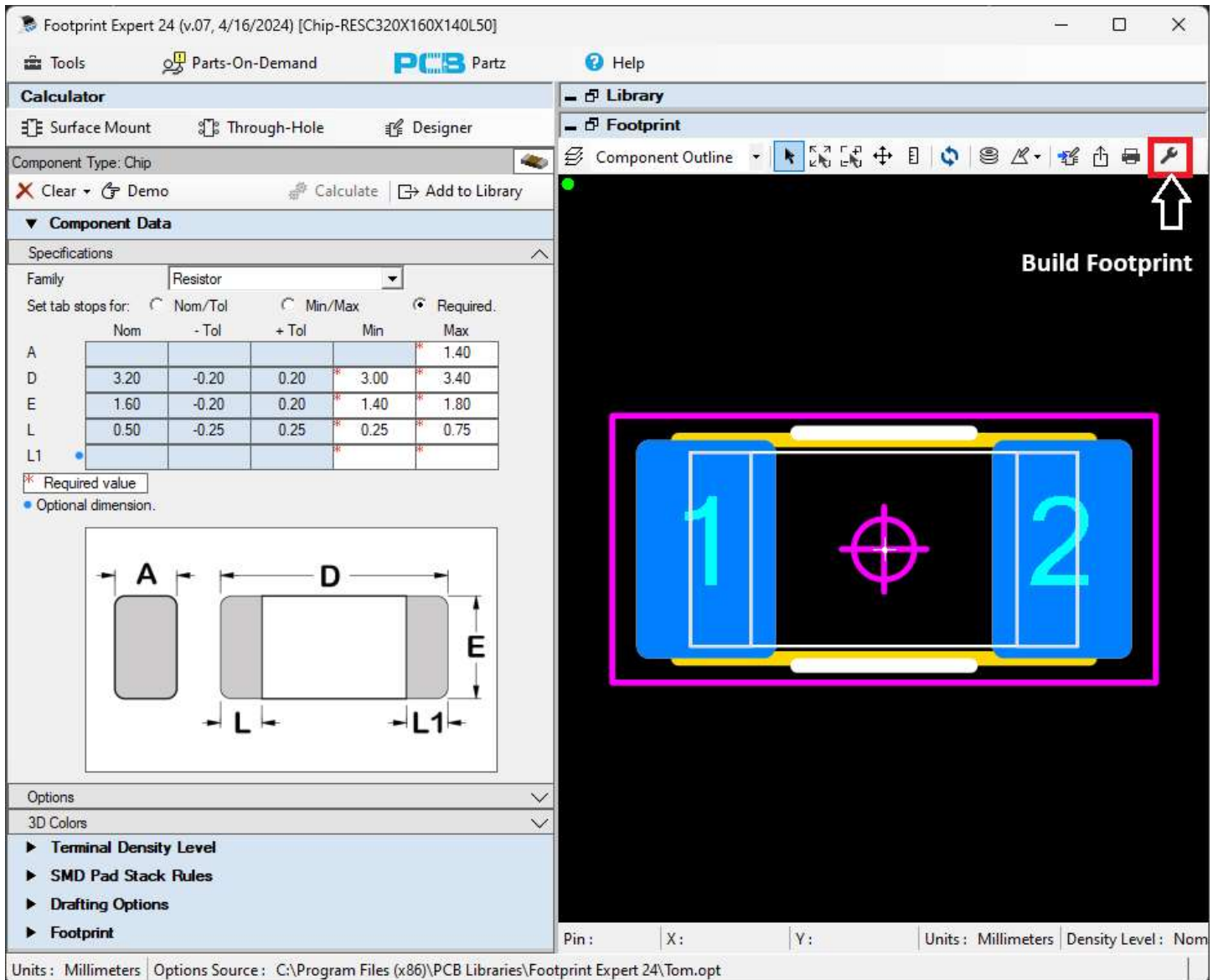


V2024 PCB Footprint Expert to Altium Designer 24 Via Script to add new Footprints to an Existing Library

To create a footprint, click the “**Build Footprint**” 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**”

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**”

PCB Libraries Footprint Expert to CAD Translator

Translator: Altium Designer ▾ 3D STEP ✓

Altium Designer © Altium Ltd Set as default ✓

Footprint Name: RESC320X160X140L50N

Library Folder Name: None (same as output directory) ▾

Option Settings

Option File (.opt): C:\Program Files (x86)\Altium_Designer_Translate_.opt

Output Directory: C:\Users\TH\Documents\CAD Tools\Altium Designer ...

Target Library: Use Existing ▾

☐ Go To Output Directory After Closing ☐ Use Post Process Script

☒ Use Altium Mask Expansion Rules ☐ Use Stroke Font

Layers	Top	Bottom
Electrical	eTopLayer	eBottomLayer
Solder Mask	eTopSolder	eBottomSolder
Paste Mask	eTopPaste	eBottomPaste
Silkscreen Outline	eTopOverlay	eBottomOverlay
Assembly Outline	eMechanical11	eMechanical14
Courtyard Outline	eMechanical15	
Component Outline	eMechanical12	
Terminal Outline	eMechanical12	
3D Model	eMechanical13	
Reference Designator	eMechanical16	

Save My Settings Restore My Settings Restore Default Settings

3D Options

Output Directory: Same as Translator Directory ▾

Version: AP214 ▾ TH Lead Extend: 2.00 ▾ MM ▾

File Extension: .STEP ▾ ☒ Include Suffix in Footprint Names



Vertical Axis: Z ▾

Quality: High ▾

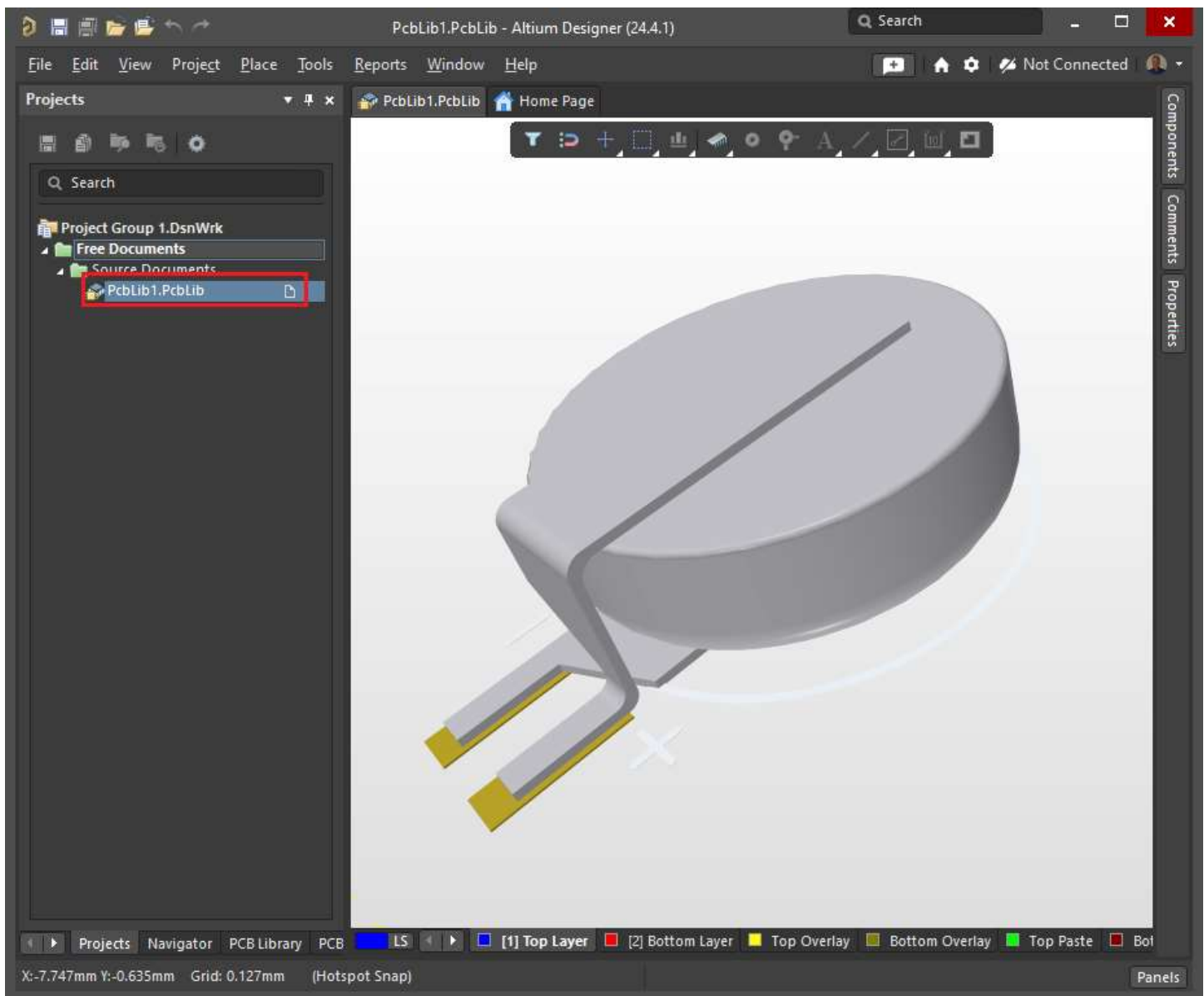
Material Condition: Nominal ▾

Status: Ready

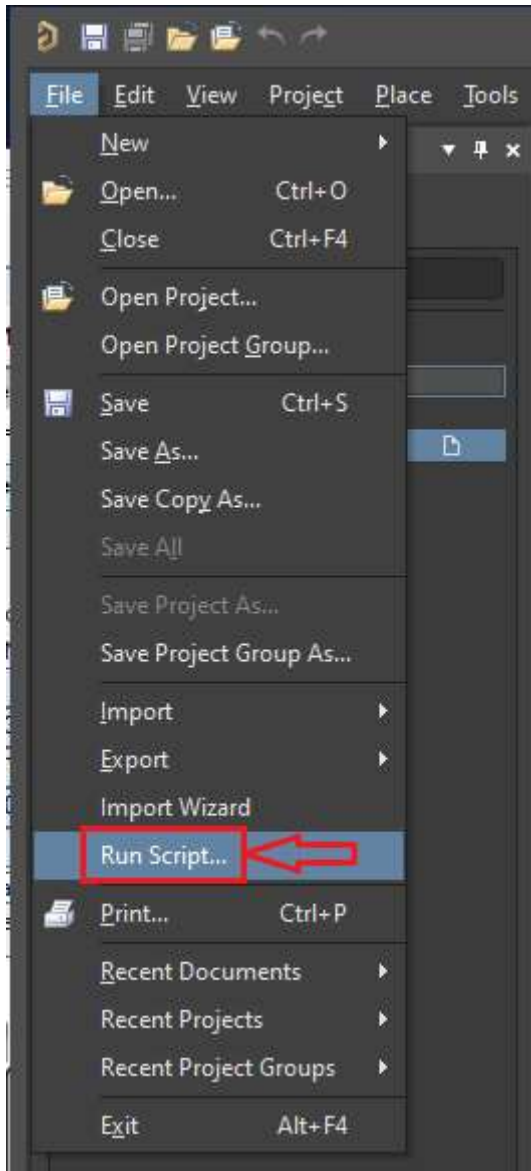
These 3 files will be created in the Output folder that you defined:

Name	Date modified	Type	Size
 RESC320X160X140L50.STEP	4/3/2021 1:55 PM	STEP File	169 KB
 RESC320X160X140L50N.pas	4/3/2021 1:55 PM	PAS File	12 KB
 RESC320X160X140L50N.prjscr	4/3/2021 1:55 PM	Altium Script Project	1 KB

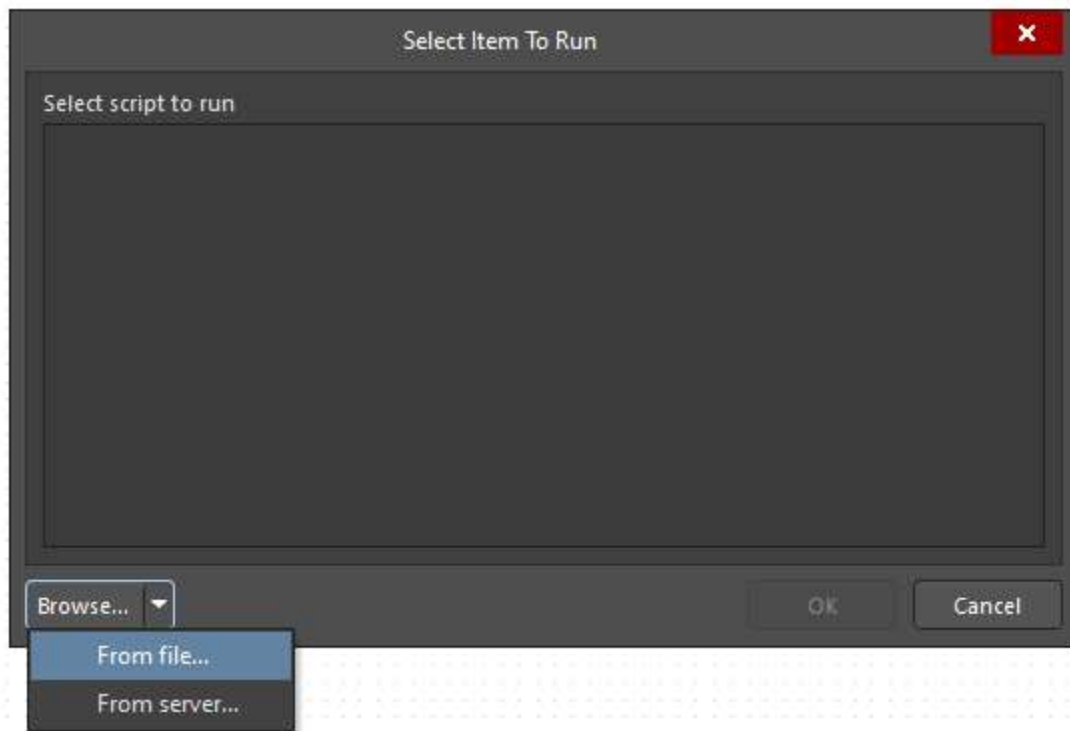
Open Altium and select a Library:



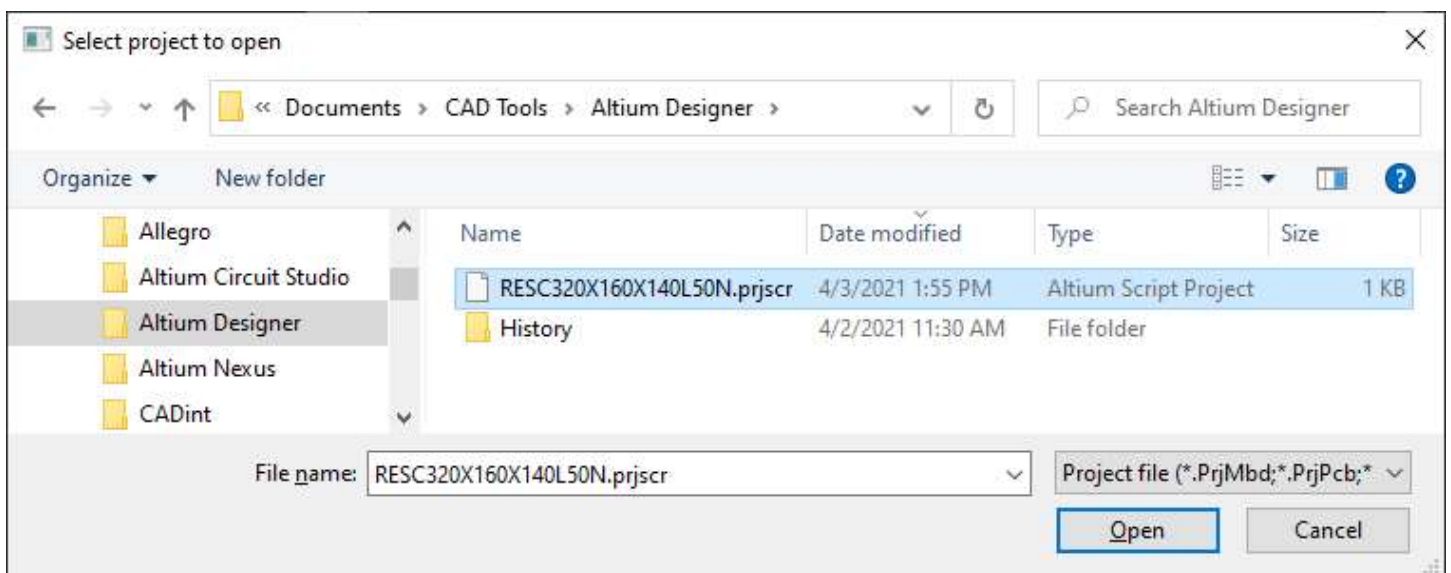
Select **“File > Run Script”**



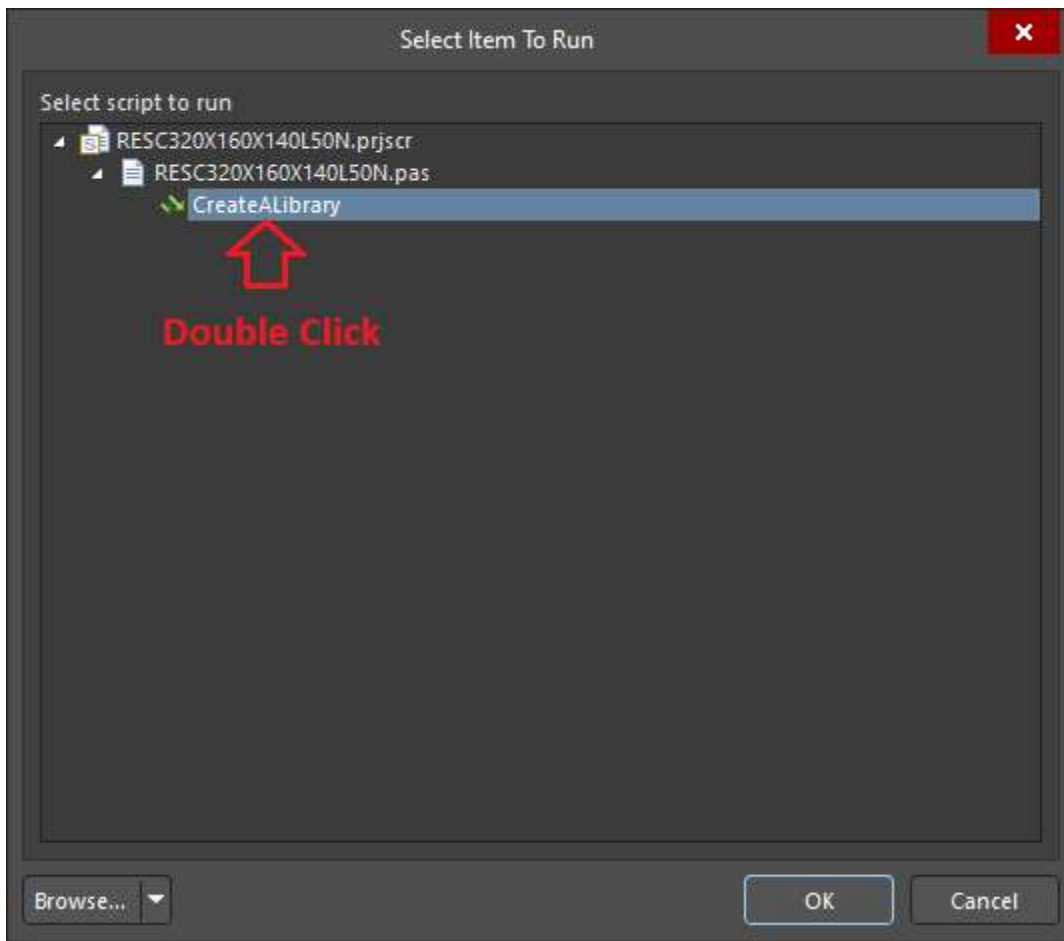
“**Select Run Script**” dialog window will appear. Select the “**Browse > From File**” button.



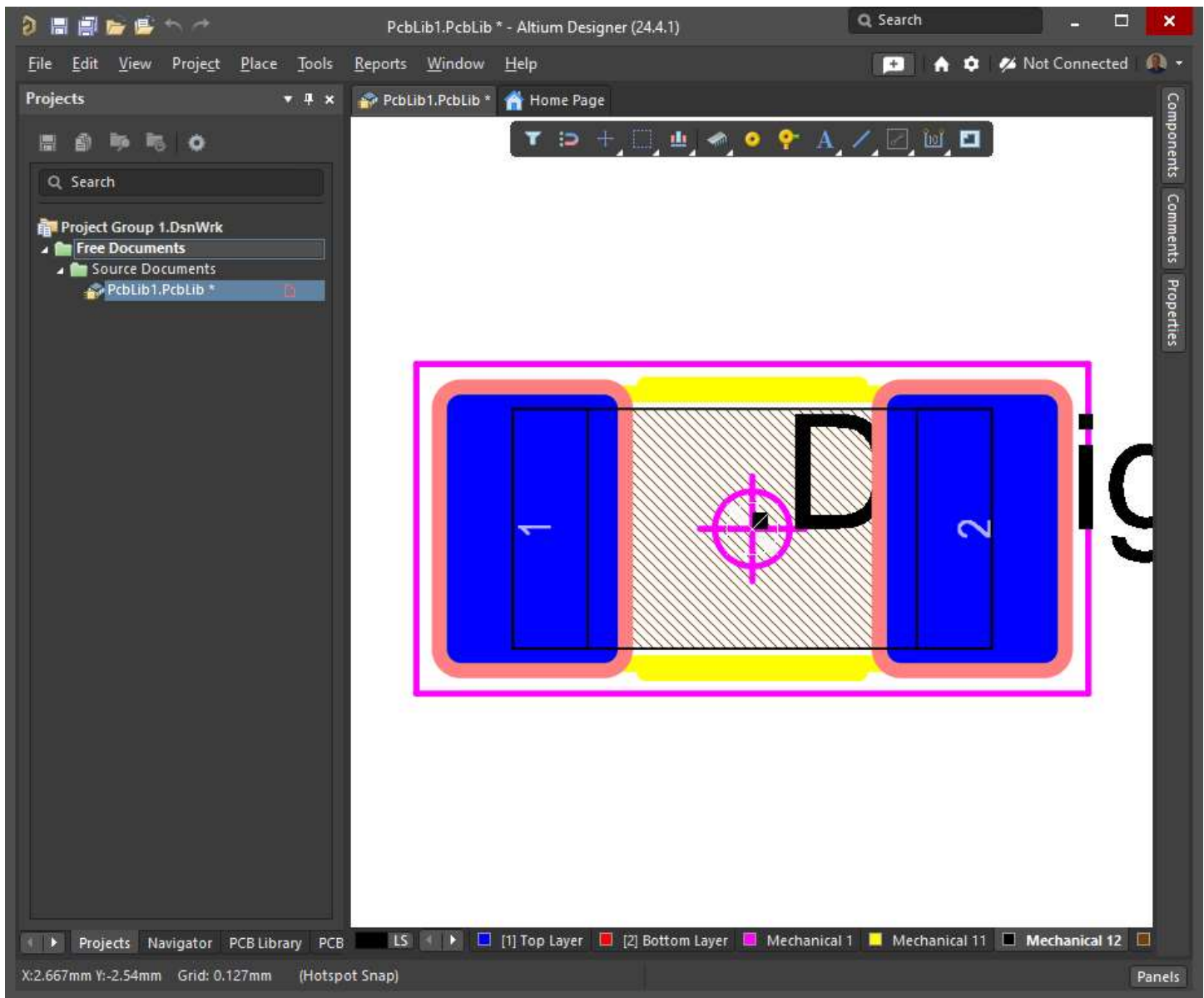
Browse to the folder that you set up in the Library Expert “**Build Part**” dialog window and Select the “**.prjscr**” file and then select the “**Open**” button.



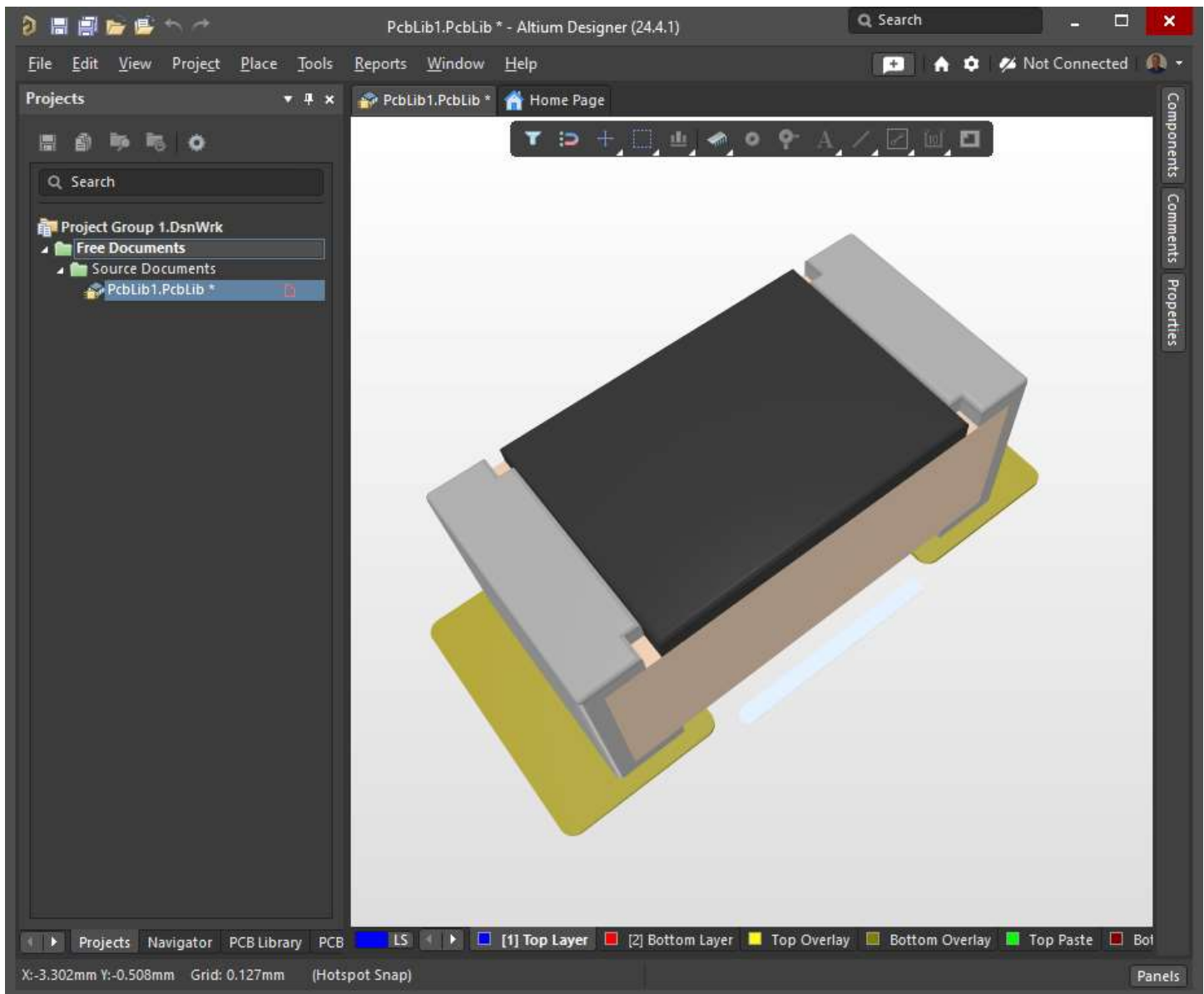
Select “**Browse > search for the .prjscr file**” and then “**CreateALibrary**” and Double Click or select “**OK**”



The new part will appear in the Library Editor. 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.