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.

Footprint Expert 2022 Enterprise Edition (2022.08, 5/30/2022) [Chip-RESC320X160X140L50] Х 🛱 Tools 🖉 Parts-On-Demand 🕝 Help - 🗗 Library Calculator - 🗗 Footprint 📳 Surface Mount 🕄 Through-Hole 💕 Designer 😰 Component Outline 🔹 💽 🐼 🕼 🕂 🛙 🔷 😂 🗷 🔹 🖆 🖶 4 Component Type: Chip 40 X Clear 🔹 🕝 Demo Calculate Add to Library **Build Footprint** ▼ Component Data Specifications Family Resistor • C Min/Max · Required Set tab stops for: C Nom/Tol Nom - Tol + Tol Min Max A 1.40 D 3.20 -0.20 0.20 3.00 3.40 Е 1.60 -0.20 0.20 1.40 1.80 0.50 -0.25 0.25 0.25 0.75 L 11 * Required value. Optional dimension. D Е -1--11-Options 3D Colors Terminal Density Level SMD Pad Stack Rules Drafting Options Footprint Y : Pin: X: Units : Millimeters Density Level : Nominal (N) Sele

Units : Millimeters Options Source : C:\Program Files (x86)\PCB Libraries\Footprint Expert 2022\Tom.opt

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 Foo	tprint E	pert to CAD Translator			×
Translator: Altium Designer 👻 3D STEP 🖌					Help 🕶
Altium Designer	r ® Altiu	im Ltd	Se	t as default 🔽	
Footprint Name RESC		320×160×140L50N			Create
Output Directory	C:\Use	ers\Tom Hausherr\Docume	nts\CAD\Altium De	signer ()	Close
Target Library	Target Library Use Existing				
Use Post Pro	ocess Sc Font	npt	I⊽ UseMaskExpans (ionRules 1 Create a Folde	er
Layers		Тор	Bottom		
Electrical		eTopLayer	eBottomLayer		
Solder Mask		eTopSolder	eBottomSolder		
Paste Mask		eTopPaste	eBottomPaste		
Silkscreen Outli	ne	eTopOverlay	eBottomOverlay		
Assembly Outlin	e	eMechanica[11]	eMechanical14		
Courtyard Outlin	ne	eMechanica <mark></mark> I15			
Component Out	line	eMechanica 112	Only cha	inge layer	
Terminal Outline	•	eMechanical12	number	s. Not Text	
3D Model		eMechanica[13			
Reference Desi	ignator	eMechanica <mark>l16</mark>			
Save My Set	tings	Restore My Settings	Restore Default	Settings	
3D Options					
Output Directory	C:\Us	ers\Tom Hausherr\Docum	ents\CAD Tools\Altiur	n Desigi 💌	
Version	AP21	4 ▼ TH Lead F	tend 200 -		
File Externsion	File Extension			1.1	
Vertical Axis Z					
Quality High -					
Material Condition	Nomir	nal 💌			
Status: Ready					

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

Name	Date modified	Туре	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:



Select "File > Run Script"



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

Select Item	n To Run	×
Select script to run		
Browse	Ģ	K. Cancel
Browse 💌	Ģ	K. Cancel

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 project to open						×
← → → ↑ 📙 « Docume	ents >	CAD Tools > Altium Designer >	×	õ	🔎 Search Altiur	n Designer
Organize 🔻 New folder					823	- 💷 🕜
Allegro	^	Name	Date modifie	d	Туре	Size
Altium Circuit Studio		RESC320X160X140L50N.prjscr	4/3/2021 1:55	ΡM	Altium Script Projec	t 1 KB
📙 Altium Designer		📙 History	4/2/2021 11:3	MA 0	File folder	
Altium Nexus						
CADint	~					
File <u>n</u> ame:	RESC	20X160X140L50N.prjscr		~	Project file (*.PrjN	lbd;*.PrjPcb;*
					<u>O</u> pen	Cancel
					-	

Select "CreateALibrary" and Double Click or select "OK"

Select Item To Run	×
Select script to run	
 RESC320X160X140L50N.prjscr RESC320X160X140L50N.pas 	
💊 CreateALibrary	
Double Click	
Browse 💌	Cancel

The new part will appear in the Library Editor. The library footprint will appear with the 3D Model automatically placed.

ð 🖩 🖷 🖬 🖆	PcbLib1.PcbLib * - Altium Designer (21.2.2)	Q Search	×
<u>F</u> ile <u>E</u> dit <u>V</u> iew Proje <u>c</u> t <u>P</u> lace <u>T</u> ools	<u>R</u> eports <u>W</u> indow <u>H</u> elp	A \$ \$	🖌 Not Connected 🚇 🝷
PCB Library 🔹 🕈	🗙 🚰 PcbLib1.PcbLib * 🚅 Golden Symbol.SchLib		
Mask	- T D +, D, U, *, • •	👌 A 🖊 🖂 🔟	
		ra ndadad	
Appiy Clear Magnif			
Normal			
🗹 Select 🗹 Zoom 🗹 Clear Existing			(1,1) = (1,1) + (1,1) + (1,1)
Footprints		· · · · · · · · · ·	
Name Pads Primitive	s of a state where we are a state of the sta	n n n nord a de	the first of the second s
RESC320X160X140L 2 29			
WURTH_79527141 2 69			
Place Add Delete Edit			
 Footprint Primitives 			\sim
Type Na X-Size Y-Size Layer			
Track 0.9 Me	to a second second second second second second	· · · · · · · · · · · ·	1. 1. 1. 1. 1. 1. 1. 1. 1. 1.
Track 0.9 Me			6 63 3 3 3 6 6 63
✓ Other			
1 1			
			10 10 11 12 11 10 10 10 10 10 10 10 10 10 10 10 10
		1 N N N N N N N N N	<u>0, 1997 (1, 0, 10, 10, 10, 10, 10, 10, 10, 10, 10</u>
Projects PCB Library Components	LS 🔹 🕨 📕 [2] Bottom Layer 📕 Mechanical 1 📕	Mechanical 11 🔲 Mec	hanical 12 📕 Mechanical
X:15mil Y:90mil Grid: 5mil (Hotspot Snap)			Panels

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

 Tools Parts-On-C Calculator
 Surface Mount
 Throw
 Surface Mount 🖉 Parts-On-Demand Hel: = 5º Libri ef Der Component Type: Small Clear + & Demo ate 🕞 Add to Library Spe Max 1 25000 Æ D 8888888 E1 E rce : Internal Defaul 🚞 > PCB Expert \sim Name Sta 😰 MS-012G-02.pdf \odot tra SOIC8P127_490X600X175L83X41.STEP \odot S SOIC8P127_490X600X175L83X41N.pas \odot SOIC8P127_490X600X175L83X41N.PcbLib \odot SOIC8P127_490X600X175L83X41N.prjscr \odot ear 🛷 Template.PcbLib \odot

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.

Name	OP-AMP-8PINS ····		
Description			
Туре		l ∞l	1 19 A
dvanced Setting	le		\mathbf{U}
		2	OP-A
rameters		⁴ → −	
lame	Value		1
Design Item ID	OP-AMP-8PINS		<u> </u>
Source	OP-AMP-BPINSSchub	PartA PartB Revision State: Up to date	
Dro	p component datasheet/s here or Add 💌 📋		
rt Choices			
			ហ្រៃ
		Add Footprint Add	d Simulation
		Wizard	New 🔻
		Neu	
		Monord	
		Wizaro	
		Existing	

And that's the magic of it.

