

Title: Generating a hierarchical Part

Product: OrCAD Capture, OrCAD PSpice A/D,
 OrCAD PSpice AA and AMS Simulator

Summary: This application note shows you how to
 generate a hierarchical part based on a
 schematic view, and how you can use this
 part for your PSpice projects

Author/Date: Wei Ling / 24.08.2010

Table of Contents

1	Introduction	2
2	Creating a Circuit to generate a Part	2
3	Generating a Part.....	3
4	Editing the new Part	5
5	Using hierarchical Part in PSpice Project	6
6	Bibliography	7

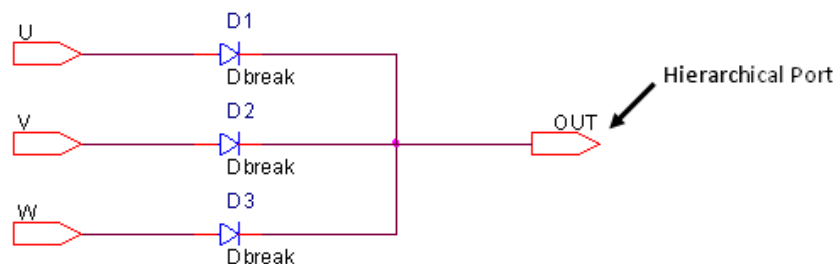
1 Introduction

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

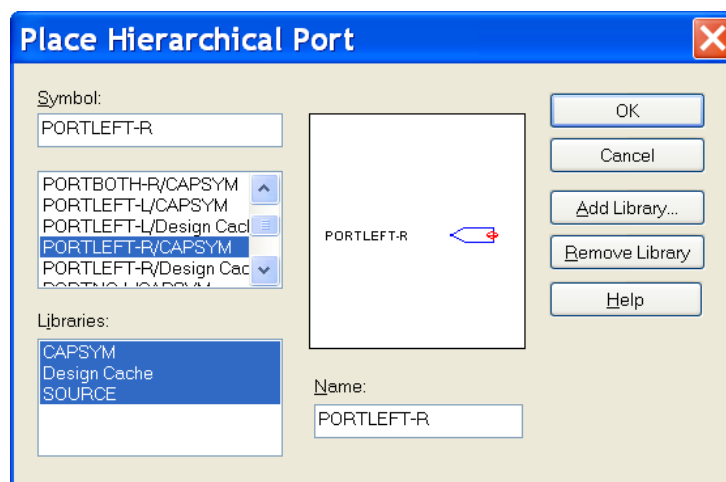
As an example, we would like to build up a circuit consisting of three diodes which we can use as a rectifier for three-phase electric power.



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... →

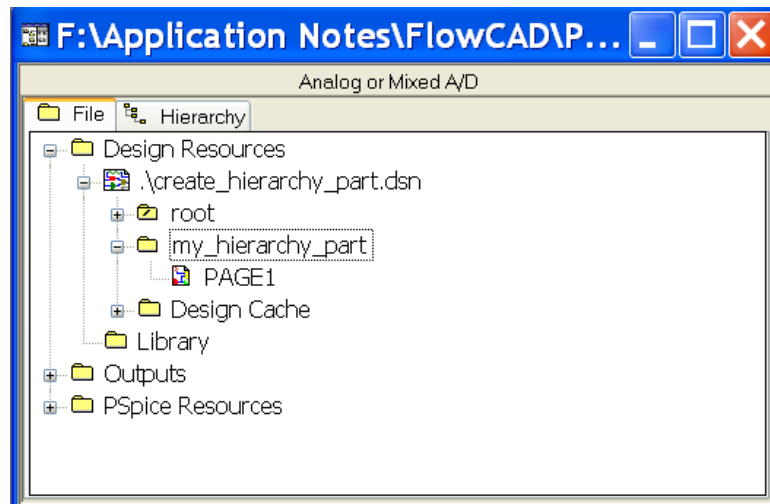
A **Place Hierarchical Port** window comes up:



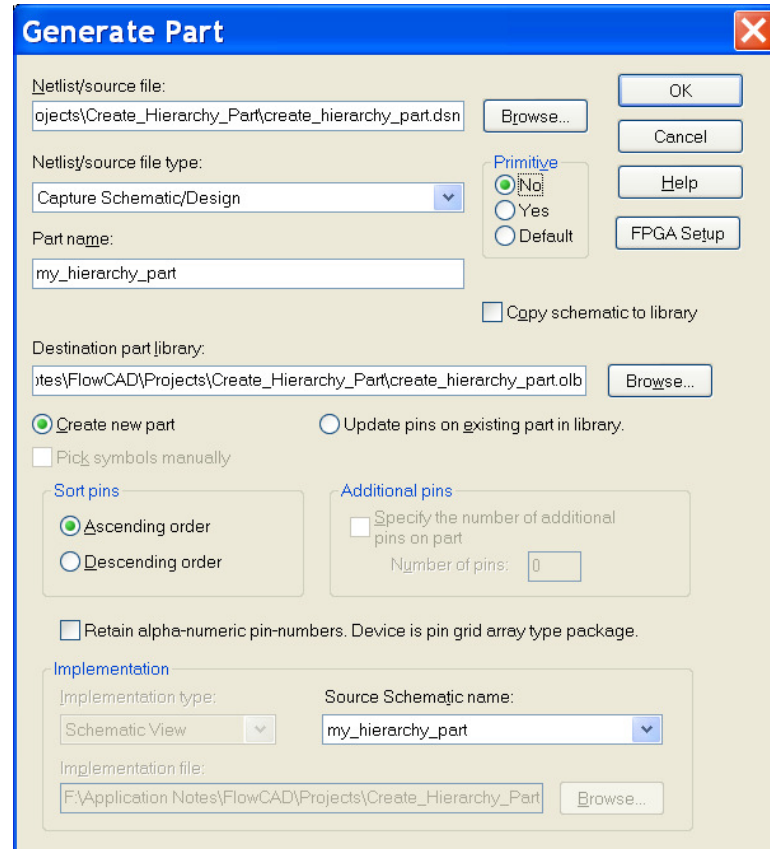
In the **Place Hierarchical Port** window you can choose the hierarchical port which you want to use in your schematic.

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.



In Project Manager window, select the schematic folder my_hierarchy_part → then click **Tools** → **Generate Part...** → the **Generate Part** dialog window comes up.



- 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	Type	Pin Visibilit	Shape	PinGroup	Position	Section
1	OUT	Output	<input checked="" type="checkbox"/>	Line		Right	1
2	U	Output	<input checked="" type="checkbox"/>	Line		Right	1
3	V	Output	<input checked="" type="checkbox"/>	Line		Right	1
4	W	Output	<input checked="" type="checkbox"/>	Line		Right	1

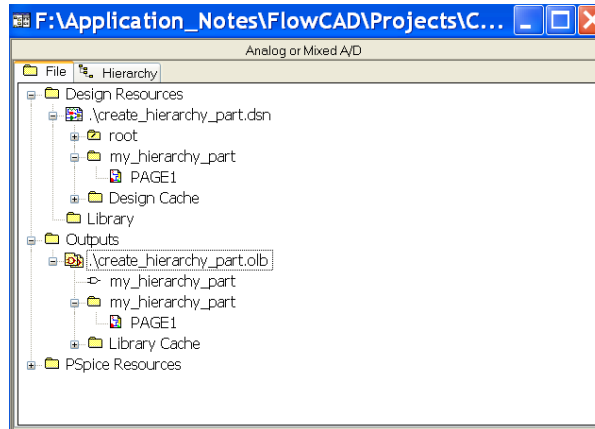
Click **Save** to save the part.

Note:

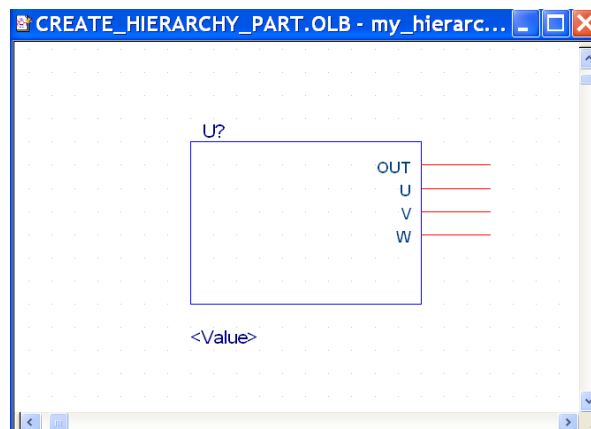
You can edit the part in this **Split Part Section Input Spreadsheet** window. You can also edit it later in **Part Editor**.

4 Editing the new Part

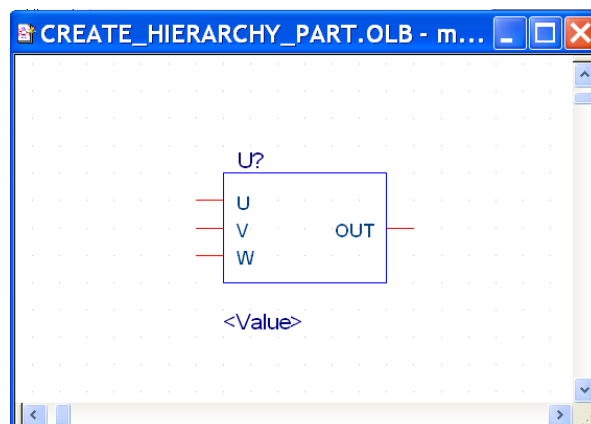
After saving the part you can find a part named my_hierarchy_part located in library create_hierarchy_part.olb. A copy of the schematic for easy portability and design reuse is also saved in this library.



Select the part → **Edit** → **Part** → the Part Editor opens



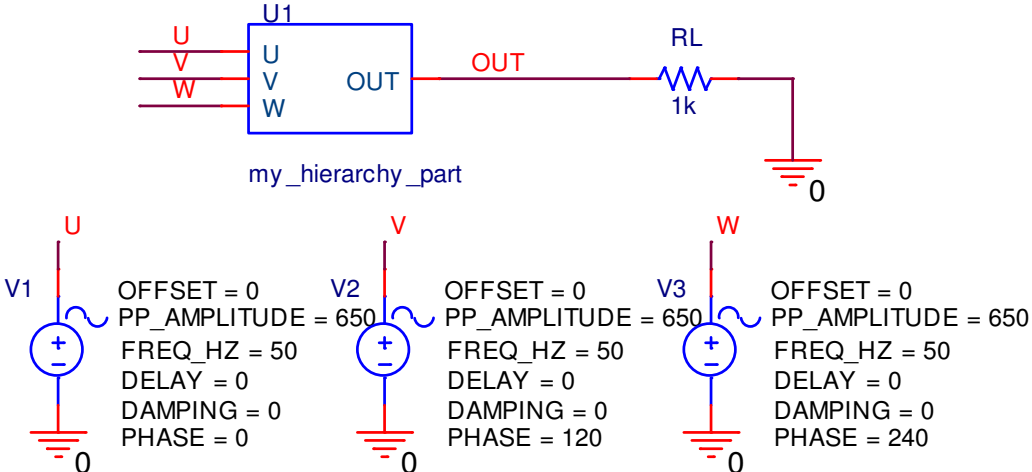
In Part Editor you can change the graphical symbol and modify the pin definition.



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

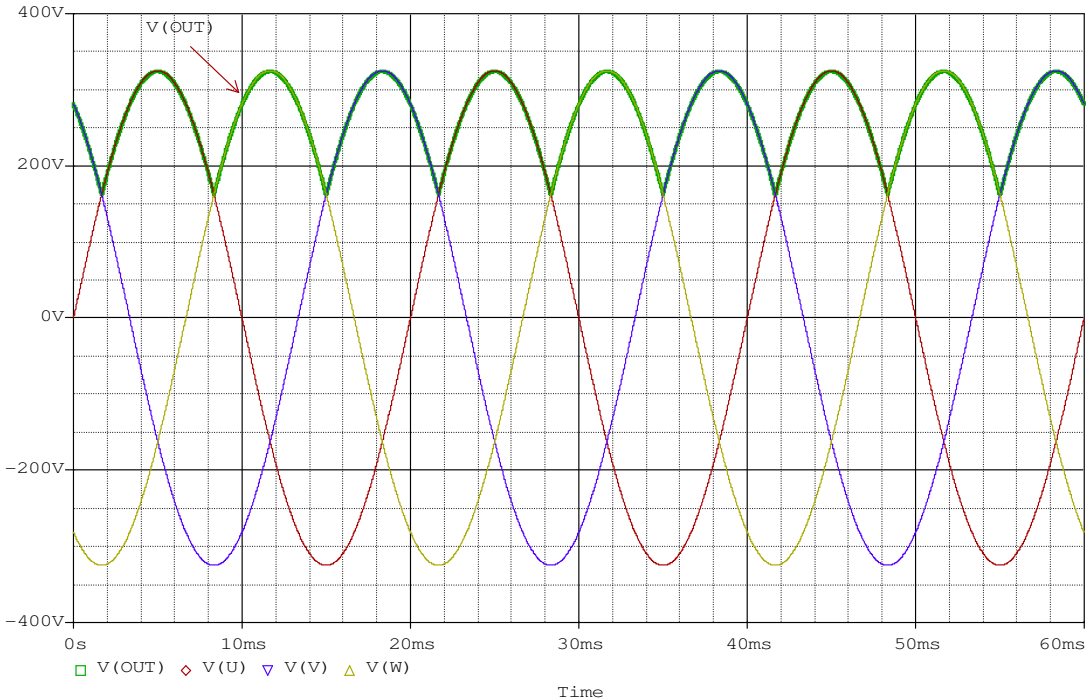
5 Using hierarchical Part in PSpice Project

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.

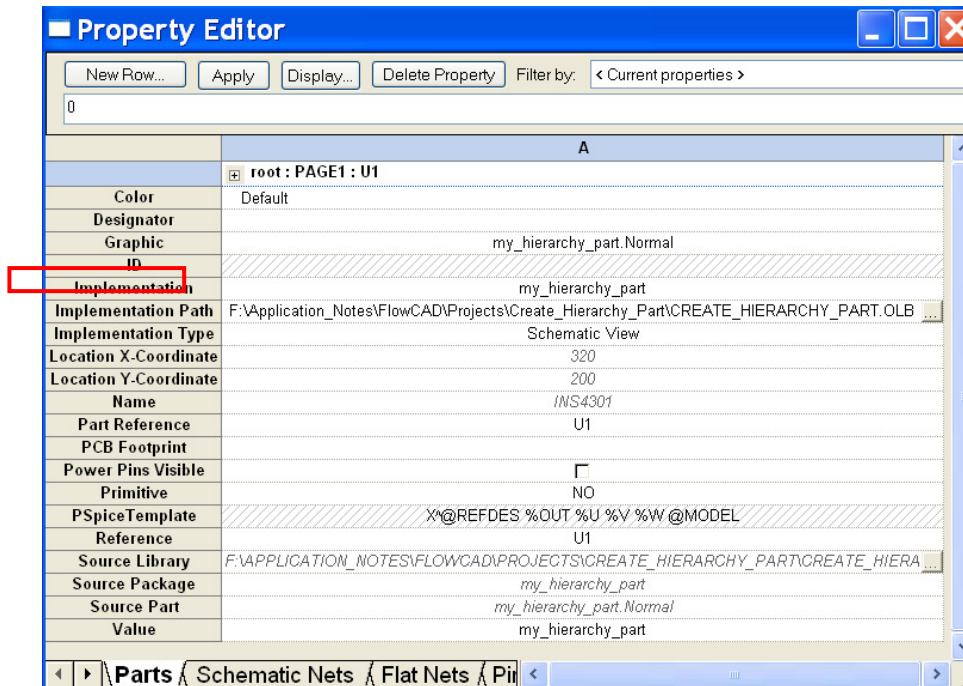
Run the simulation for example, for 60ms. The three inputs V(U), V(V) and V(W) are displayed in the following diagram. The rectified output voltage V(OUT) is also displayed in probe window.



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 **Implementaion Path**.
2. If you saved the library (.OLB) somewhere, you must check the **Implementation Path**. If necessary, change the path of the library.



6 Bibliography

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