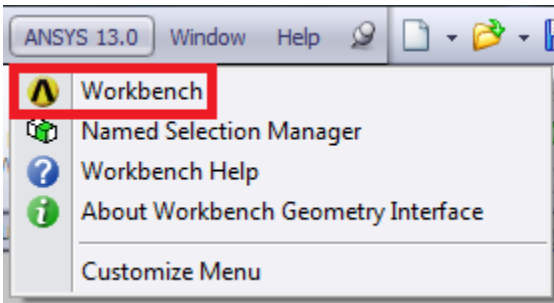
Open Solidworks, and confirm that the "ANSYS 13.0" button appears on the toolbar. This shows that the ANSYS-Solidworks plug-in has been installed.



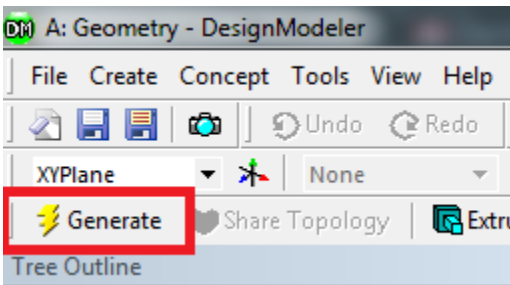
Create the Solidworks part/assembly as you normally would. Next, we need to define the parameters so they will be imported into ANSYS. In SolidWorks, open the part and then click on the part or on the feature in the tree. Then right-click the dimension on the model, and edit the name of the dimension under the "Primary Value" window. An important note about parameter names: the key to ANSYS recognizing a CAD parameter are the letters DS or ds. Also, parameter names cannot contain spaces or dashes "-", so separate with an underscore, "_", instead. Examples of acceptable parameter names would be DS_radius, lengthds, or DSwidth.



After following these steps, we now need to open the geometry in ANSYS. To do so, we can simply click the Workbench button in the ANSYS 13.0 menu.

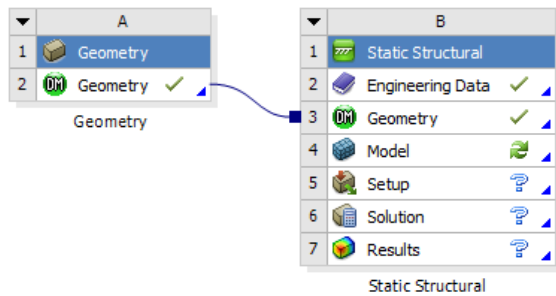


After ANSYS initializes, we will have an imported geometry. First, double click on the Geometry to open it and check that it has imported correctly. After opening the Design Modeler, click Generate.



After doing this return to the main project window and simply drag the type of analysis desired, such as "Static Structural" in the plate with a hole optimization, onto the Geometry to link the two. Your Project Schematic window should appear similar to the image below, depending on the type of analysis selected.

Project Schematic



You can now follow the steps listed in the Plate with a Hole Optimization tutorial now that your parameters have been defined.

[Return to Step 3-Input & Output Parameters](#)

[See and rate the complete Learning Module](#)

[Go to all ANSYS Learning Modules](#)