PCB Library Expert to Altium Designer 17 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.

Create Footprint			×
Footprint Name C	APC320X160X180L55N		
Translator	tium Designer 🛛 🗸 🗸	☑ Set as Default Format	Create
Output Directory C	:\Users\Tom Hausherr\Document	s\CAD Tools\Altium Design	Create and Close
Target Library	reate New		Cancel
Use Post Prod	se Existing 🗾 🗸 Use	e Mask Expansion Rules	3D Model
Layers		D ::	
	Iop	Bottom	
Electrical	e lopLayer	eBottomLayer	
Solder Mask	e lopSolder	eBottomSolder	
Paste Mask	e lopPaste	eBottomPaste	
Legend Outline	eTopOverlay	eBottomOverlay	
Assembly Outline	eMechanical11	eMechanical14	
Courtyard Outline	eMechanical15		
Component Outline	eMechanical12		
Terminal Outline	eMechanical12		Entries to
3D Model	eMechanical13		Defaults
Reference Designate	eMechanical16		
Altium Designer® is a tr	ademark of Altium Limited		Save Entries as Preferences
3D Model			
Version A	P214 ~	Quality High \vee	
Output Directory C	\Users\Tom Hausherr\Document	s\CAD Tools\Altium Designe	
TH Lead Extend 2	Use .STEP	Include Suffix in Names	
Vertical Axis Z	✓ Mater	rial Condition Nominal \checkmark	
Progress			User Guide

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

Documents library Altium Designer			A	Arrange by: Folder 🔻
Name	Date modified	Туре	Size	
CAPC2012X110.pas	10/18/2014 10:37 AM	PAS File	7 KB	
CAPC2012X110.prjscr	10/18/2014 10:37 AM	Altium Script Project	1 KB	
CAPC2012X110.STEP	10/18/2014 10:37 AM	STEP File	93 KB	

Open Altium:

DXP Eile View Project Window Help	DXP://Workspace	• • • • •
Projects • 🕈 X 🐴 Home		Favo
Workspace1.DsnWrk Vorkspace Start Workspace Documentation Vault Users Admin		rites
Project Workspace		Preferences
File View O Structure Editor		ard L
		braries
Files Projects Navigator		
	System Design Compiler Ho	elp Instruments >>

Select "DXP > Run Script"



"Select Item to run" dialog window will appear. Select the "Browse" button.

Select Item To Run	? ×
Select script to run	
Browse	OK 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

Choose Project to Open								×
Correction - Do	ocuments 🕨 My Docum	nents I	 CAD Tools Altium Desig 	Iner	▼ 4 ₇	Search Altium	Desig	ner 🔎
Organize 🔻 New folder							•	
My Documents Add-in Express		*	Documents librar	у	Arrange	by: Folder 🔻		
3D Models		=	Name	Date modified	Туре	Size		
Ja Allegro			CAPC2012X110.prjscr	10/18/2014 10:37 AM	Altium Script Project	1 KB		
Altium Designer								No preview
CrCAD PCB								available.
PADS								
🌗 POD Downloads								
Proteus								
Pulsonix								
📙 Quadcept		-						
File <u>n</u> ame:	CAPC2012X110.prjscr				•	Project file (*.P Open	rjPcb; *	*.PrjFpg; * 💌 Cancel

Select Item To Run	?
Select script to run CAPC2012X110.prjscr CAPC2012X110.pas CreateALibrary CreateComponentCAPC2012X110	
Browse	OK Cancel

Select "**PCB Library**" at the lower left of the project window to display the library components

PCB Library			•	ч ×
Mask				•
	Clear	Magnify		
Normal	Select N	<u>Z</u> oom <u>V</u>	<u>C</u> lear Existing	9
Components				- 1
Name	△ Pac	is	Primitives	-11
) 2 ENT 1 0		20	
PCBCOMPONI			0	
Component Prim	itives			^ ^
Type Name 🛆	X-Size	Y-Size	Layer	
Track	4.724mil		Mechanical	
Track	4.724mil		Mechanical	
Track	4.724mil		TopOverlay	=
Pad 1	4.724mii 45.276mil	57 874 mil	Top Laver	
	43.2701111	57.6741111	- ·	Ψ.
				^
	1	2		
	_			
Files Projects	Navigator	PCB Library	PCBLIB Filt	ter
X:-48110mil Y:-3985	mil Grid: 9	ōmil (Ho	tspot Snap)	

Check Attributes: Double Click on the library component Name

D <u>X</u> P <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> ep
i 🗋 💕 🛃 🖉	3 🛕 🗶 🖡	🗟 🗖 🍳	. 🔍 🖸		X R
PCB Library				• # ×	4
Mask				•	
Apply	🗸 Clear 🔊	Magnify			
Normal 🔻	<mark>▼ S</mark> elect ▼	<u>Z</u> oom 🔽 🤇	lear Exis	ting	
Components					
Name	∠ Pads		Primitive	s	
CAPC2012X1	10 2		20		
PCBCOMPO	NENI_1 0		0		

Notice that the Description and Height are automatically populated. Select the "**OK**" button.

💐 PCB Library C	omponent [mil]	? ×
Library Comp	oonent Parameters	
Name	CAPC2012X110	Height 1.10mm
Description	Capacitor, Chip;2.00 mm L X 1.2	25 mm W X 1.10 mm H body
		OK Cancel

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

PCB Library Mask Apply Clear Magnify Normal Select Zoom Clear Existing Components Name Pads Primitives
PCB Library Home Mask Apply Clear Magnify Normal Select Zoom Clear Existing Components Pads Primitives Image: CAPC2012X Pads Primitives Image: PCBCOMPC New Blank Component Component Wizard
Mask Apply Clear Magnify Normal Select Components Name Pads Primitives CAPC2012X*** New Blank Component Component Component Vizard
✓ Apply ✓ Clear ✓ Magnify Normal ✓ Select ✓ Zoom ✓ Clear Existing Components ✓ Pads Primitives ✓ CAPC2012X**** ✓ New Blank Component ✓ PCBCOMPC Component Wizard
Normal Image: Components Name A Pads Primitives Image: CAPC2012X110 New Blank Component New Blank Component Image: PCBCOMPC Component Wizard
Components Name Pads Primitives CAPC2012X110 New Blank Component Component Component Wizard
Name A Pads Primitives Image: CAPC2012X1101 New Blank Component New Blank Component
CAPC2012X110 New Blank Component Component Component Wizard
Component Wizard
Component wizard
Cut
Сору
Copy Name
Paste
Delete
Select All
Component Properties
Place
Update PCB With All
Report
Delete All Grids And Guides in Library

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