

Generating a Hierarchical Part



OrCAD Capture, PSpice Application Note | V2



Table of Contents

1	Introduction	3
2	Creating a Circuit to Generate a Part	3
3	Generating a Part	4
4	Edit the New Part	5
5	Using Hierarchical Part in PSpice Project	6
6	Bibliography	7



1 Introduction

Based on a schematic view you can generate a library and an associated part which represents your design. When you use the **Generate Part** command to do this, Capture creates a library file (.OLB) and a part based on the pins defined in the schematic. The library file (.OLB) containing the new part and a copy of the schematic for easy portability and design reuse. If the library already exists, the new part is appended to the existing library. If the part already exists in the library, the new part replaces it.

You can use the symbol you generate to represent your circuit for other PSpice projects.

2 Creating a Circuit to Generate a Part

As an example, we would like to build up a circuit consisting of three diodes which we can use as a rectifier for three-phase electric power.



You must use the hierarchical port to define the pins of a part which you want to generate. The ports are placed on nodes in the lower level schematic that will be pins on the upper level symbol. If the schematic is active, you can access the hierarchical port through **Place > Hierarchical Port...**

Place Hierarchical Port		
Symbol: PORTLEFT-R PORTBOTH-L/CAPSYM PORTBOTH-R/CAPSYM PORTLEFT-L/CAPSYM PORTLEFT-R/CAPSYM PORTNO-L/CAPSYM CAPSYM Design Cache SOURCE	PORTLEFT-R	OK Cancel Add Library Remove Library Help
NetGroup Port		

A Place Hierarchical Port window comes up:

In the **Place Hierarchical Port** window you can chose the hierarchical port which you want to use in your schematic.



3 Generating a Part

Let's assume you have opened the example PSpice project **create_hierarchy_part.opj** and saved the circuit which consists of three diodes in PAGE1 in the schematic folder my_hierarchy_part.

TT CREATE_HI
🗀 File 🖏 Hierarchy
🖃 🖳 Design Resources
.\create_hierarchy_part.dsn
u in the second
📄 👘 my_hierarchy_part
PAGE1
🖬 🖬 💼 Design Cache
Library
📄 📩 Outputs
.\create_hierarchy_part-pspicefiles\root\root.net
.\create_hierarchy_part.olb
Referenced Projects
📗 🗄 🗝 PSpice Resources

In Project Manager window, select the schematic folder my_hierarchy_part, then go to **Tools > Generate Part...** The **Generate Part** dialog window comes up.

Generate Part			
Netlist/source file:			
art\Create_Hierarchy_Part\create_hi	erarchy_part.dsn	Browse	
Netfiet (eeuwee Gie tune)		Drimitium	Cancel
Redist/source me type.		 No 	Help
Lapture Schematic/Design	•	O Yes	
Part name:		🔘 Default	FPGA Setup
my_hierarchy_part			
		Copy scher	matic to library
Destination part library:			
Application_Notes\Create_Hierarchy	_Part\create_hiera	archy_part.olb	Browse
Oreate new part ()	DUpdate pins on	existing part in li	ibrary.
Pick symbols manually			
- Sort pins	Additional pins		
Ascending order	Specify the i	number of addit	ional
Descending order	Number of	pins: 0	
		·	
🔲 Retain alpha-numeric pin-num	bers. Device is pin	grid array type (package.
Implementation type:	Source Schematic	name:	
Schematic View 👻	my_hierarchy_part		•
Implementation file:			
D:\PSpice\Application_Notes\0	LD_APP_UPDATE	EVZIPsVI Bro	owse

- In the field **Netlist/source file**, the current design create_hierarchy_part.dsn is automatically selected.
- Using Capture Schematic/Design as **Netlist/source file type**.
- In the field **Part name** you can give a name for the part which you want to create.
- As default a library named create_hierarchy_part.olb will be saved under the current path.
- The my_hierarchy_part is also automatically selected in the field **Source Schematic name**.

You can just leave other default settings in **Generate Part** window and click **OK**. The **Split Part Section Input Spreadsheet** window comes up as follows:

	Number	Name	Туре	Pin Visibility	Shape	PinGroup	Position	Section	
1	Í	OUT	Output		Line		Right	1	1
2]	U	Output		Line		Right	1	1
3		V	Output		Line		Right	1]
4		W	Output		Line		Right	1	1

Click Save to save the part.

Note

You can edit the part in this **Split Part Section Input Spreadsheet** window. You can also edit it later in **Part Editor.**

4 Edit the New Part

After saving the part you can find a part named my_hierarchy_part located in library create_hierarchy_part.olb. A copy of the schematic for easy portability and design reuse is also saved in this library.

 File Hierarchy Design Resources Create_hierarchy_part.dsn my_hierarchy_part mg PAGE1 Design Cache Library Outputs Acreate_hierarchy_part.olb Acreate_hierarchy_part Mg PAGE1 Acreate_hierarchy_part.olb Acreate_hierarchy_part Mg PAGE1 Design Cache Library Acreate_hierarchy_part-pspicefiles\root\root.net Acreate_hierarchy_part.olb The my_hierarchy_part Mg PAGE1 Design Cache Design Cache Referenced Projects PSpice Resources 	T CREATE_HI
 File Hierarchy Design Resources Create_hierarchy_part.dsn my_hierarchy_part PAGE1 Design Cache Library Acreate_hierarchy_part-pspicefiles\root\root.net Acreate_hierarchy_part.olb Acreate_hierarchy_part My_hierarchy_part Acreate_hierarchy_part Acreate_hierarchy_part.olb Interaction my_hierarchy_part Acreate_hierarchy_part Acreate_hierarchy_	
 Design Resources .create_hierarchy_part.dsn .create_hierarchy_part .my_hierarchy_part .create_hierarchy_part .create_hierarchy_part-pspicefiles\root\root.net .create_hierarchy_part.olb .create_hierarchy_part .create_hierarchy_part .create_hierarchy_part .create_hierarchy_part .create_hierarchy_part.olb .create_hierarchy_part .create_hierarchy	🛅 File 특, Hierarchy
 .create_hierarchy_part.dsn root my_hierarchy_part PAGE1 Library Outputs .create_hierarchy_part-pspicefiles\root\root.net .create_hierarchy_part.olb .create_hierarchy_part .create_hierarchy	🖃 🖷 Design Resources
 root my_hierarchy_part PAGE1 Design Cache Library Outputs .\create_hierarchy_part-pspicefiles\root\root.net .\create_hierarchy_part.olb .\create_hierarchy_part A PAGE1 A PAGE1 C PSpice Resources 	🚊 📲 .\create_hierarchy_part.dsn
 my_hierarchy_part PAGE1 Library Outputs \create_hierarchy_part-pspicefiles\root\root.net \create_hierarchy_part.olb \create_hierarchy_part my_hierarchy_part my_hierarchy_part my_hierarchy_part my_hierarchy_part Referenced Projects PSpice Resources 	i root
PAGE1 Design Cache Library Outputs .create_hierarchy_part-pspicefiles\root\root.net .create_hierarchy_part.olb .create_hierarchy_part .create_hierarchy_part .create_hierarchy_part .create_hierarchy_part .create_hierarchy_part .create_hierarchy_part .create_hie	🖶 🗁 my_hierarchy_part
 Design Cache Library Outputs \create_hierarchy_part-pspicefiles\root\root.net \create_hierarchy_part.olb \create_hierarchy_part \mathcal{P}_my_hierarchy_part PAGE1 Library Cache Referenced Projects PSpice Resources 	PAGE1
 Library Outputs .\create_hierarchy_part-pspicefiles\root\root.net .\create_hierarchy_part.olb PAGE1 Referenced Projects PSpice Resources 	🗄 🖷 🛄 Design Cache
 Outputs .\create_hierarchy_part-pspicefiles\root\root.net .\create_hierarchy_part.olb my_hierarchy_part my_hierarchy_part PAGE1 Library Cache PSpice Resources 	Library
.\create_hierarchy_part-pspicefiles\root\root.net	🖕 📩 Outputs
 .\create_hierarchy_part.olb my_hierarchy_part my_hierarchy_part my_hierarchy_part model PAGE1 model Library Cache Referenced Projects model PSpice Resources 	.\create_hierarchy_part-pspicefiles\root\root.net
→ my_hierarchy_part → my_hierarchy_part → my_hierarchy_part → mg PAGE1 → mg Library Cache → mg Referenced Projects → mg PSpice Resources	.\create_hierarchy_part.olb
PAGE1 ⊕ ☐ Library Cache Referenced Projects ⊕ PSpice Resources	📄 🖷 my_hierarchy_part
Eferenced Projects Popice Resources	PAGE1
Referenced Projects	🗄 🖮 🧰 Library Cache
PSpice Resources	Referenced Projects
	📗 🗄 🗠 🛅 PSpice Resources

FlowCAD



Select the part > Edit > Part. The Part Editor opens.

8	CR	ΕA	TE	J	IIE	RA	RC	:H	Υ_	P/	٩R	т.(ЭL	В -	m	y_	hie	era	arc				×
								-												-			^
							L	<u>ج</u> ر															
															C	UT	┣			_			
																U	-			-			
																V				_			
																w							
							<	/al		>													
									-														
																							~
<	ш																					>	

In Part Editor you can change the graphical symbol and modify the pin definition.

(CR	EA	TE	_}	11E	RA	RCH	Y_	PA	RT	.0	LB	- п	 .	. [×
1						-		-	-	-	-				1	1		^
							_U?											
							U											
							V			OL	JT	-						
							w											
							<val< td=""><td>ue></td><td>></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></val<>	ue>	>									
																		~
<																	>	

Save the part after editing it. It is ready for using in your PSpice project.

5 Using Hierarchical Part in PSpice Project

Place the part my_hierarchy_part from the library create_hierarchy_part.olb in root schematic and build a test circuit as follows:



Note

You can select the hierarchical part U1 and click the right mouse button to pop up a command menu. Using the command Descend Hierarchy to descend this hierarchical part.



Run the simulation for example, for 60ms. The three inputs V(U), V(V) and V(W) are displayed in the following diagram. The rectified output voltage V(OUT) is also displayed in probe window.



Note

Of course you can use the library for other PSpice projects. However, after placing the part in the schematic, you should check the property **Implementation Path.** For example:

- 1. Using the part for other designs which are located in the same system where the design create_hierarchy_part.dsn is located, you may not need to modify the **Implementation Path.**
- 2. If you saved the library (.OLB) somewhere, you must check the **Implementation Path.** If necessary, change the path of the library.

New Property App	ly Display Delete Property Pivot Filter by: < Current properties >
	A
	+ root: PAGE1: U1
Graphic	my_hierarchy_part.Normal
ID	
Implementation	my_hierarchy_part
Implementation Path	D:\PSpice\Application_Notes\FlowCAD_AN_PSpice_Create_HierPart\CREATE_HIERARCHY_PART.OLB
Implementation Type	Schematic View
Location X-Coordinate	320
Location Y-Coordinate	200
Name	INS49782
Part Reference	U1
PCB Footprint	

6 Bibliography

- [1] PSpice User's Guide, Cadence
- [2] OrCAD Capture User's Guide, Cadence