PCB Library Expert to Altium Designer 17 Via Script for creating a New Library
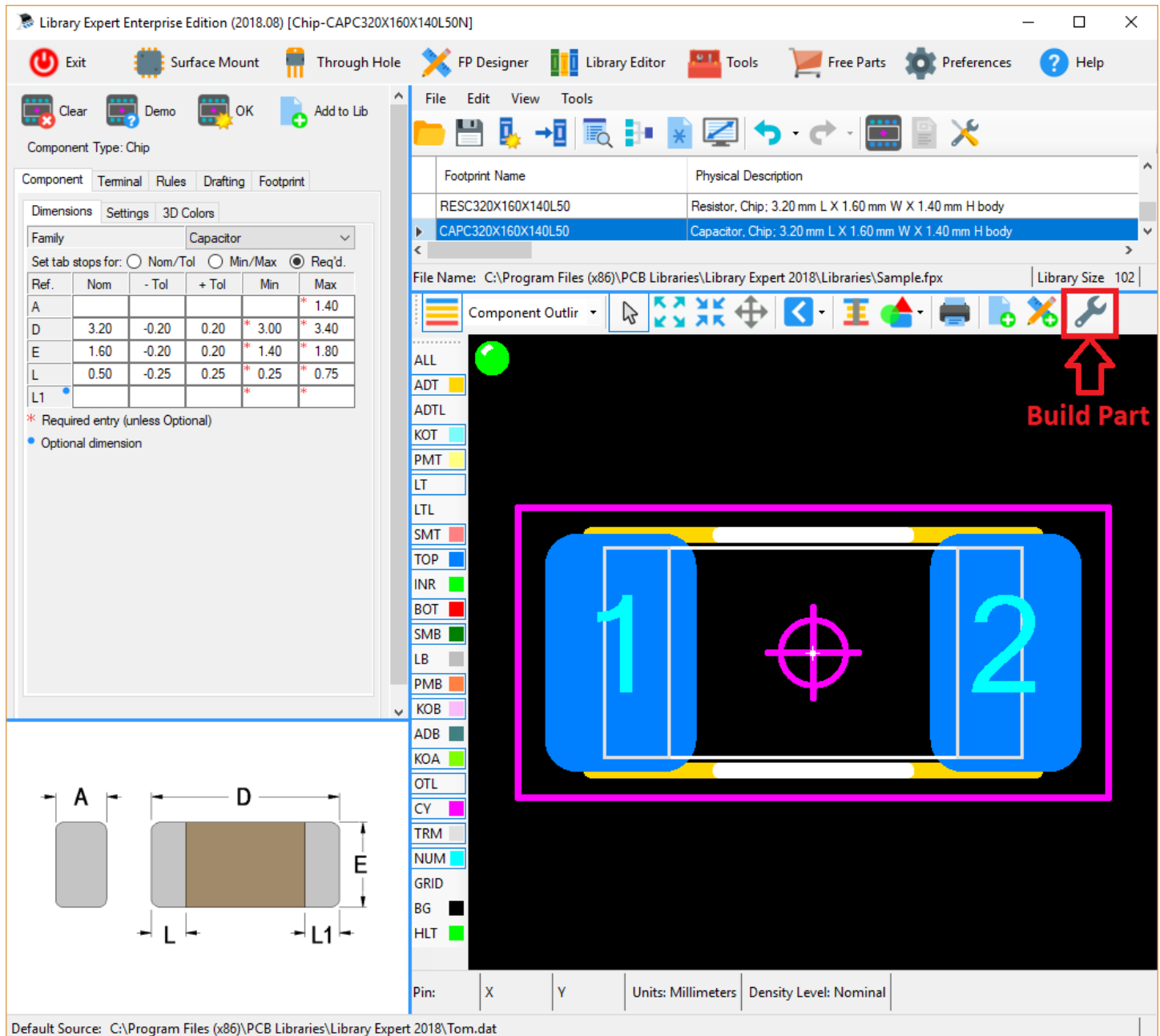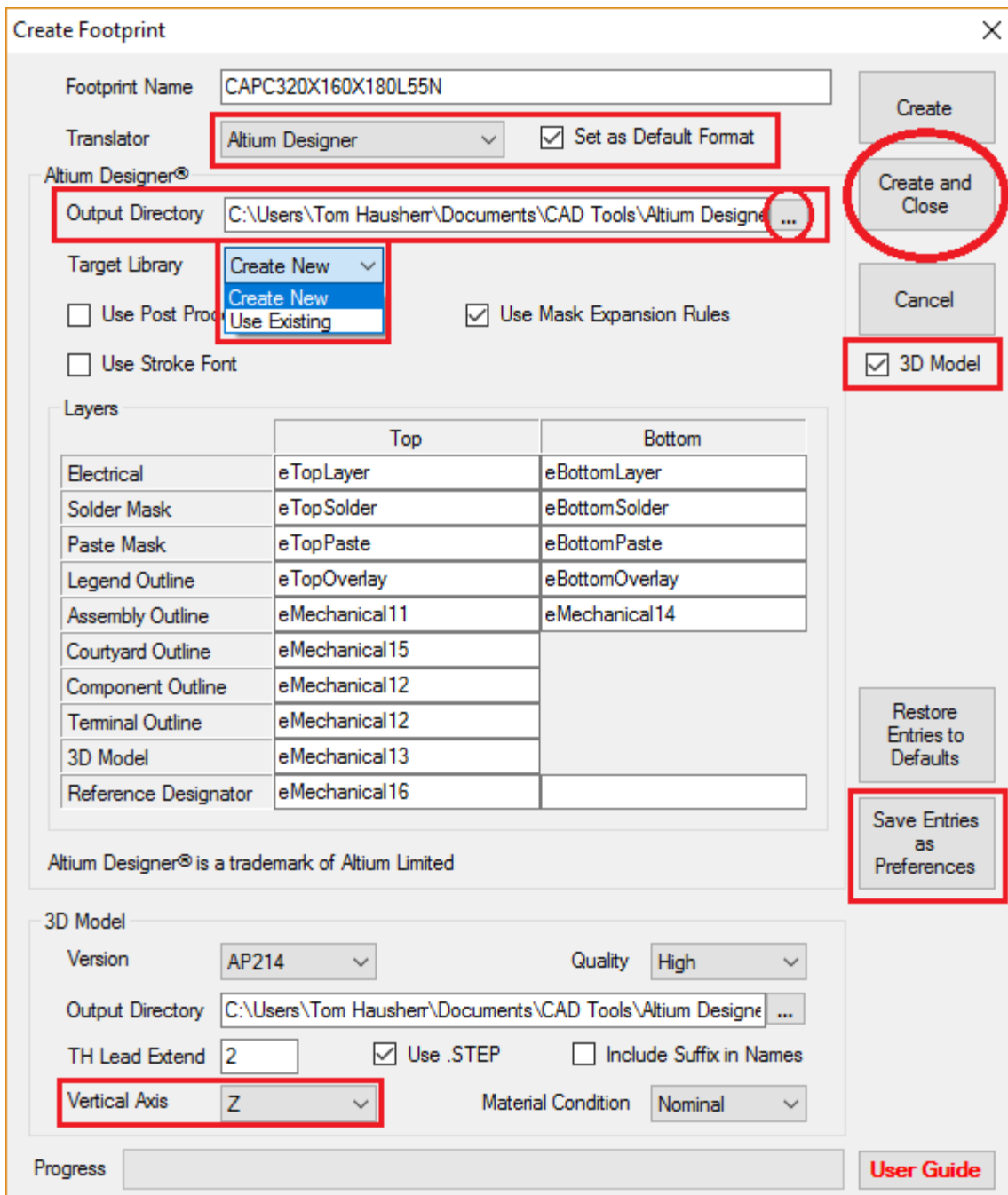
To create a footprint, click the "**Add Footprint to CAD Library**" Button. The Wrench icon in the toolbar.

The CAD tool interface dialog box will open. Select the Altium Translator and select the radio button "Set as Default Format", select the Output Directory folder for the Script files, select the Output Directory folder for the 3D STEP model, select Vertical Axis "**Z**", select the "Save Entries as Preferences" and last select the "Create and Close" button.

The "Target Library" selection is for adding the library part to an existing library or creating a new library.
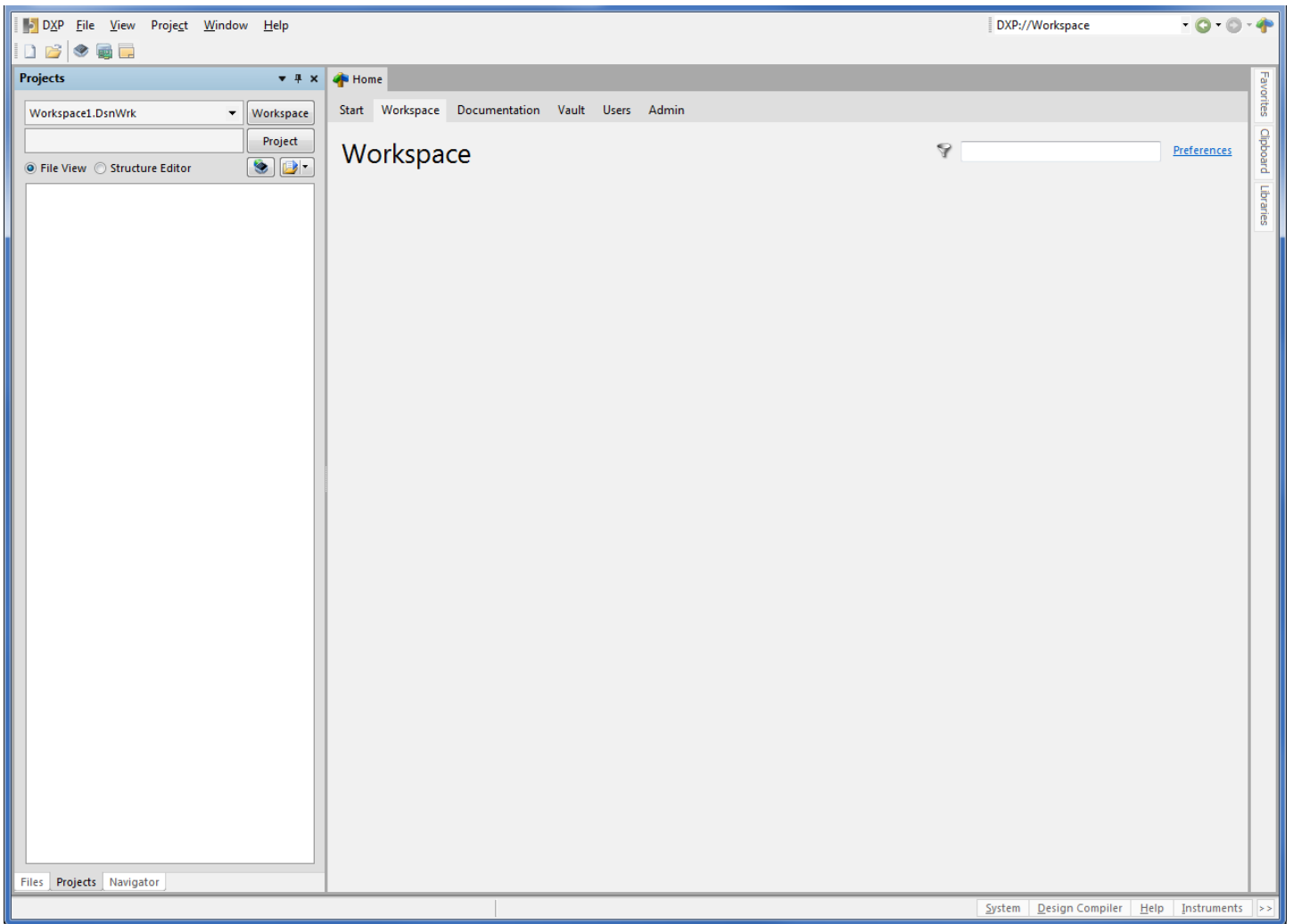
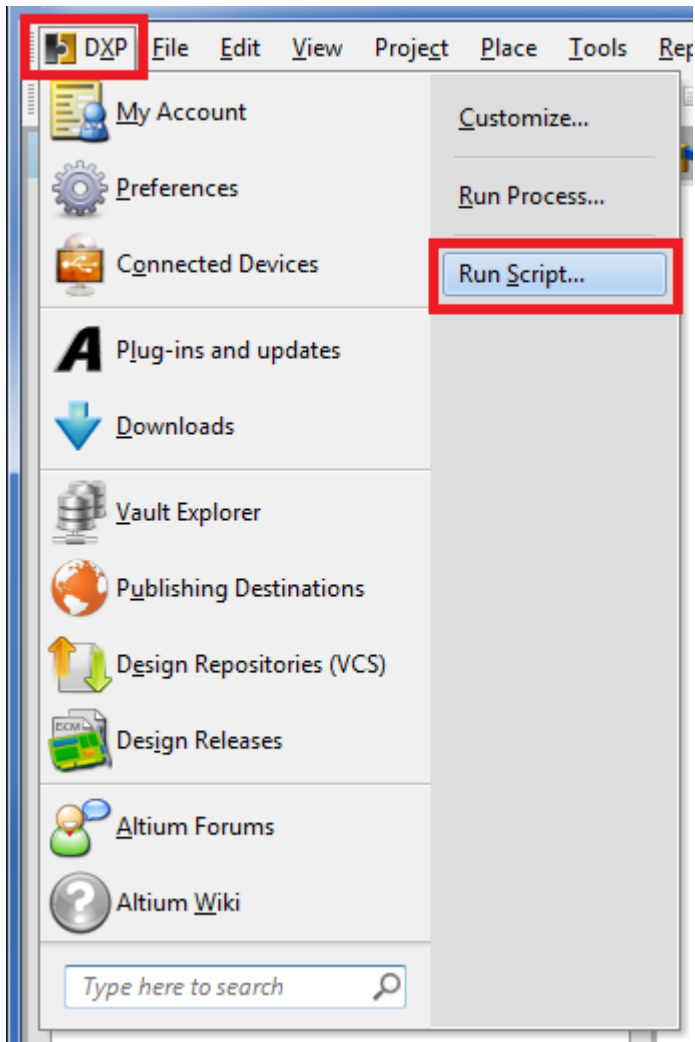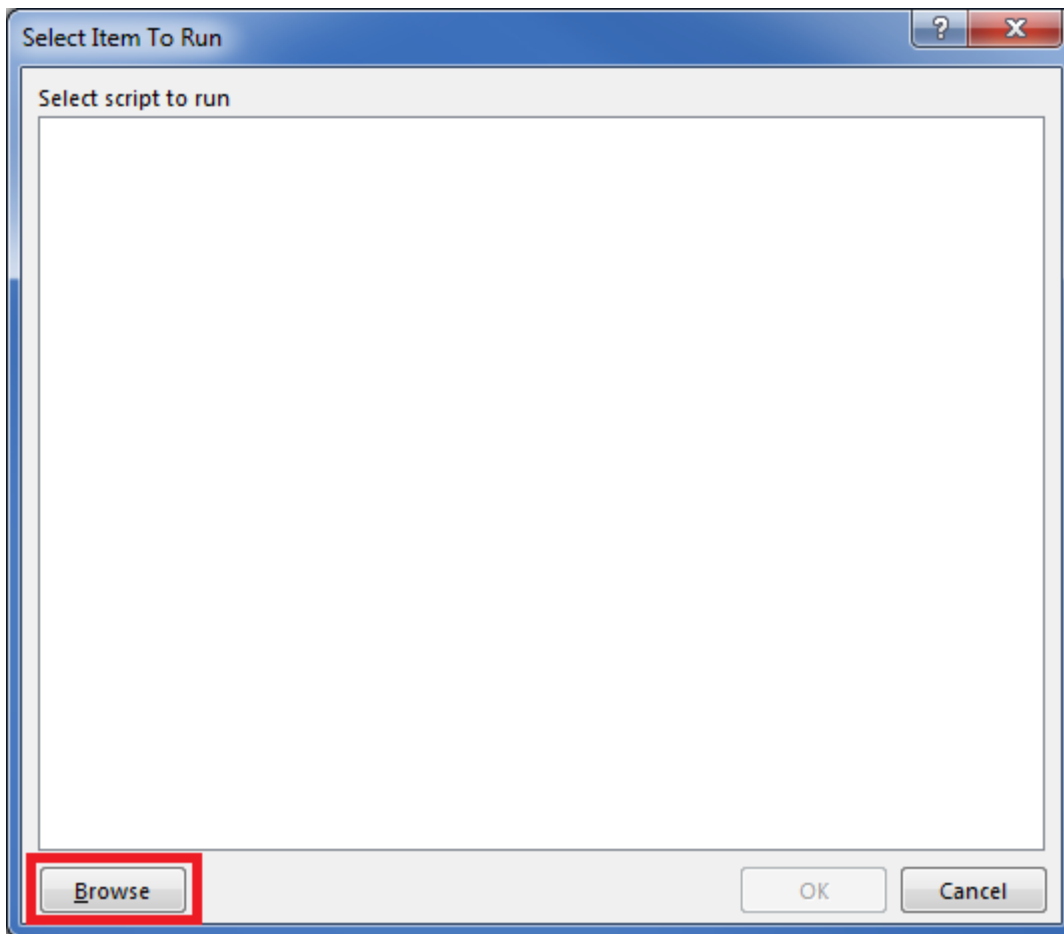These 3 files will be created in the Output folder that you defined:
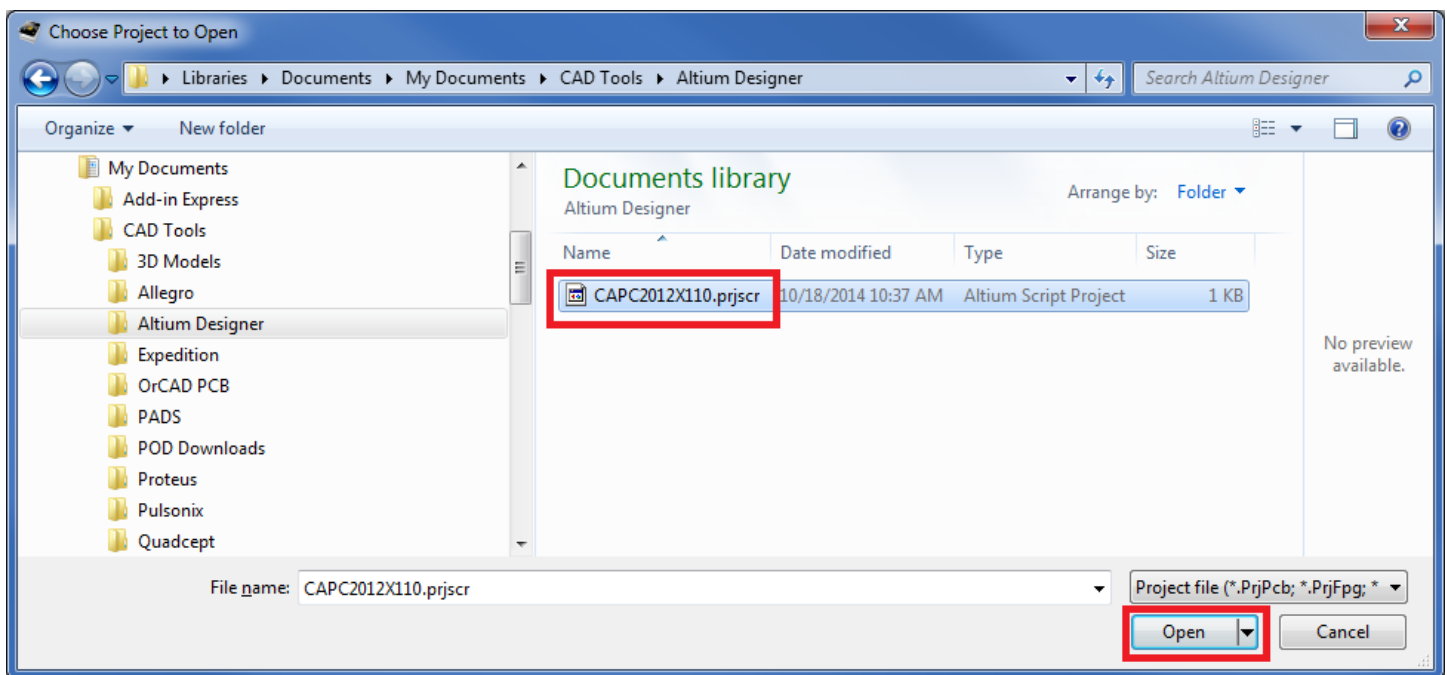

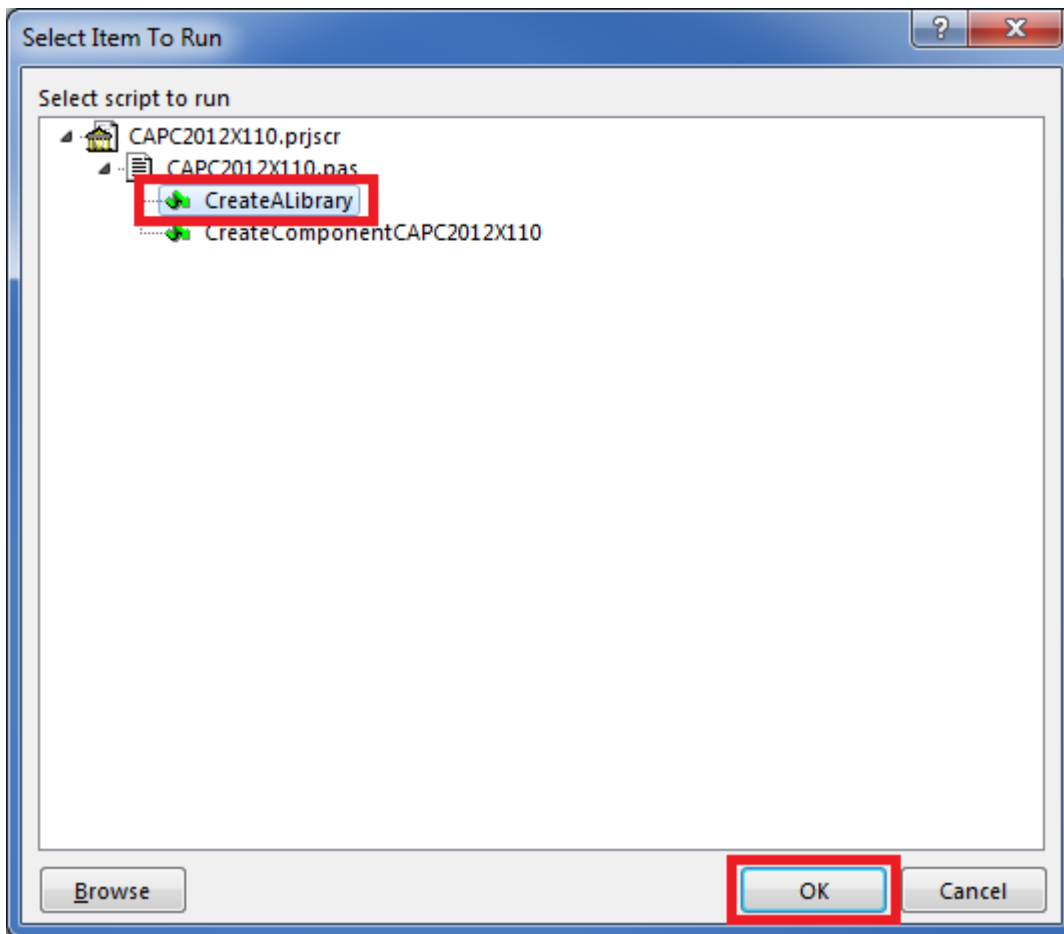Open Altium:

Select "**DXP > Run Script**"

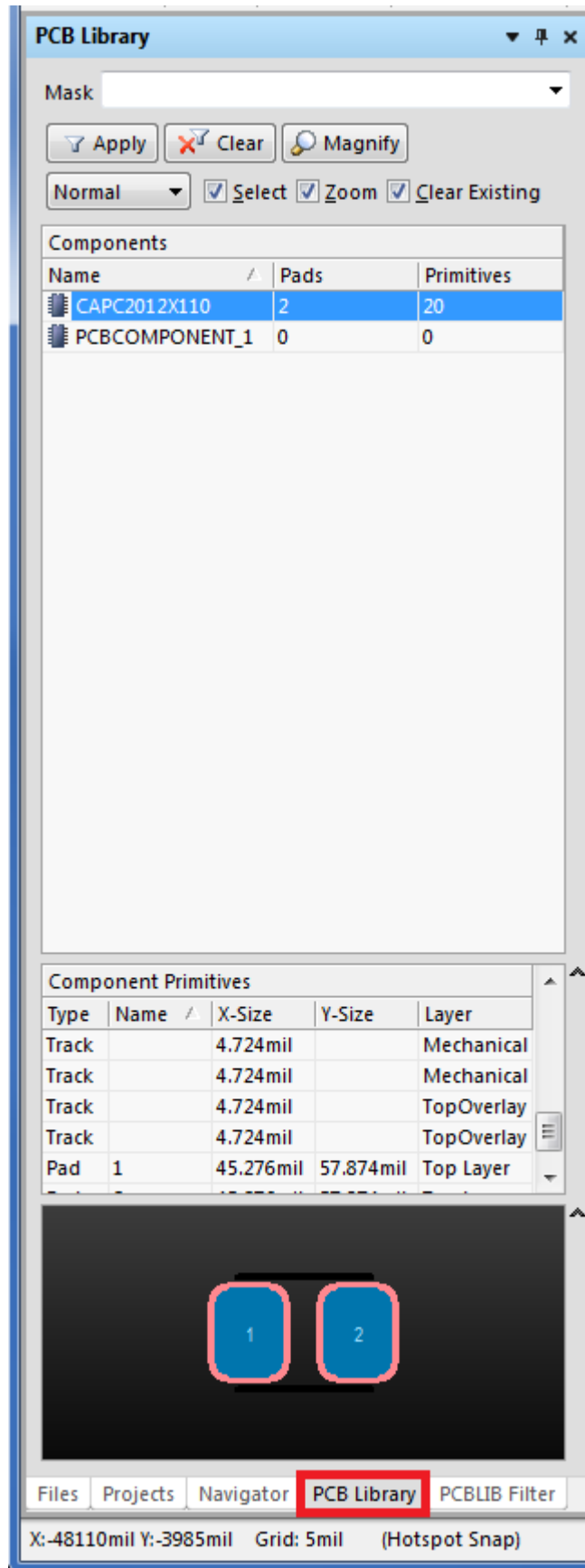"**Select Item to run**" dialog window will appear. Select the "**Browse**" button.

Browse to the folder that you set up in the Library Expert "Build Part" dialog window and Select the ".**prjscr**" file and then select the "**Open**" button
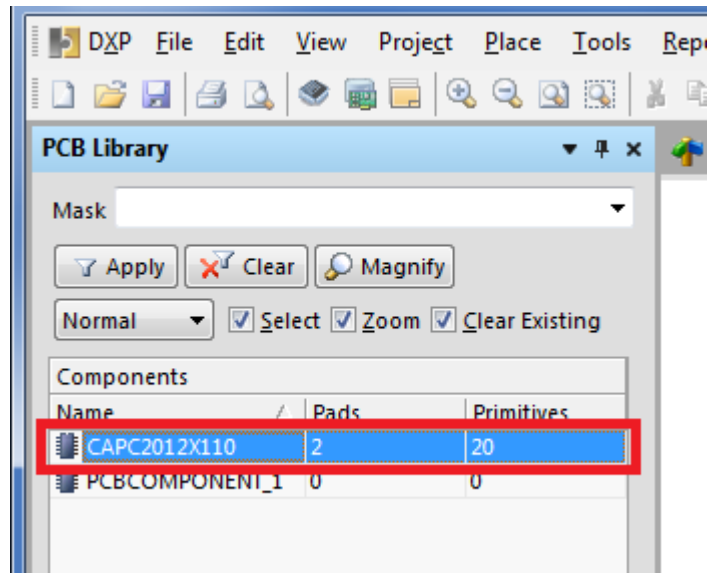

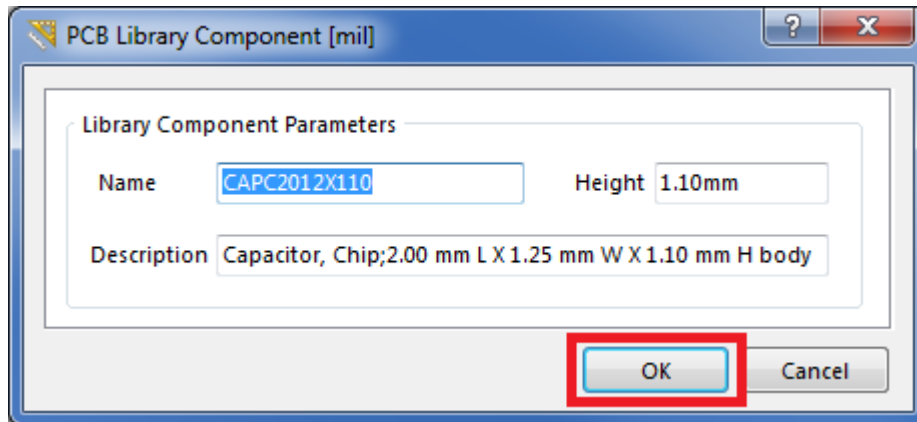Select "**CreateALibrary**" and click "**OK**"

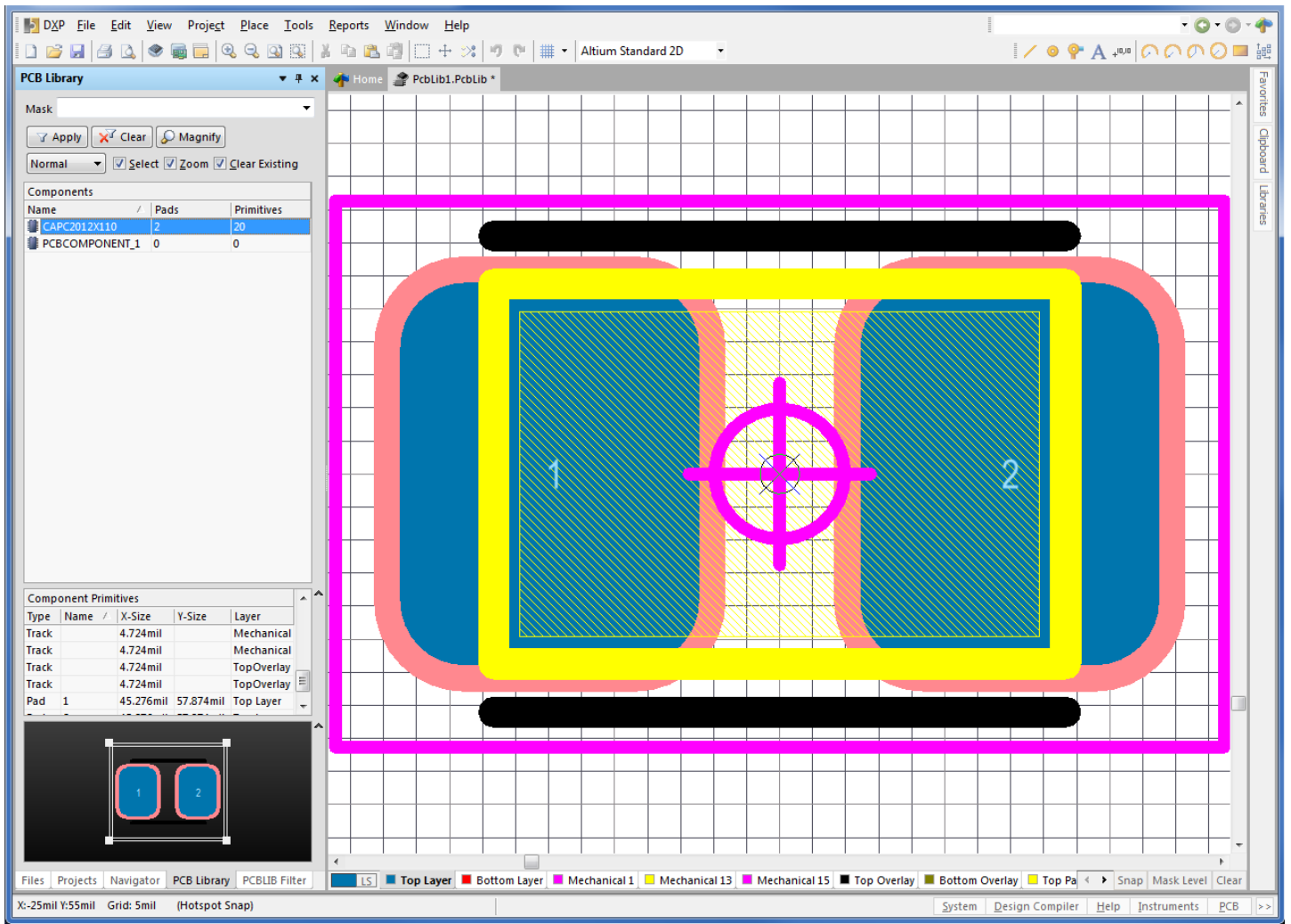Select "**PCB Library**" at the lower left of the project window to display the library components

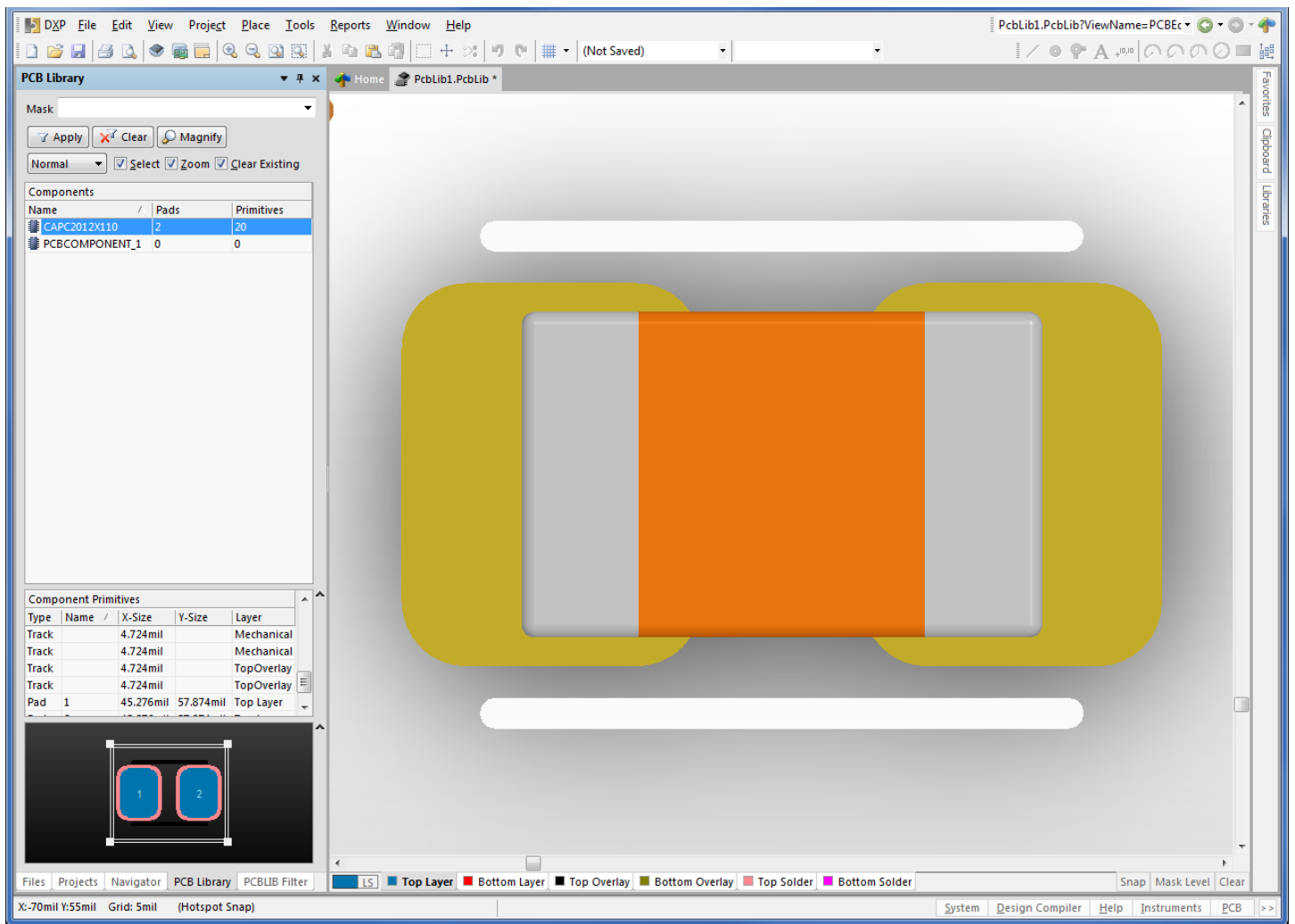Check Attributes: Double Click on the library component Name

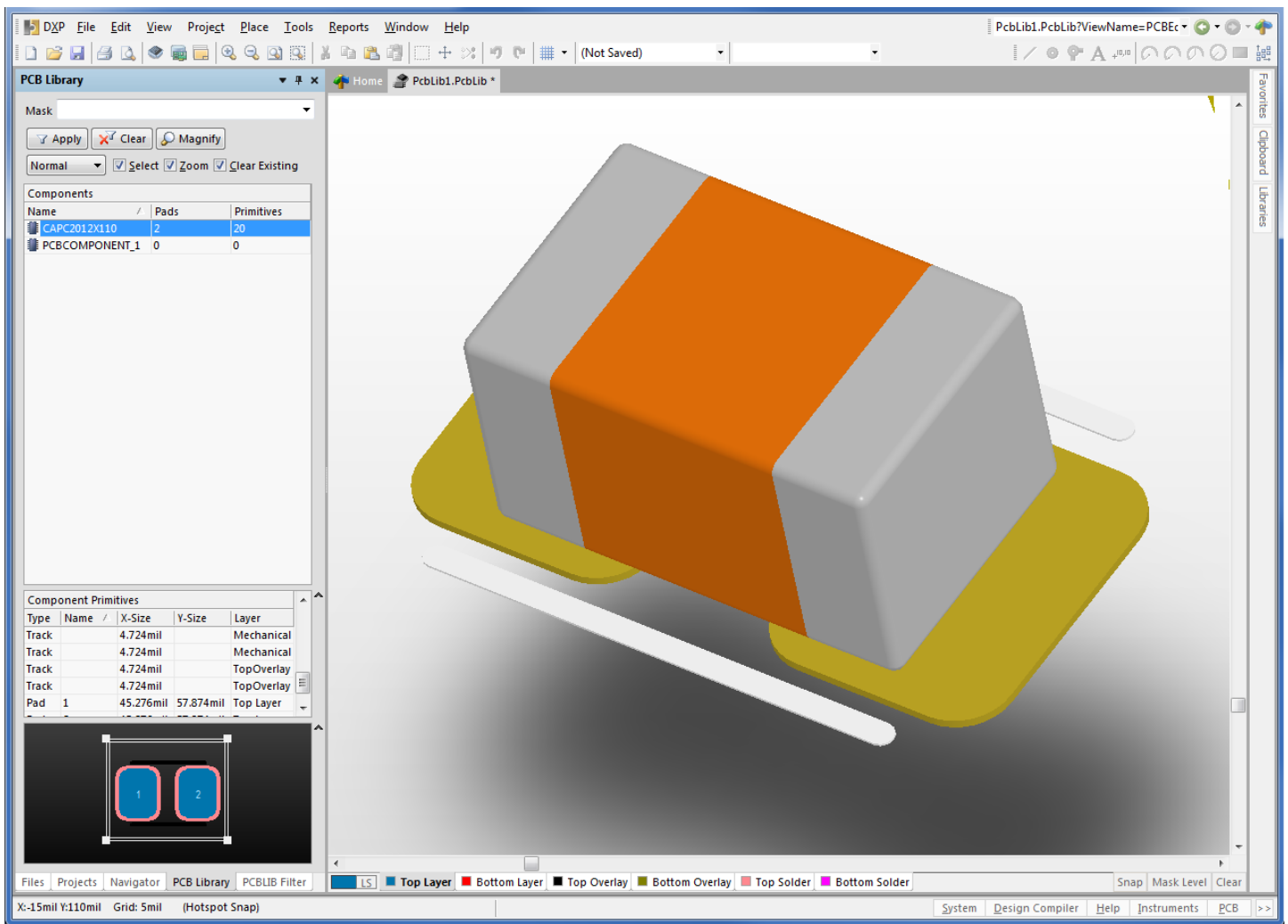Notice that the Description and Height are automatically populated. Select the "**OK**" button.

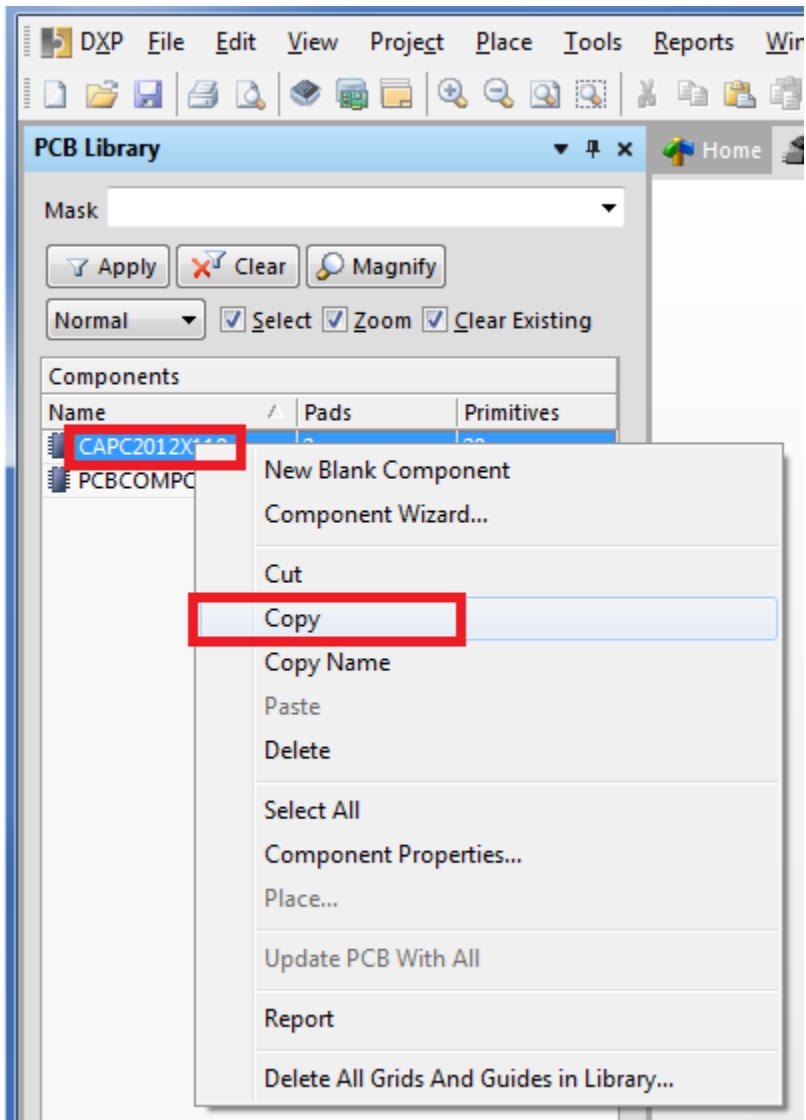The library footprint will appear with the 3D Model automatically placed.

Select the keyboard "3" to go into 3D mode

Use the "Shift + Middle Mouse Button" to rotate the 3D model

Right Mouse Button on the Library **Part Name** and select **Copy**

Open your main PCB Library and **Right Mouse Button > Paste** the new part into your personal library.