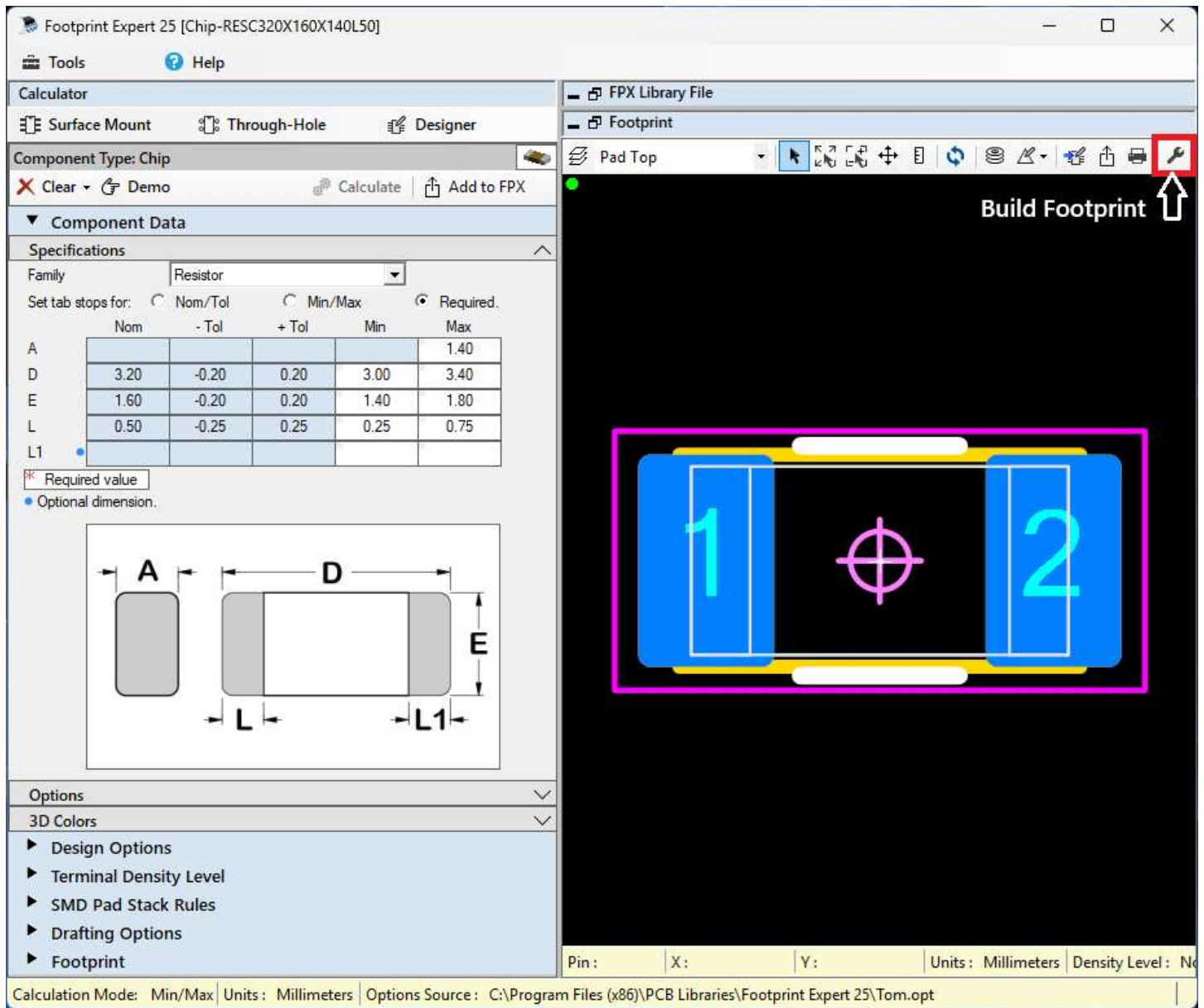


V25 PCB Footprint Expert to Altium Designer 25

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

Help

Altium Designer © RenesasSet as default

Footprint NameRESC320X160X140L50N

Library Folder NameNone (same as output directory)

Option Settings

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

Output DirectoryC:\Users\TH\Documents\CAD Tools\Altium Designer

Target LibraryUse Existing

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

☒ Use Altium Mask Expansion Rules☐ Use Stroke Font

☒ Enable All Layers

Layers	Top	Bottom
Electrical	eTopLayer	eBottomLayer
Solder Mask	eTopSolder	eBottomSolder
Paste Mask	eTopPaste	eBottomPaste
Silkscreen Outline	eTopOverlay	eBottomOverlay
Assembly Outline	eMechanical9	eMechanical10
Courtyard Outline	eMechanical15	
Origin Outline	eMechanical17	
Component Outline	eMechanical21	
Terminal Outline	eMechanical21	
3D Model	eMechanical13	
Reference Designator	eMechanical11	

Create a Folder

Only change layer numbers
Not Text

Save My SettingsRestore My SettingsRestore Default Settings

3D Options

Output DirectorySame as Translator Directory

VersionAP214

File Extension.STEP

Vertical AxisZ

QualityHigh

Material ConditionNominal

TH Lead Extend2.00MM

☐ Include Suffix in 3D Model Name

Status: Ready

Definition of the check box options:

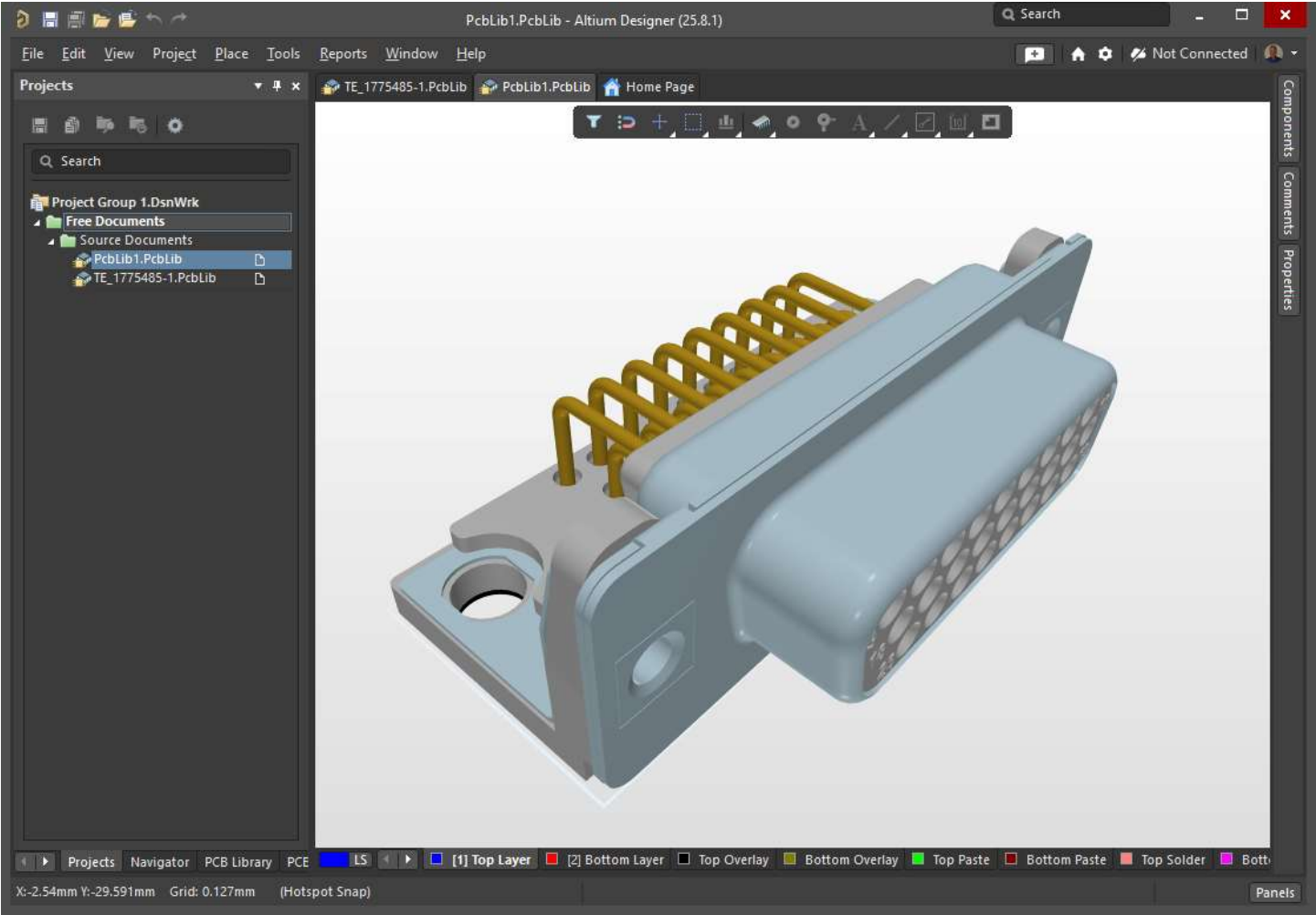
☐ Go To Output Directory After Closing	☐ Use Post Process Script
☐ Use Altium Mask Expansion Rules	☐ Use Stroke Font
☐ Use Altium Default Text Justification	☒ Enable Only Used Layers

1. Go To Output Directory After Closing
 - a. After selecting 'Save and Close' or 'Create and Close', the Windows folder that the script file was placed will open in Windows Explorer.
2. Use Altium Mask Expansion Rules
 - a. In the PCB Libraries Altium translator scripts there are script instructions for **SMFromRules** (Solder Mask From Rules - *SMFR*) and **PMFromRules** (Paste Mask From Rules - *PMFR*). These instructions apply only to surface mount pad stacks. When called, the instructions allow an Altium program rule to define the mask expansion.
 - b. In the Altium translator there are two possible ways the Altium script instructions *SMFR* or *PMFR* are implemented. With *Surface Mount* pad stacks, **FromRules** will be implemented: (1) automatically, if the *Footprint Expert (FPE)* pad mask expansion is zero; or (2) if the Altium Translator check box option for "Use Altium Mask Expansion Rules" is checked. Otherwise, **FromRules** will not be applied and the expansion will be as defined by the *FPE* pad stack.
 - c. The "Use Altium Mask Expansion Rules" check box option does not affect *Through-Hole (TH)* pad stacks and there is no **FromRules** equivalent in *TH*. With *TH* pad stacks, the *Altium Mask Expansion Rules* are only applied if the *Footprint Expert (FPE)* paste or solder mask pad expansion is equal to zero. (This is why *TH* mask expansions of zero cannot be achieved through scripting.) If not zero, the expansion will be as defined by the *FPE* pad stack. Otherwise, the expansion will be as defined by the *Altium Mask Expansion Rules*.
3. Use Altium Default Text Justification
 - a. Unchecked = Center Justification
 - b. Checked = Left Justification
4. Use Post Process Script
 - a. Need text
5. Use Stroke Font
 - a. When the selection box is checked the Altium text will be 'Stroke Font'. When the selection box is unchecked the Altium True Type Font will be used.
6. Enable Only Used Layers
 - a. Enable Used Layers, when enabled all layers with primitives (pads, tracks, lines and regions) will be displayed in Altium.

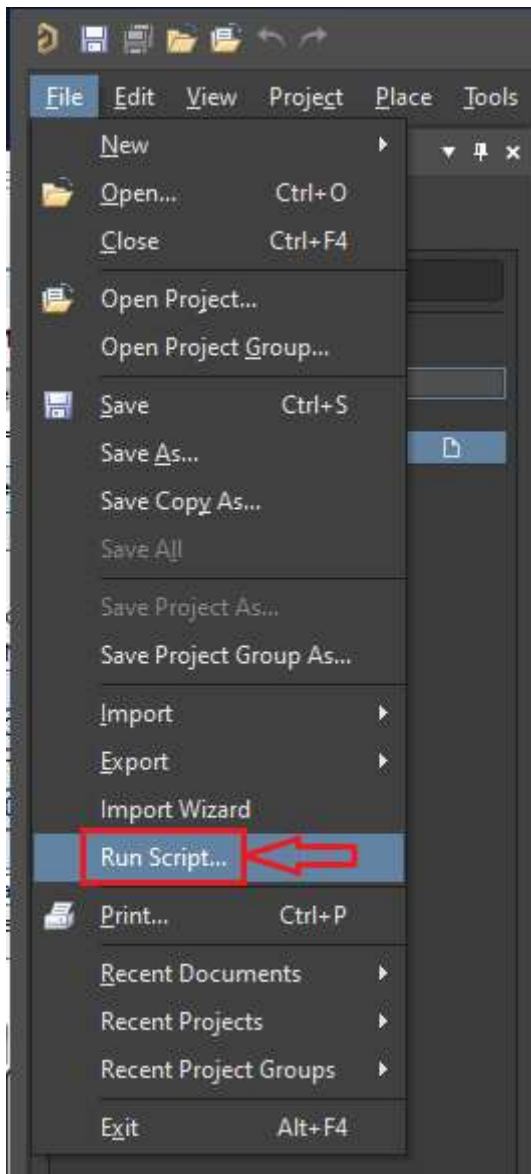
These 3 files will be created in the Output folder that you defined:

Name	Date modified	Type	Size
 RESC320X160X140L50.STEP	7/31/2025 9:05 AM	STEP File	169 KB
 RESC320X160X140L50N.pas	7/31/2025 9:05 AM	PAS File	17 KB
 RESC320X160X140L50N.prjscr	7/31/2025 9:05 AM	Altium Script Project	1 KB

Open Altium and select a Library:



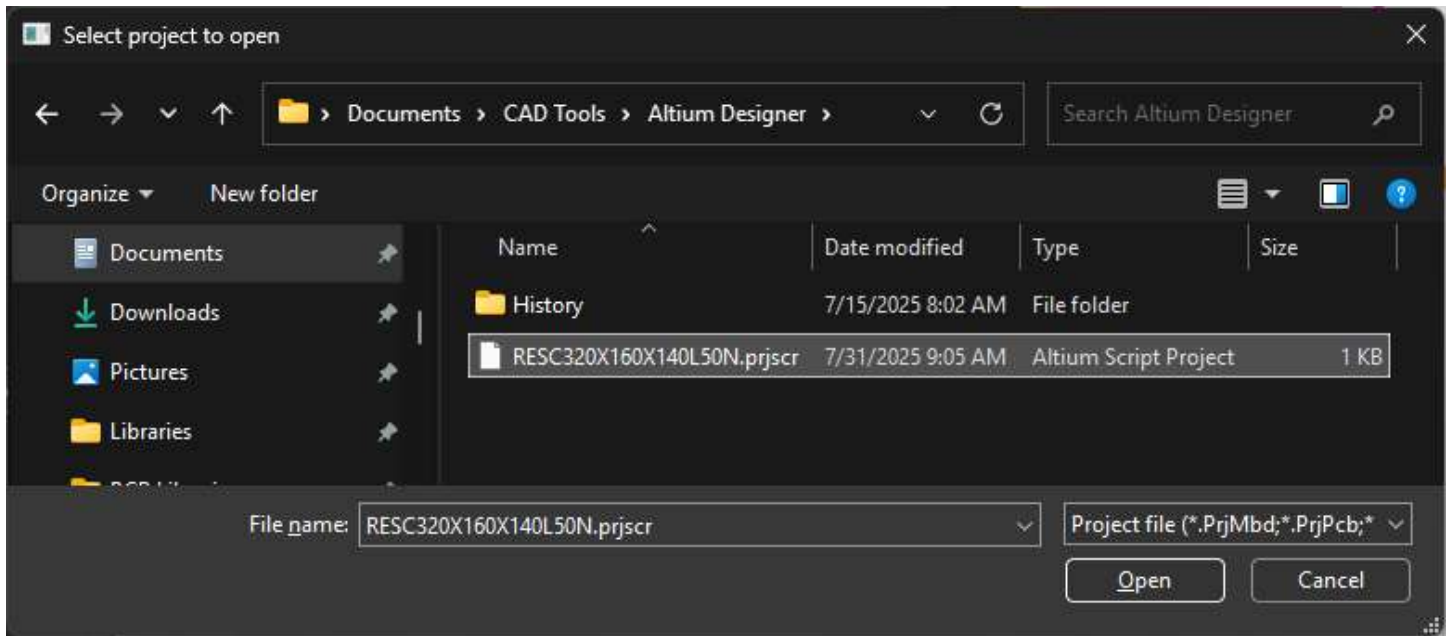
Select “**File > Run Script**”



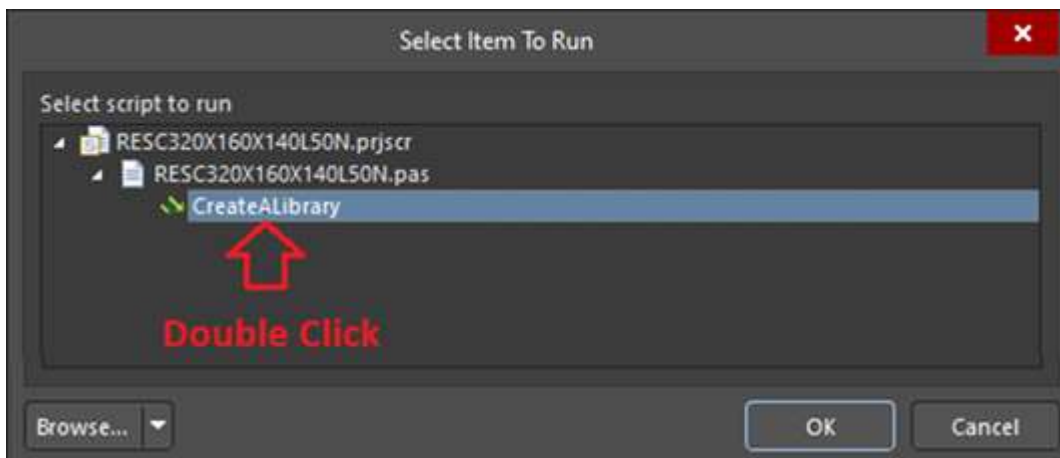
“**Select Run Script**” dialog window will appear. Select the “**Browse > From File**” button.



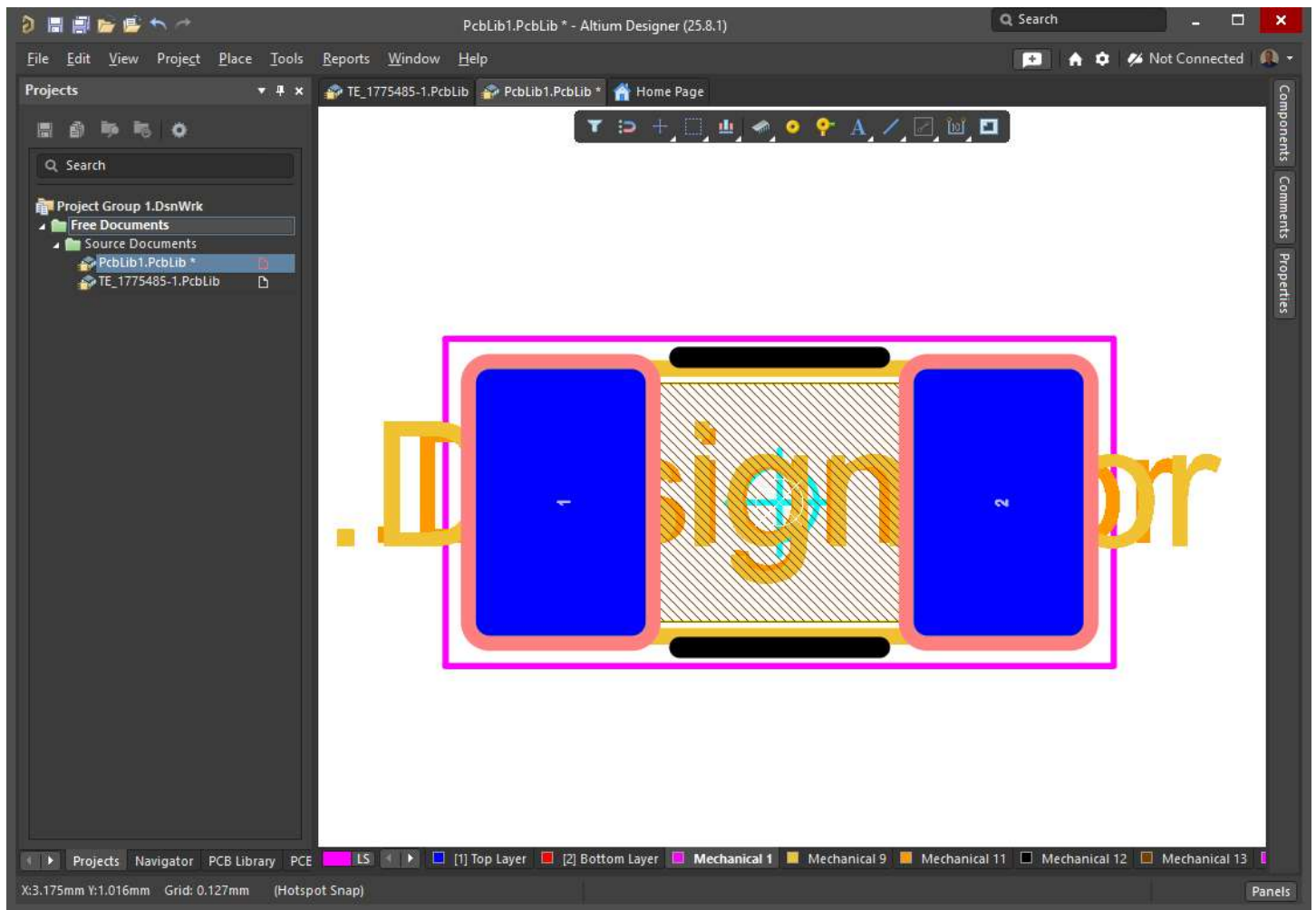
Browse to the folder that you set up in the Library Expert “**Build Part**” dialog window and select the “**.prjscr**” file and then select the “**Open**” button.



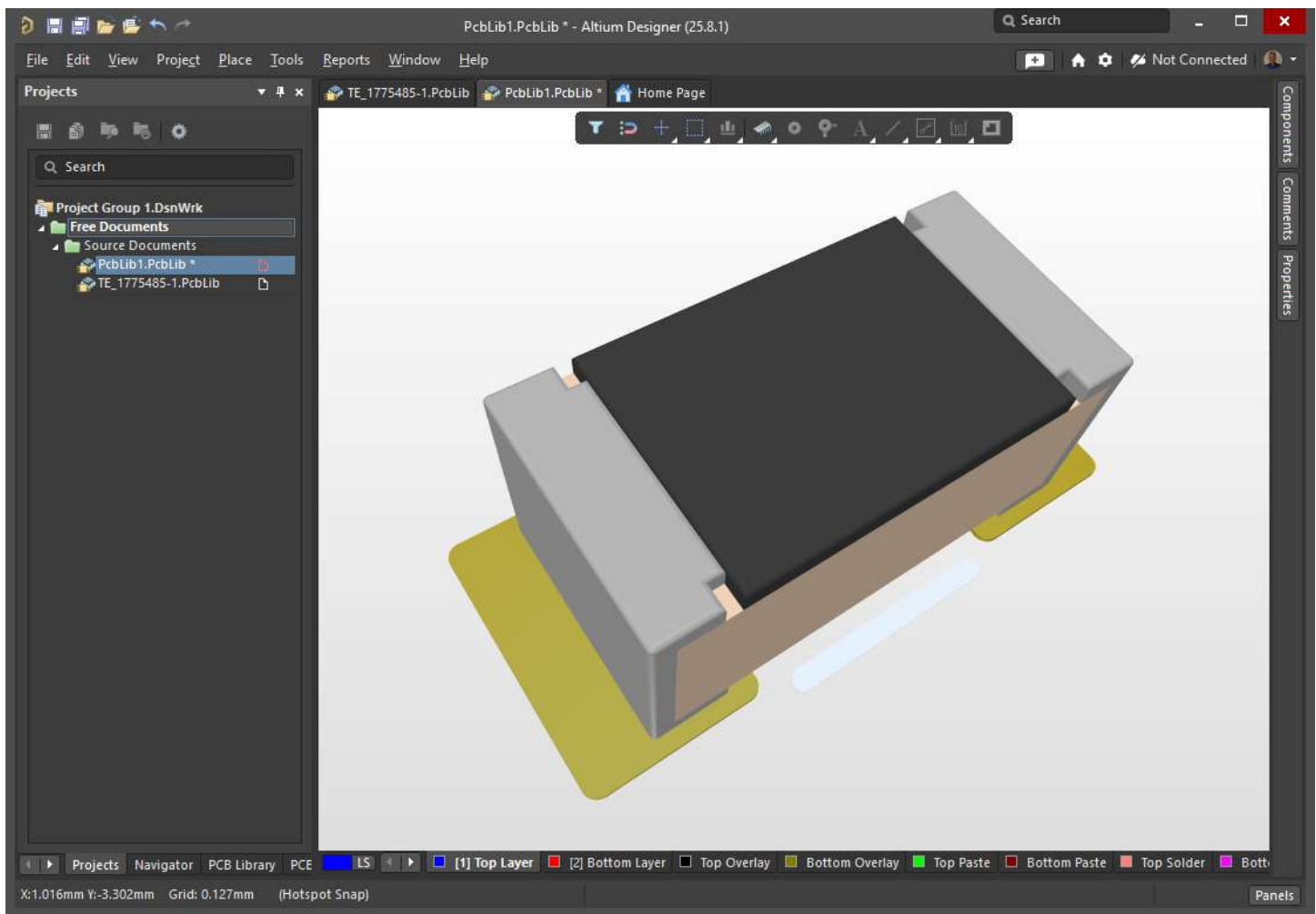
Select “**Browse > search for the .prjscr file**” and then “**CreateALibrary**” and Double Click or select “**OK**”



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.