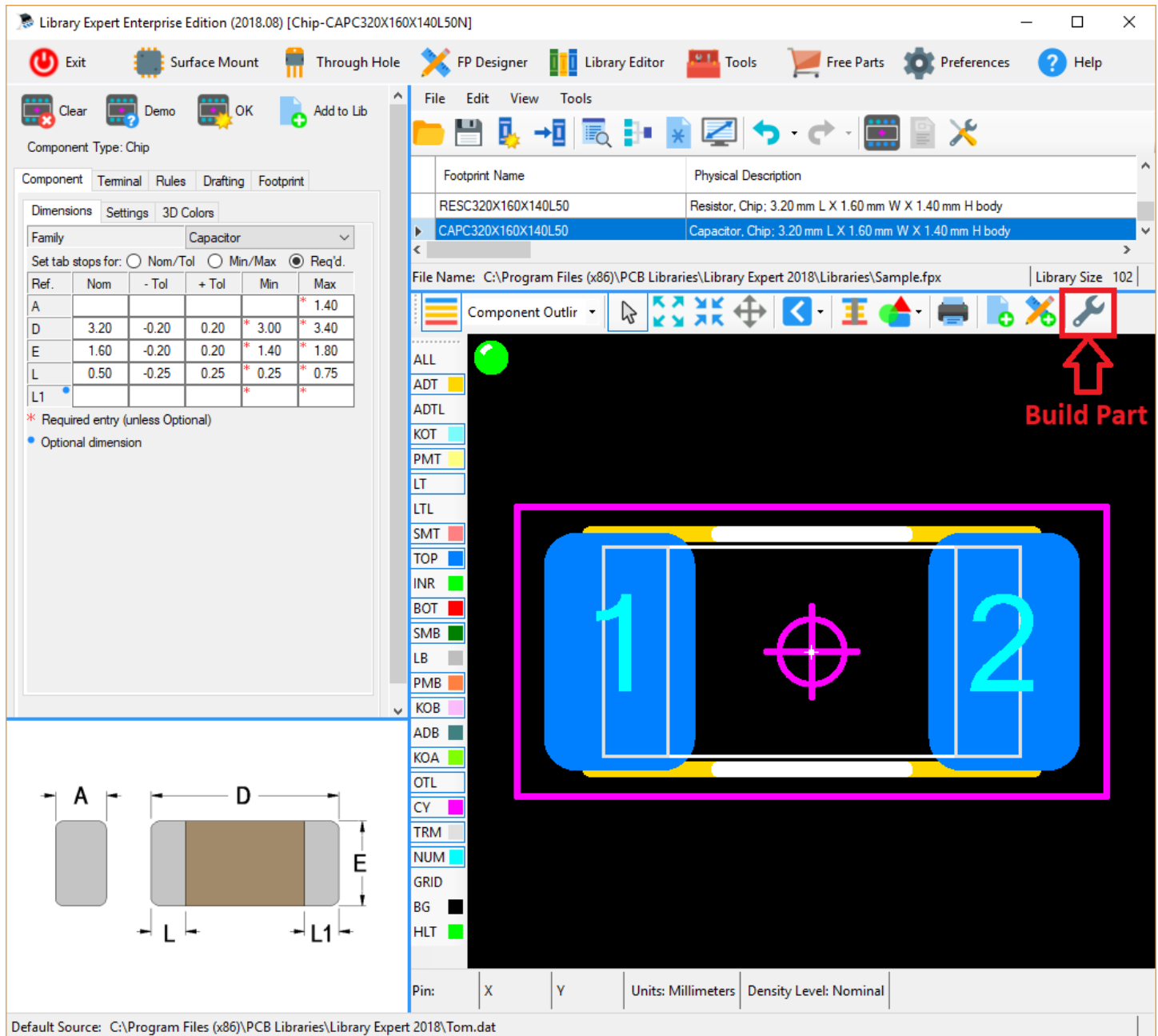


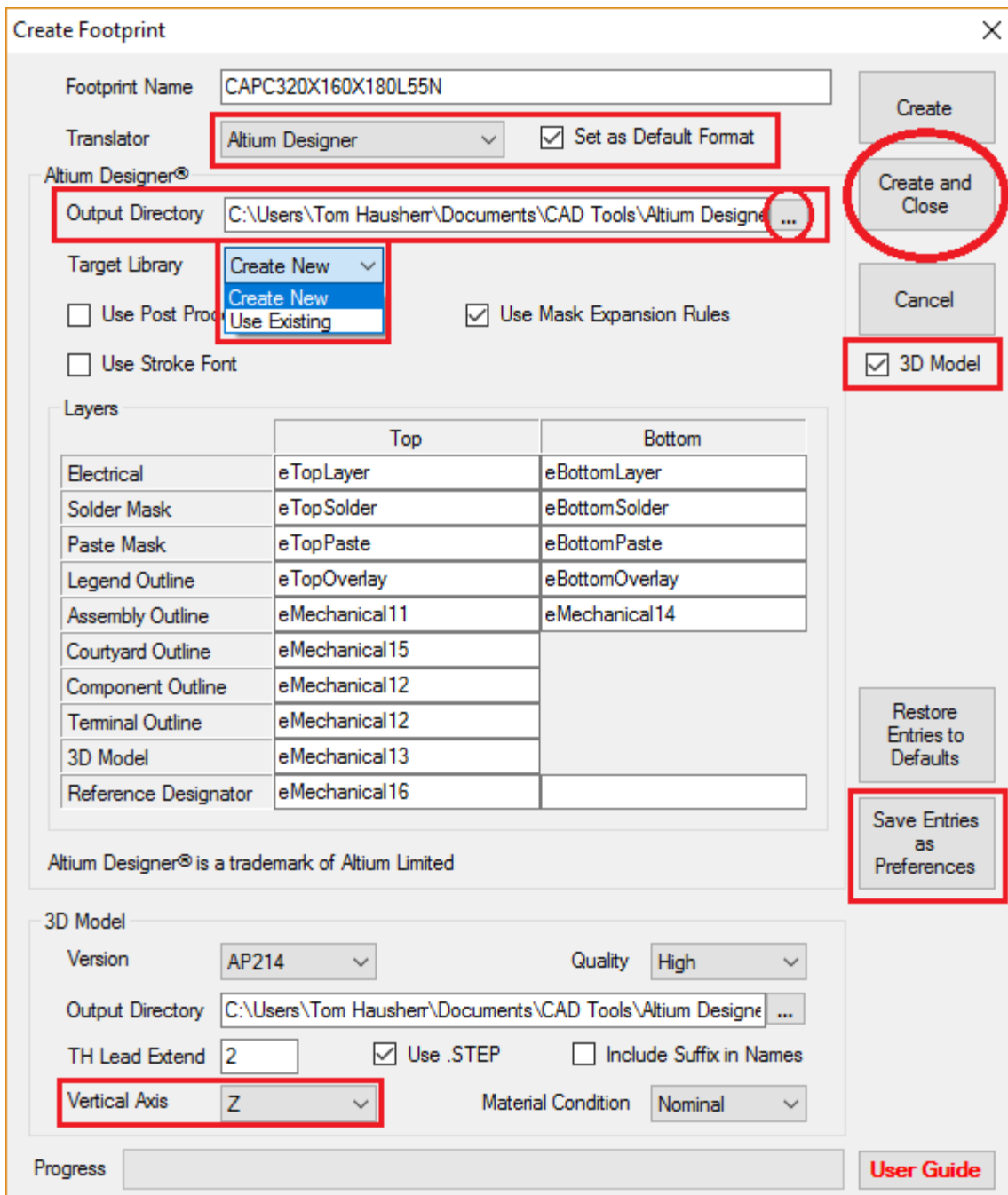
PCB Library Expert to Altium Designer 18 Via Script for creating a New Library

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.

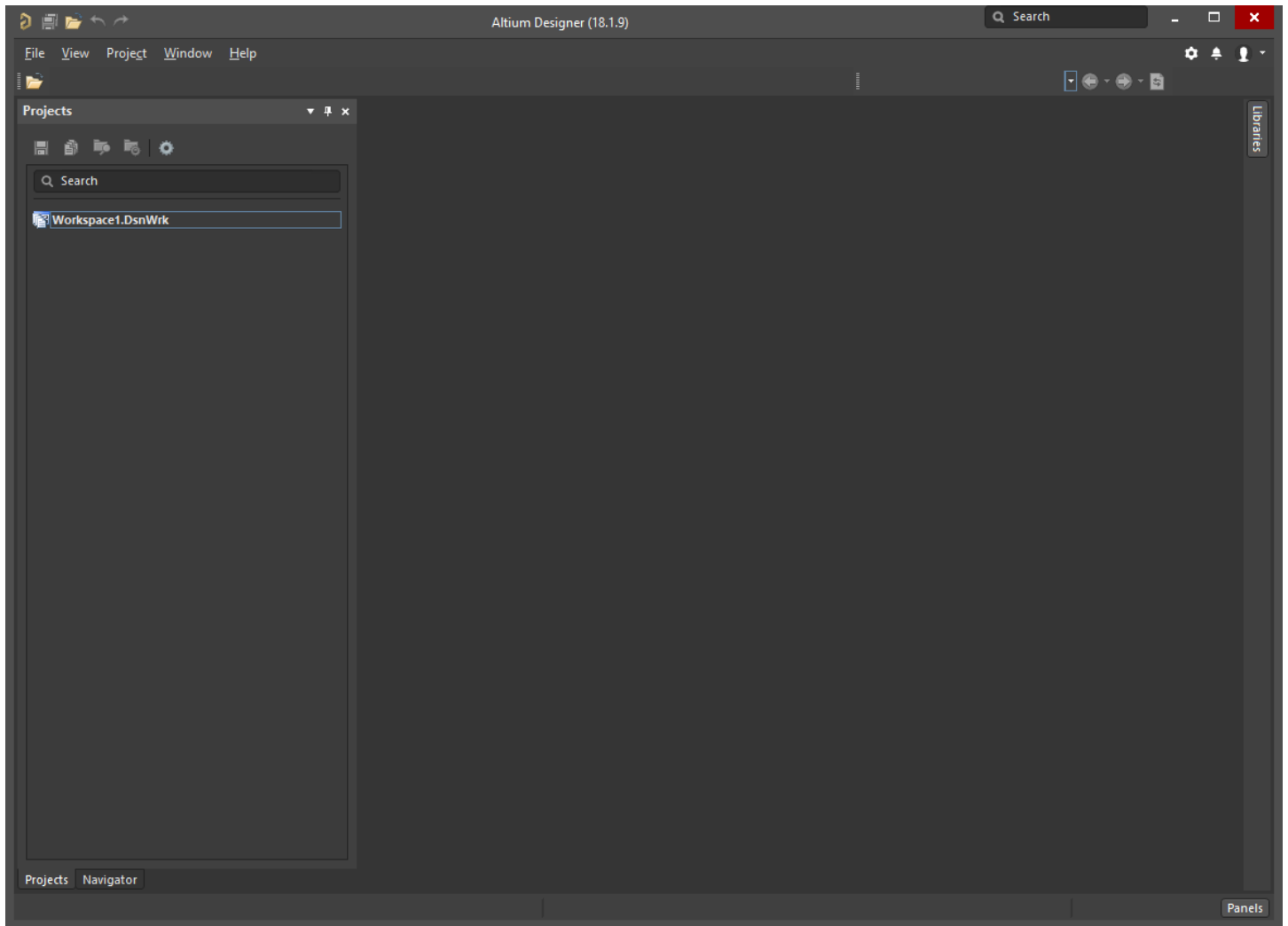
The “Target Library” selection is for adding the library part to an existing library or creating a new library.



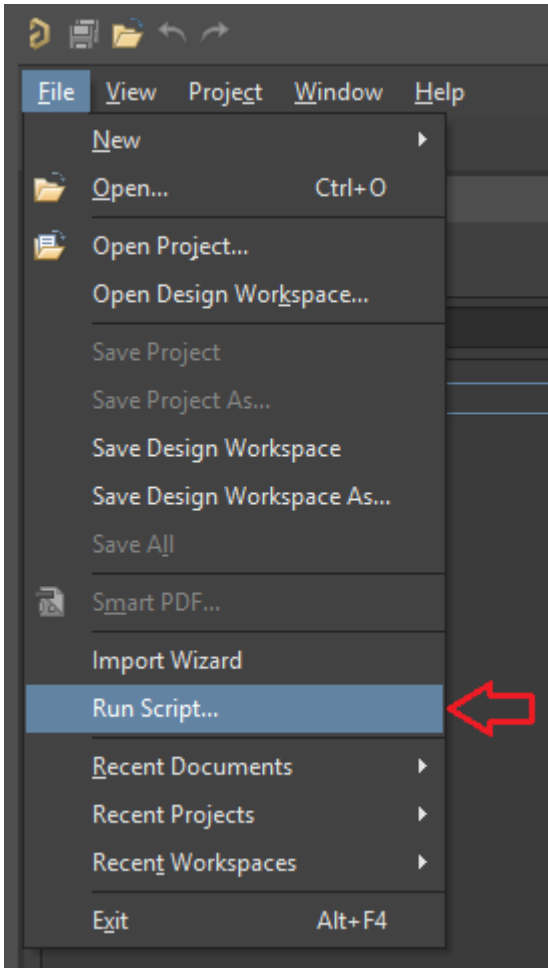
These 3 files will be created in the Output folder that you defined:

Name	Date modified	Type	Size
 CAPC320X160X140L50.STEP	11/8/2018 3:05 PM	STEP File	93 KB
 CAPC320X160X140L50N.pas	11/8/2018 3:05 PM	PAS File	12 KB
 CAPC320X160X140L50N.prjscr	11/8/2018 3:05 PM	Altium Script Proj	1 KB

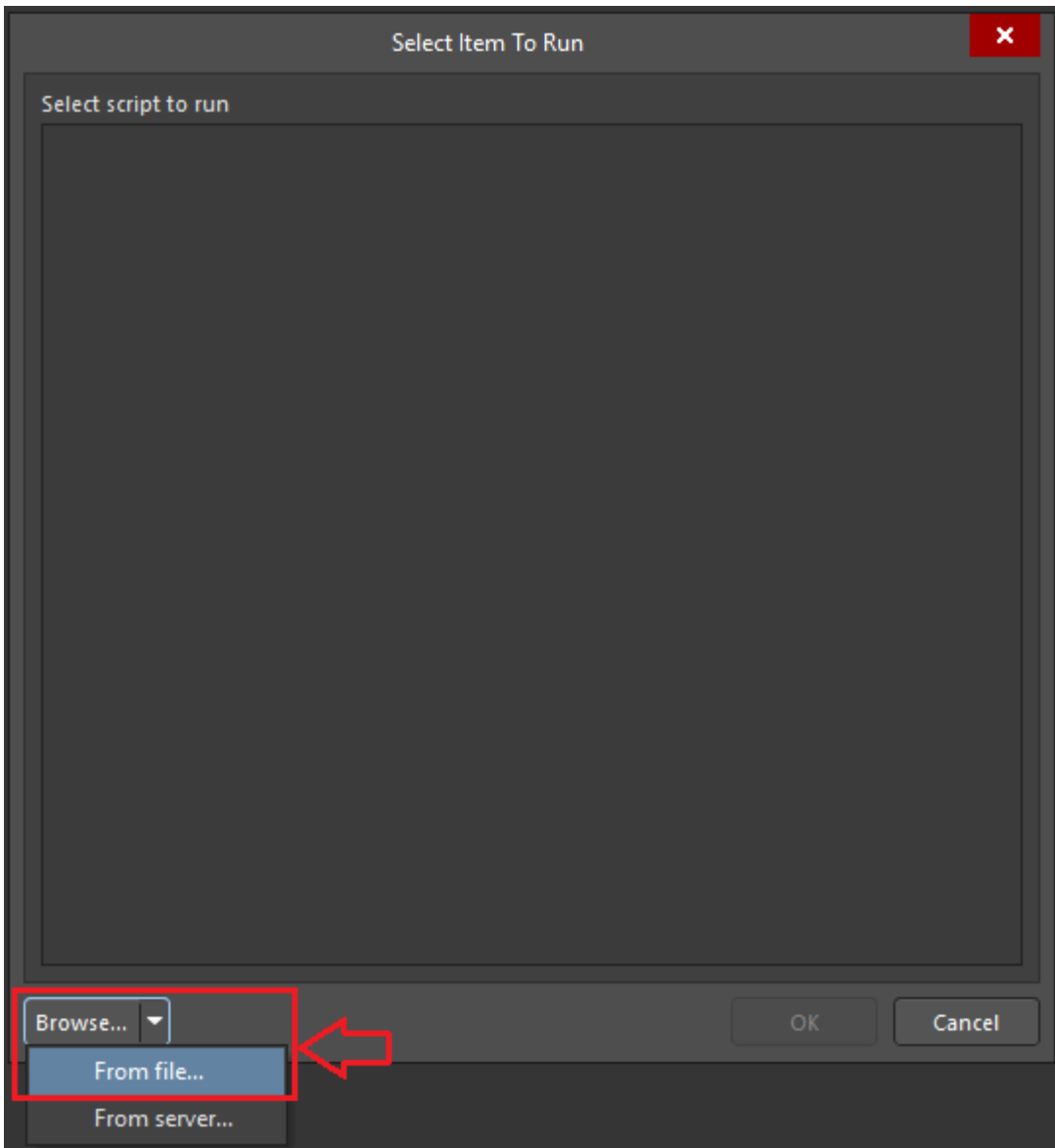
Open Altium:



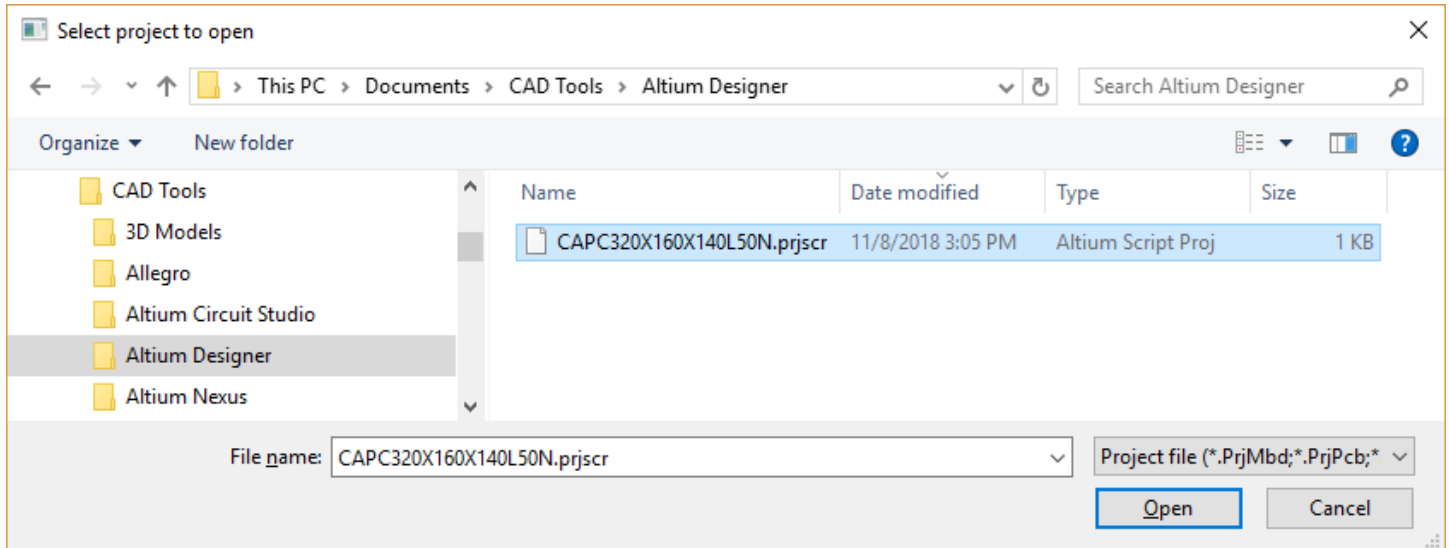
Select "File > Run Script"



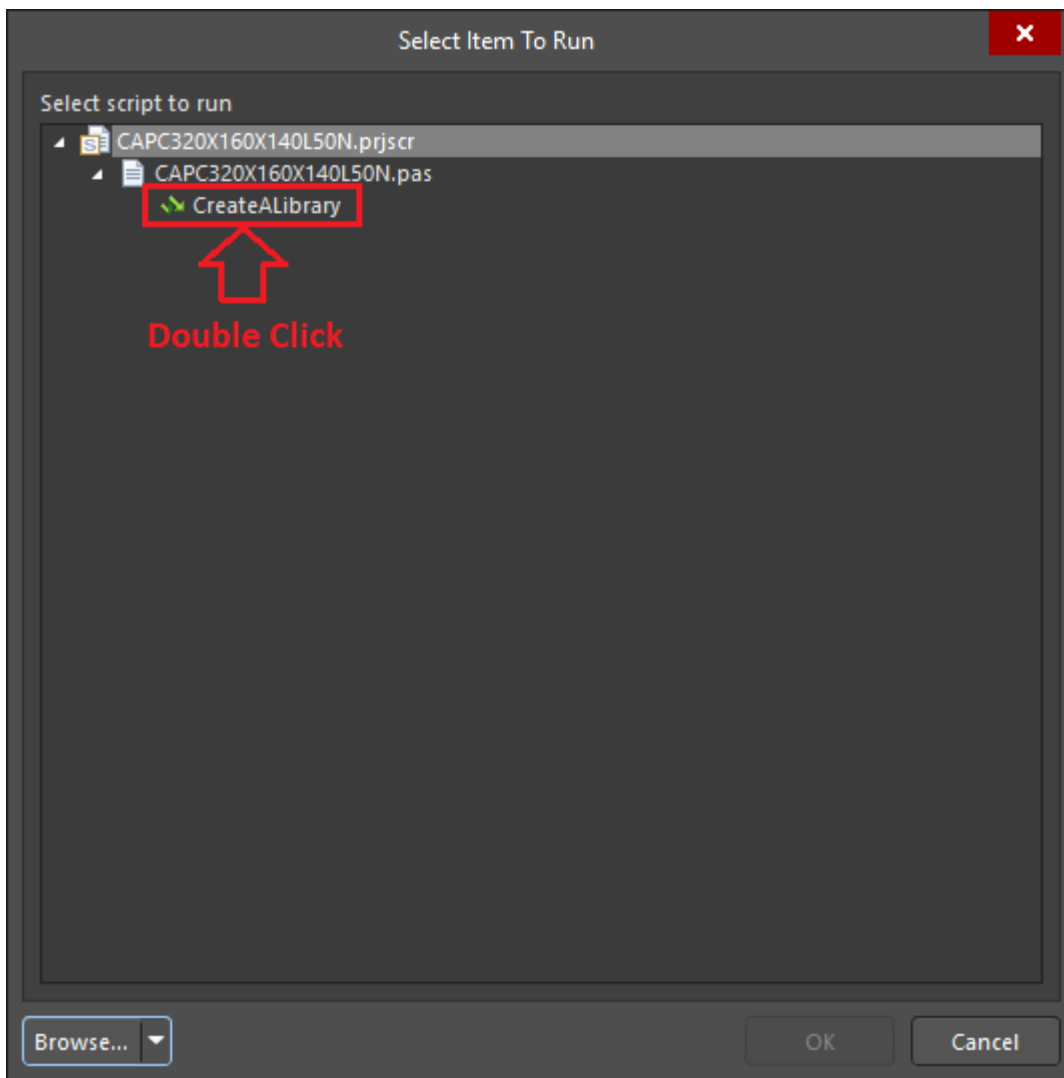
“Select Run Script” dialog window will appear. Select the “Browse > From File” button.



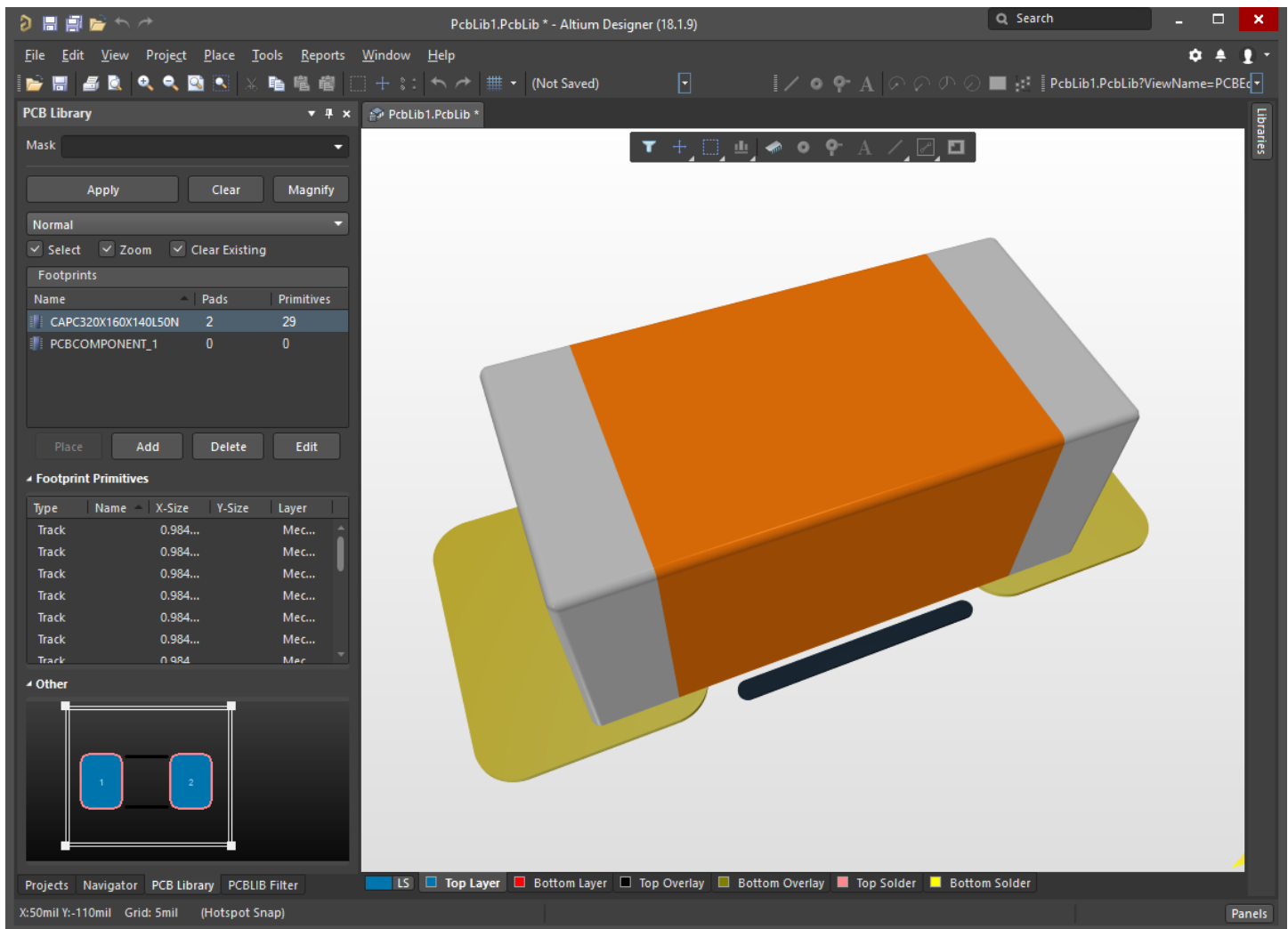
Browse to the folder that you set up in the Library Expert “Build Part” dialog window and Select the “.prjscr” file and then select the “Open” button



Select “CreateALibrary” and Double Click or select “OK”



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.