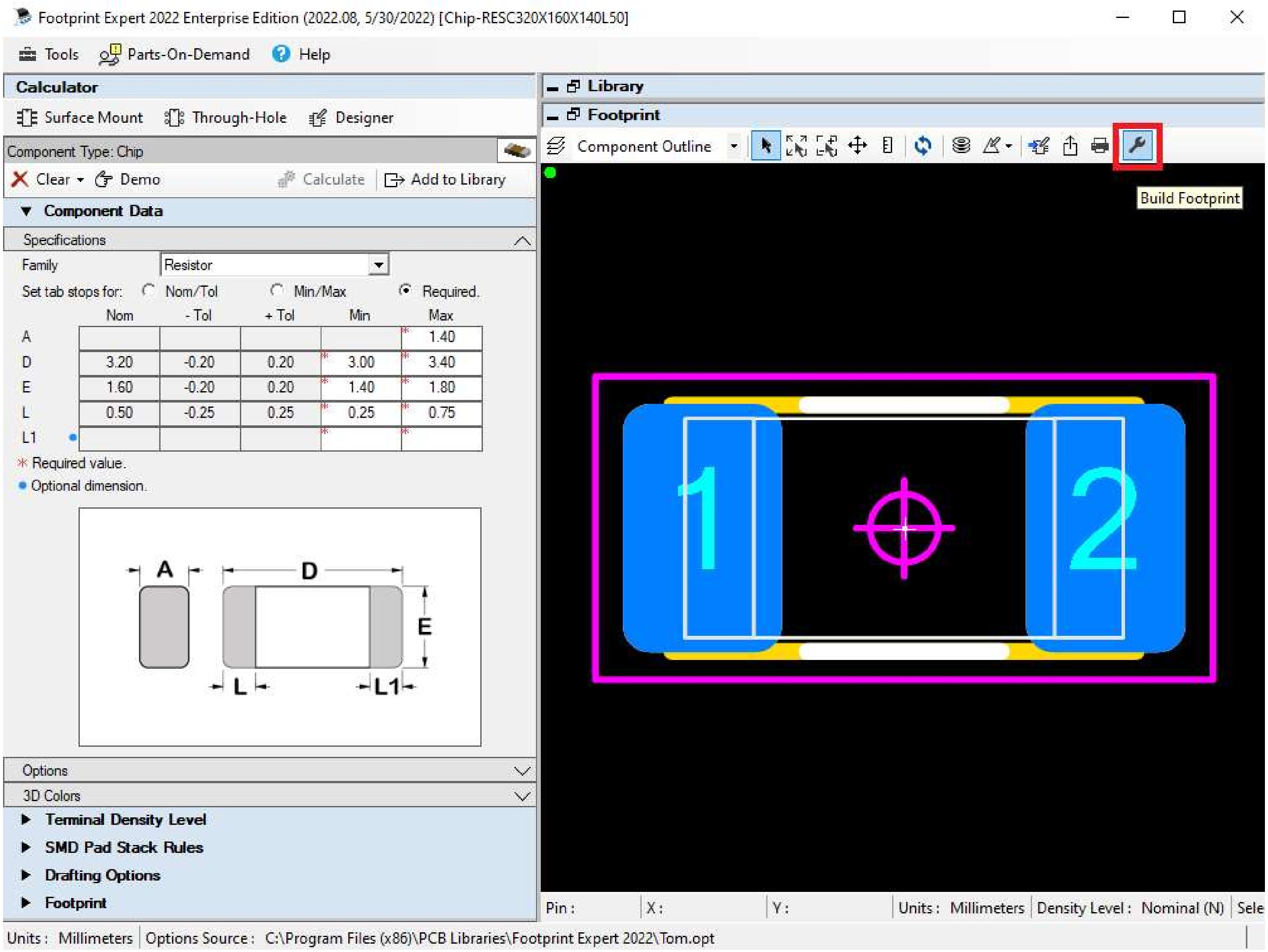
V2022 PCB Footprint Expert to Altium Designer 22 Via Script to add new Footprints to an Existing Library

To create a footprint, click the "**Build Footprint**" Wrench icon in the toolbar.



The CAD tool interface dialog box will open.

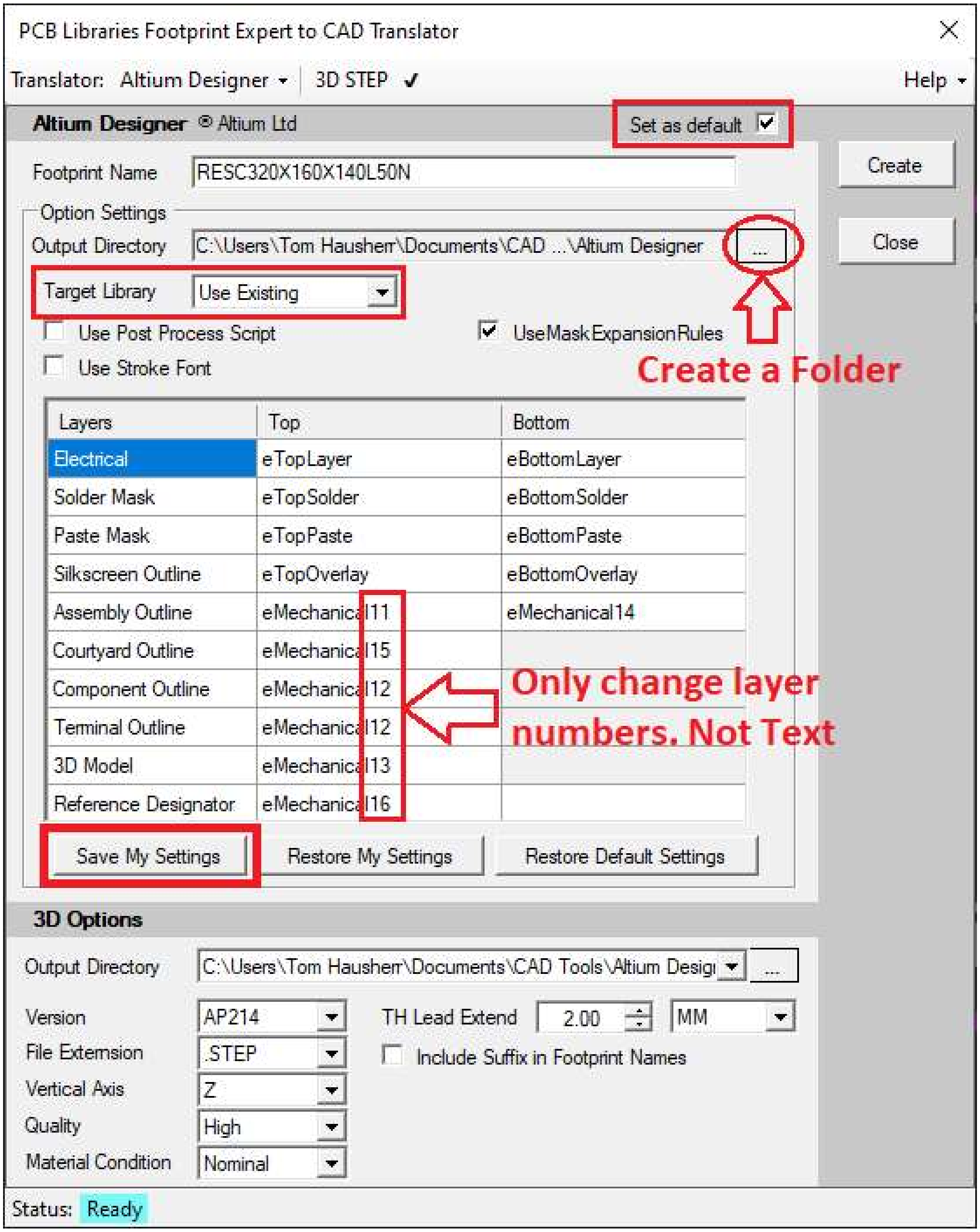
Select the Altium Translator and select the radio button "**Set as Default Format**"

Select the Output Directory folder for the Script files, select the Output Directory folder for the 3D STEP model

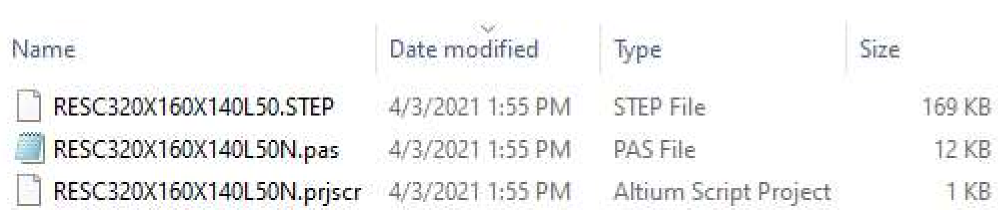
Select Vertical Axis "**Z**", select the "**Save Entries as Preferences**"

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**"

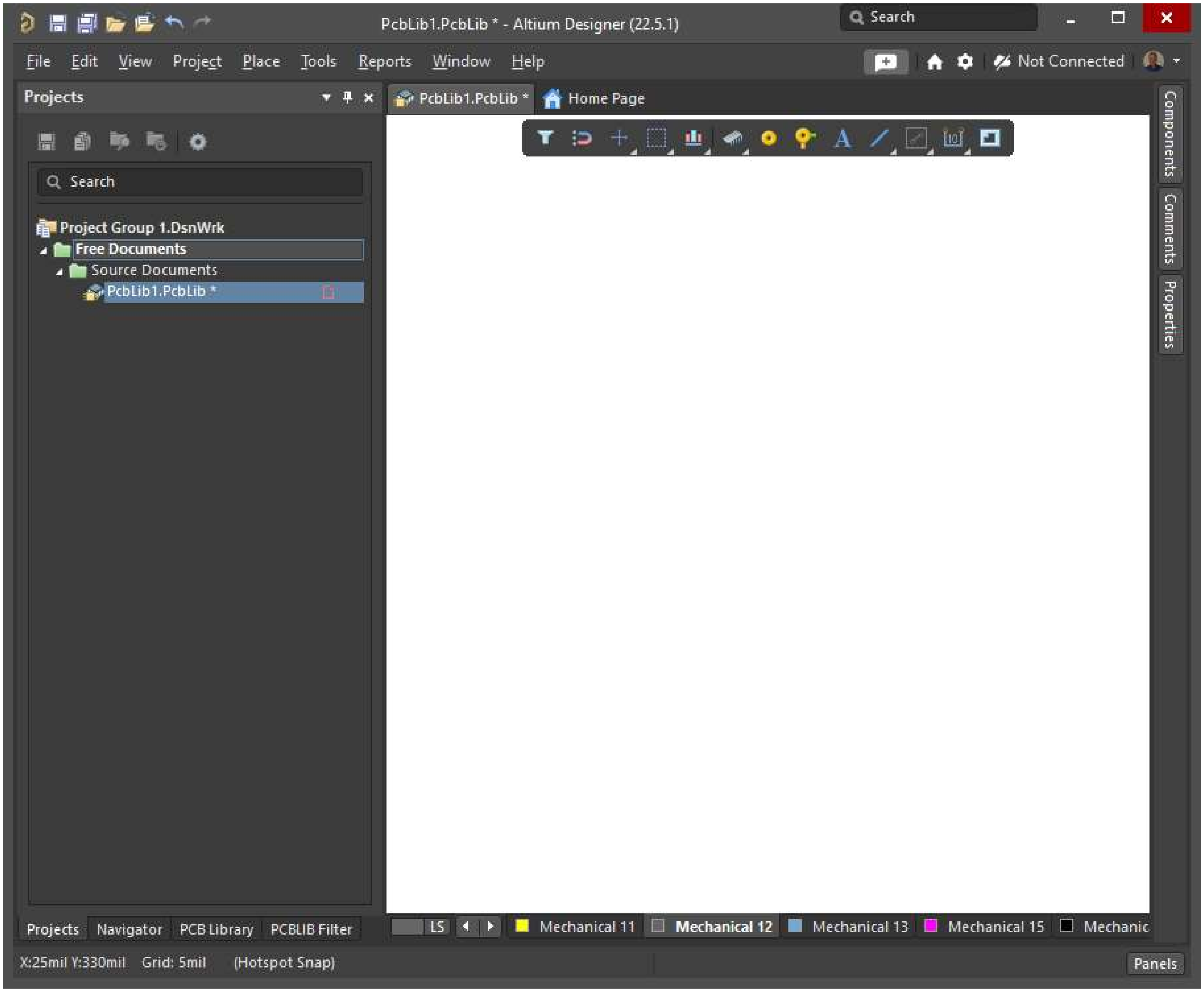


These 3 files will be created in the Output folder that you defined:



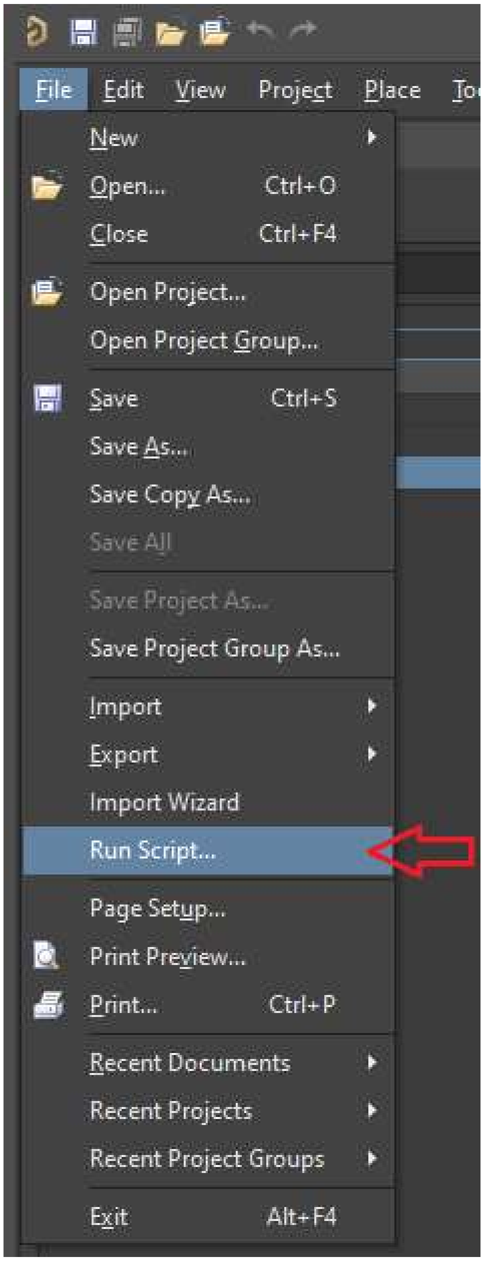
Name	Date modified	Type	Size
RESC320X160X140L50.STEP	4/3/2021 1:55 PM	STEP File	169 KB
RESC320X160X140L50N.pas	4/3/2021 1:55 PM	PAS File	12 KB
RESC320X160X140L50N.prjscr	4/3/2021 1:55 PM	Altium Script Project	1 KB

Open Altium:

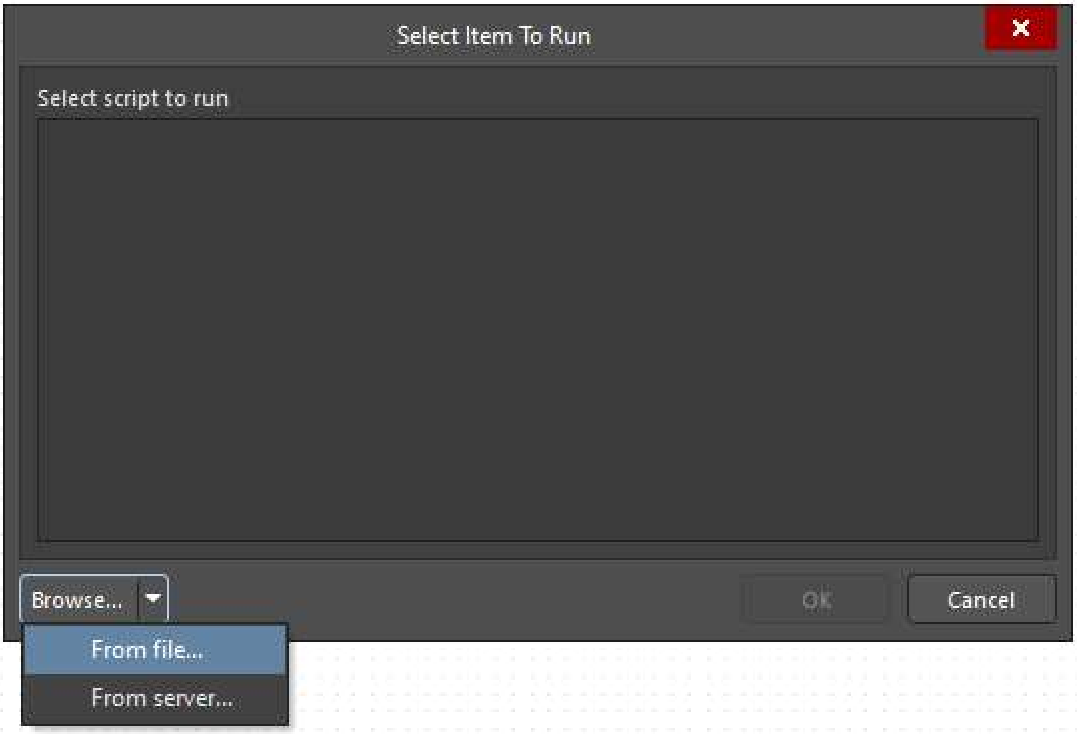


PcbLib1.PcbLib * - Altium Designer (22.5.1)
Search
File Edit View Project Place Tools Reports Window Help
Not Connected
Projects
Search
Project Group 1.DsnWrk
Free Documents
Source Documents
PcbLib1.PcbLib *
PcbLib1.PcbLib *
Home Page
Components
Comments
Properties
Projects Navigator PCB Library PCBLIB Filter
LS
Mechanical 11 Mechanical 12 Mechanical 13 Mechanical 15 Mechanic
X:25mil Y:330mil Grid: 5mil (Hotspot Snap)
Panels

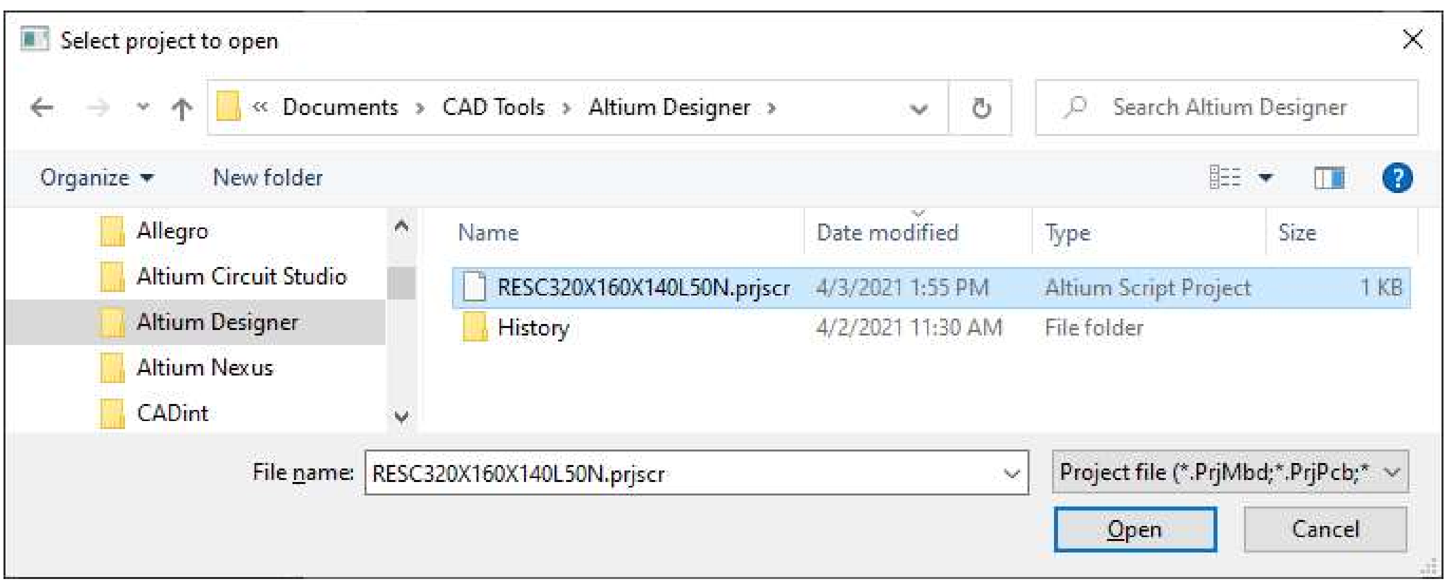
Select “**File > Run Script**”



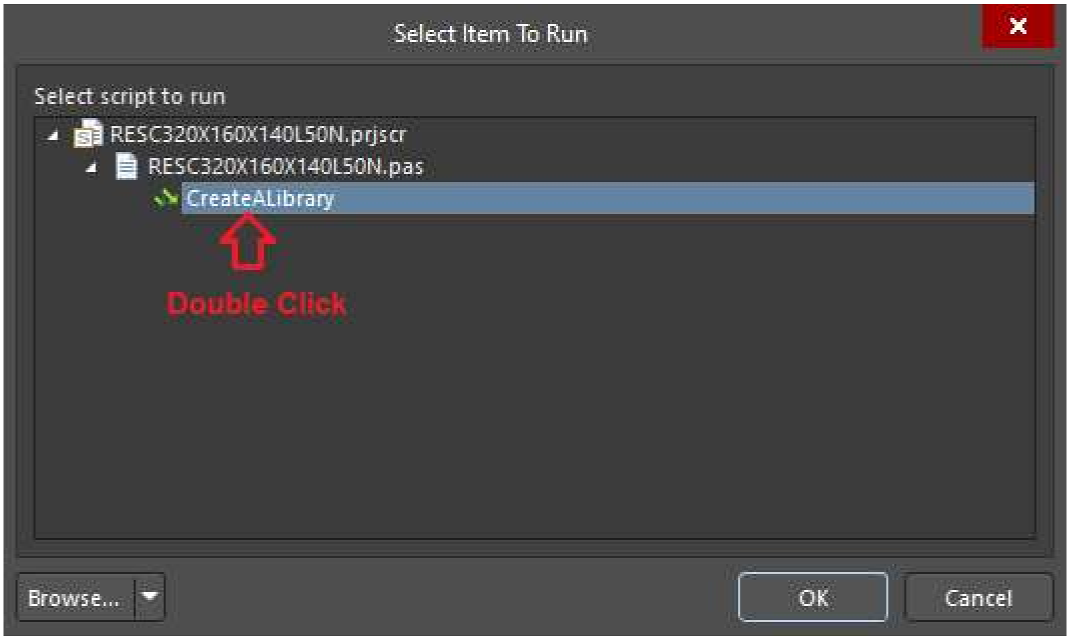
“**Select Run Script**” dialog window will appear. Select the “**Browse > From File**” button.



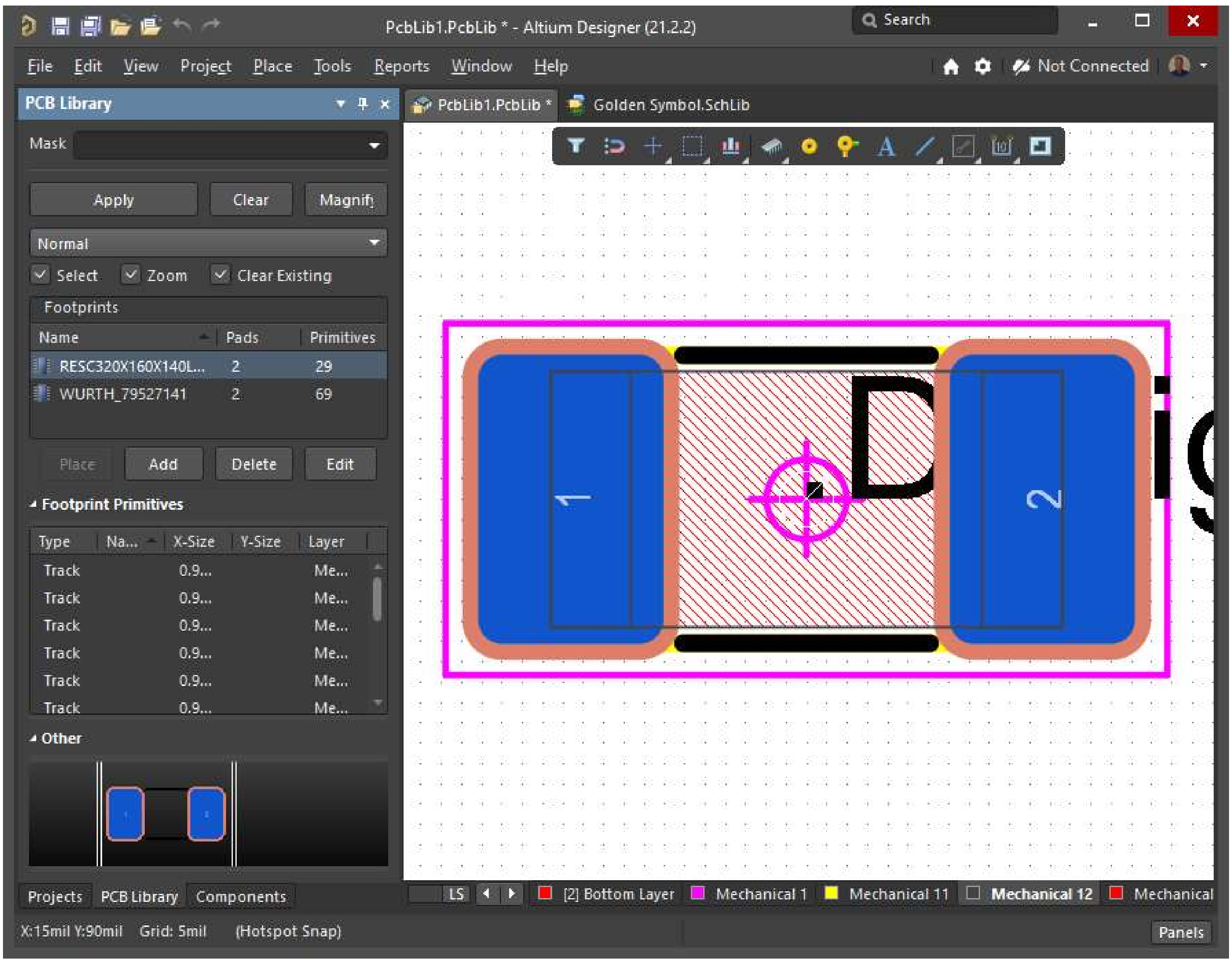
Browse to the folder that you set up in the Library Expert “**Build Part**” dialog window and Select the “**.prjscr**” file and then select the “**Open**” button.



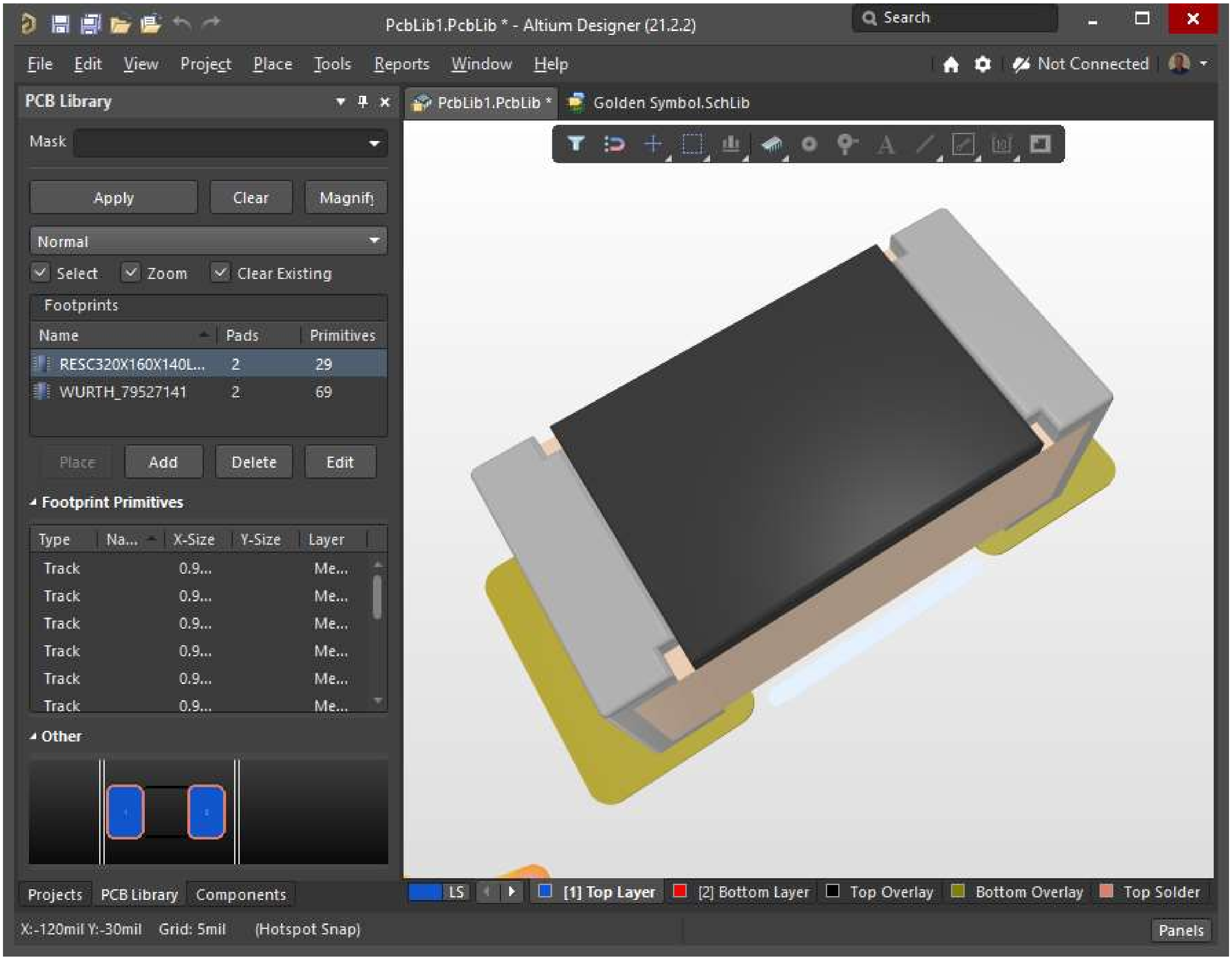
Select “**CreateALibrary**” and Double Click or select “**OK**”



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



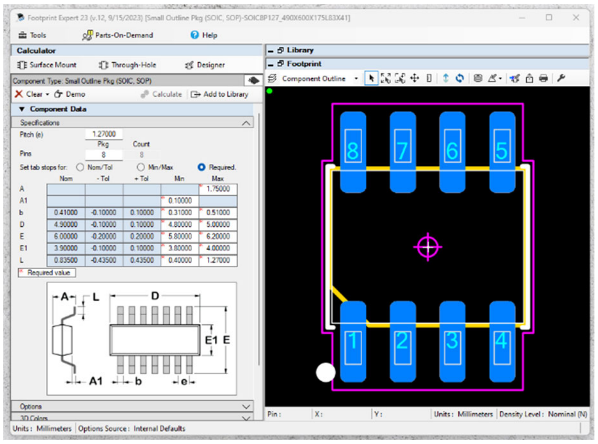
Use the “Shift + Middle Mouse Button” to rotate the 3D model

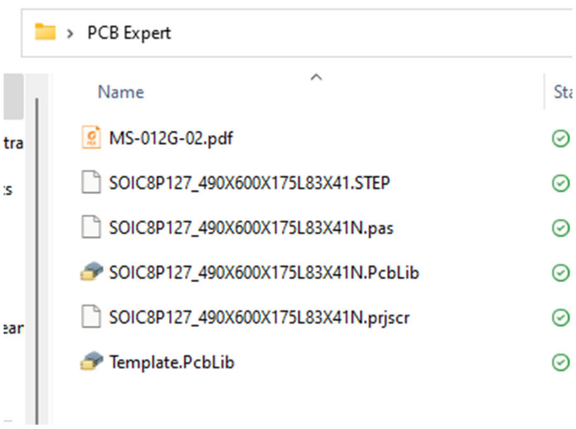
Right Mouse Button on the Library Part Name and select Copy

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

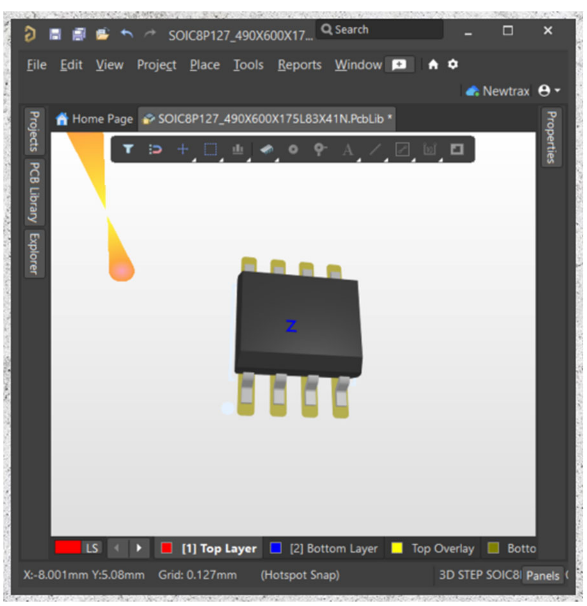
Use the Footprint Expert with Altium 365

Build or output your part from the PCB Footprint Expert.



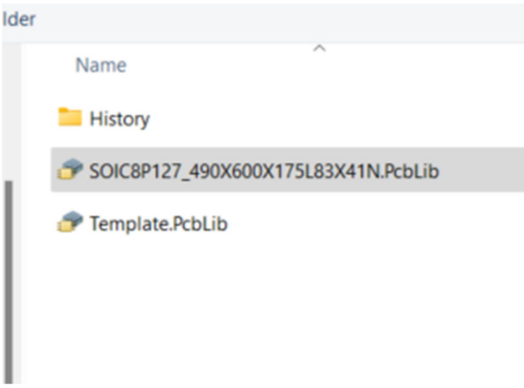


After creating the files for a SOIC-8 (JEDEC MO-012, AA variation). Import them into a PcbLib file from Altium. For more flexibility in the following steps, use only one footprint per file.

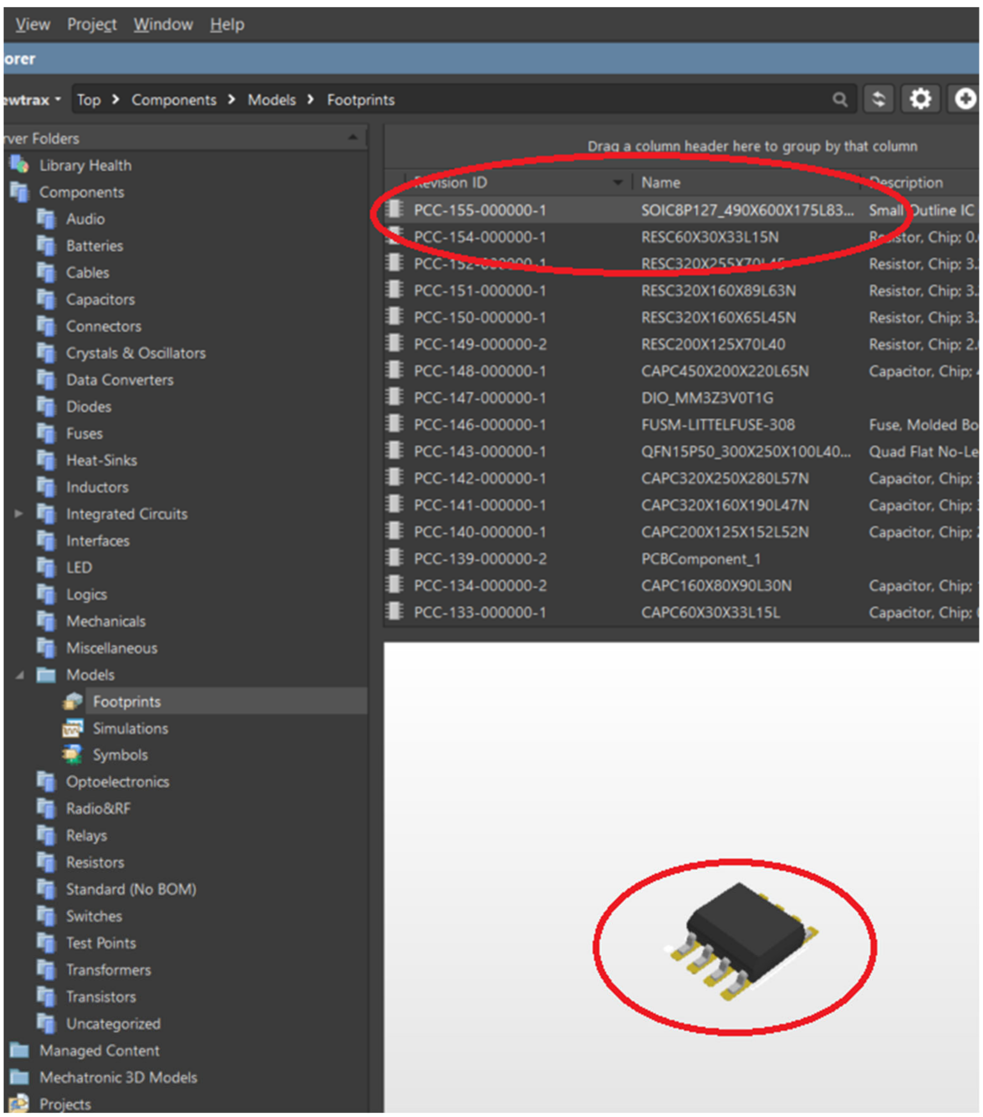


Import it into Altium 365.

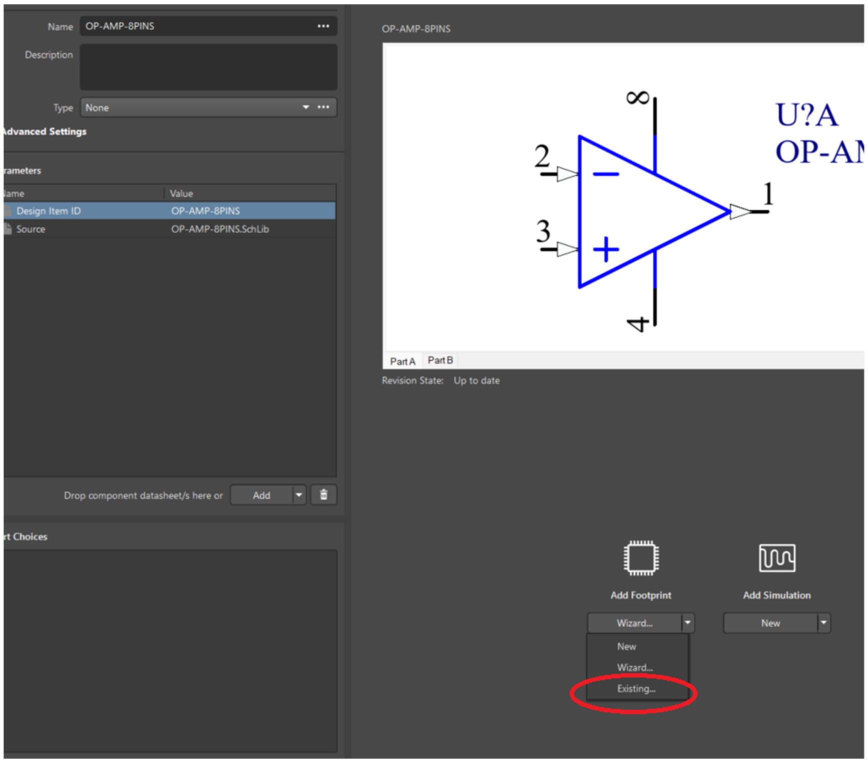
From the explore it, go to the "Models" folder and choose "Footprints" and "Import Library".



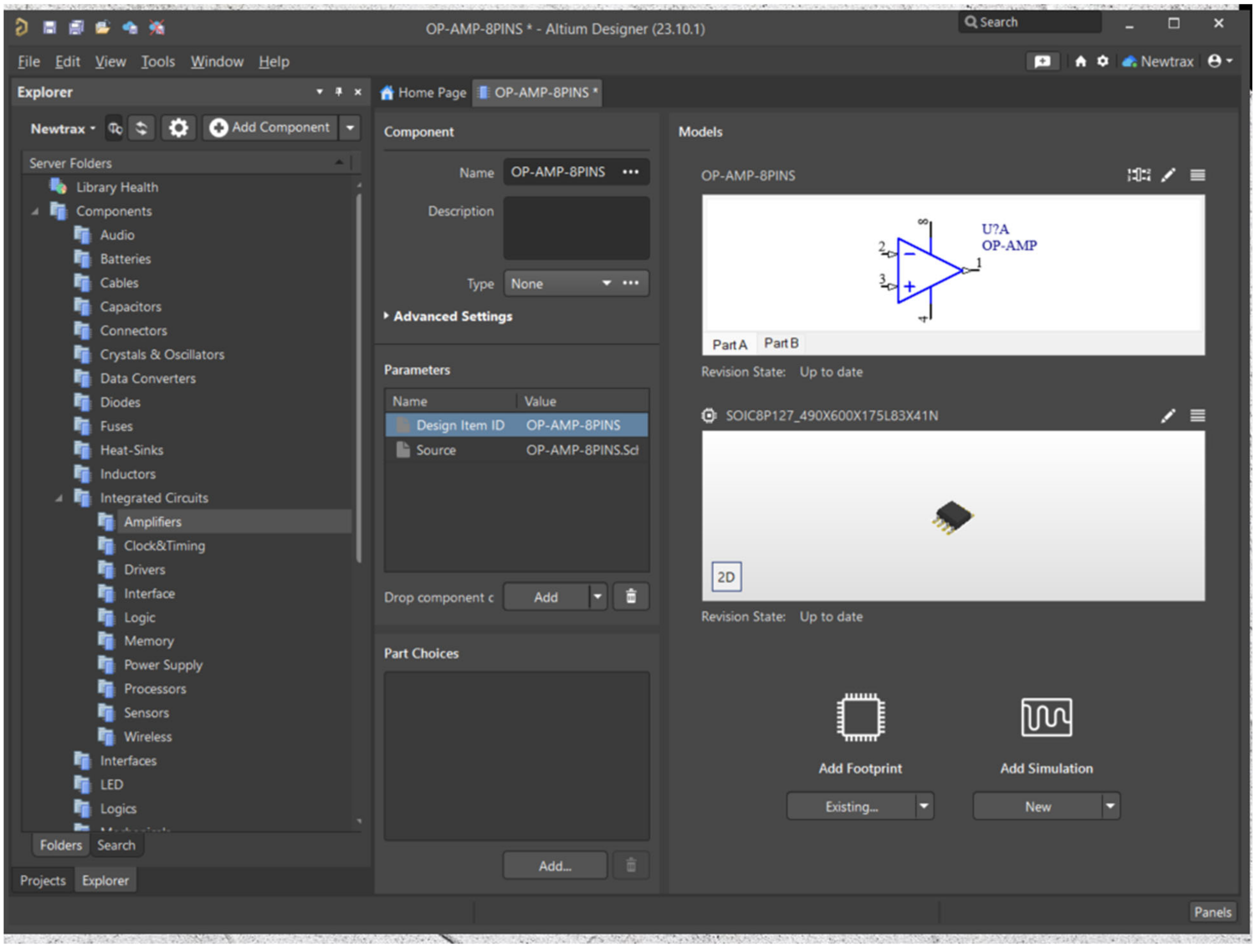
Your footprint is now available to be attached to a room in Altium 365.



From a part, to add a footprint to it, when we edit it, we add an existing footprint. This is the one we just imported.



And that's the magic of it.



OP-AMP-8PINS * - Altium Designer (23.10.1)
File Edit View Tools Window Help
Explorer
Newtrax
Add Component
Server Folders
Library Health
Components
Audio
Batteries
Cables
Capacitors
Connectors
Crystals & Oscillators
Data Converters
Diodes
Fuses
Heat-Sinks
Inductors
Integrated Circuits
Amplifiers
Clock&Timing
Drivers
Interface
Logic
Memory
Power Supply
Processors
Sensors
Wireless
Interfaces
LED
Logics
Folders Search
Projects Explorer
Home Page
OP-AMP-8PINS *
Component
Name OP-AMP-8PINS
Description
Type None
Advanced Settings
Parameters
Name Value
Design Item ID OP-AMP-8PINS
Source OP-AMP-8PINS.Scl
Drop component c Add
Part Choices
Add...
Models
OP-AMP-8PINS
U?A
OP-AMP
Part A Part B
Revision State: Up to date
SOIC8P127_490X600X175L83X41N
2D
Revision State: Up to date
Add Footprint
Add Simulation
Existing...
New
Panels