

Sources with Tolerances



PSpice Application Note | V2



Table of Contents

1	Introduction	3
2 2.1	Detailed Explanation on a DC Voltage Source	3 3
2.1.1	Designing the Equivalent Circuit Diagram	4
2.1.2	Transferring Values from Symbol to Subcircuit	5
2.2	Creating the New Part	5
2.2.1	Creating Automatically the Symbol	5
2.3	Designing the Source Symbol	7
3	Using the New Source in a Design	8
4	AC Voltage Source	8
5	Sinus Voltage Source	9
6	DC Current Source	9
7	AC Current Source	9
8	Sinus Current Source	9
9	Reuse of the .olb1	0
10	Use of the Appended Files1	0



1 Introduction

This document gives an overview of how a user can create a source that varies its voltage or current during the Monte Carlo Analysis in PSpice AD.

2 Detailed Explanation on a DC Voltage Source

Step by step is explained how to create your own source.

2.1 Creating a New Project

OrCAD Capture is started. No Project or Design is opened.

- File > New > Project
- Name it My_Sources and select Analog or Mixed A/D



Click OK.



• Choose Create a blank project and click OK.





• Right-click on the **new Schematic Folder > New Page > OK.**

2.1.1 Designing the Equivalent Circuit Diagram

- Open the new Schematic.
- Place > PSpice Component for the DC-Source and the Resistor.
- In the Library Analog you find the Part E, or Place > PSpice Component > Source > Controlled Source > VCVS.
- Place the 0 Ground.
- Place the Hierarchical Port.



• Add the wires, change value of the resistor and the names and of the Hierarchical Ports.



To simplify the schematic there is a current source. The voltage value will be interpreted as current value. $[V] \rightarrow [A]$. As long as the Resistor R1 is 1 Ω , this works fine. At the Input of E1 will be the voltage you define.

FlowCAD



2.1.2 Transferring Values from Symbol to Subcircuit

So that the tolerance and the voltage can be set in the schematic, in which the My_Source will be placed, it is necessary the following:

- Add from the special Library the part SUBPARAMETERS.
- Add new Properties to the Part SUBPARAMETERS and call them for example: DCVoltage and DCVTol. Add some default values to the new Properties.
 For better readability they are set visible.

	SUBPARAMETERS DCVoltage = 1 DCVTol = 10	<u>.</u>			
DC = @DCVoltage	R1 1 TC 1	DLERANCE = @I	DCVTol	1 © GAIN = 1	+

- Double click on the source.
- Add the value **@DCVoltage** to the Property DC and set it to visible.
- DoubleClick on R1.
- Add the value @DCVToI to the Property TOLERANCE and set it to visible.
- Save the schematic page and close it.

Note

Do note create Properties with a name that is already used by the system.

The percent sign % for the TOLERANCE Property shall only be placed ones from the main schematic to the subcircuit. If it is missing, or more as one time in the variable chain, you will get a similar error message:

ERROR(ORNET-1019): Pin in template not found on V5.R1

2.2 Creating the New Part

2.2.1 Creating Automatically the Symbol

- Select the schematic folder in the Project Manager.
- Click on **Tools > Generate Part.**

FlowCAD

enerate Part			2
Netlist/source file:			ОК
I_PSpice_Sources_with_tolera	nces\my_sources.dsn	Browse	Canad
Netlist/source file type:		Primitive	
Capture Schematic/Design	•	No No Yes	Help
Part name:		 Default 	FPGA Setup
MY_DC_V_Source			
		📃 Copy sche	matic to library
Destination part library:			
s\FlowCAD_AN_PSpice_Sour	rces_with_tolerances\m	y_sources.olb	Browse
Oreate new part	🔘 Update pins on	existing part in	ibrary.
Pick symbols manually		2.	
Sort pins	Additional pins		
Ascending order	Specify the	number of addi	ional
Descending order	Number o	f pins: 0	
_			
🔲 Retain alpha-numeric pir	n-numbers. Device is pin	n grid array type	package.
Implementation			
Implementation type:	Source Schematic	; name:	
Schematic View 👻	MY_DC_V_Sourc	e	•
Implementation file:			
importions as of thes			

- Click on OK.
- The Split part window opens. Close it clicking on **Save**.

Part Na	ame: [MY_DC_V]	_Source	No. of Sections:	1	Part Ref Prefix:	U	Part Numb	ering eric 🔘 Alphabetic
	Number	Name	Туре	Pin Visibility	Shape	PinGroup	Position	Section
		+	Output		Line		Right	1
2		-	Output		Line		Right	1



2.3 Designing the Source Symbol

The symbol that is generated automatically with the generate Part function is a square with pins. You can design it the way you like:

• Double click on the Symbol in the .olb.



• Edit the Symbol as you like.

	U?		· · · ·			
	+					V?
					<value></value>	+
					DCVOLTAGE = 1	
			 		DCVTOL = 10%	
· · · -		_1				
· · · · C	CVTOL = 10					
				For example:		

- To change the reference Designator or the part name click on Options > Part Properties...
- Add the percent sign in the DCVTol Property.
- Change the settings of the Pins clicking on the pins.

n Properties		23
Name: + Number: 1	Shape: Line Type: Output	OK Cancel
Width	Pin Visible	Help

• Save your new symbol.



3 Using the New Source in a Design

- File > New > Project
- Add the new .olb My_Sources:



- Create a new simulation Profile.
- Add the Library in Simulation Profile.
- Add the Monte Carlo settings with the Output Variable.



• **Run** the simulation.

4 AC Voltage Source



Equivalent Circuit Diagram





Equivalent Circuit Diagram

6 DC Current Source

	SUBPARAMÉTERS: DCCURRENT = 1	
	DCITol = 5	
		· · · · · · ·
	.	
DC = @DCCurrent	≶ . R1	GAIN = 1
DC = @DCCurrent	R1 1 TOLERANCE = @DCITol	GAIN = 1
	R1 1 TOLERANCE = @DCITol	GAIN = 1
	R1 1 TOLERANCE = @DCITol	GAIN = 1

Equivalent Circuit Diagram

7 AC Current Source



Equivalent Circuit Diagram

8 Sinus Current Source



Equivalent Circuit Diagram



9 Reuse of the .olb

Even though you have not chosen **Copy schematic to library** in the **Generate Part** window, you can reuse the part for other designs. You just have to make sure the path for the schematic is correct in the property **Implementation Path**.

Note

You can use the library for other PSpice projects. After placing the part in the schematic, you should check the property Implementation Path.

When you use the part in a design that is located in the same folder as My_Sources.dsn, you may not need to modify the Implementation Path.

If you save the library (.olb) somewhere, you must check the Implementation Path, and change it if necessary.

Property Editor					
New Property Apply Display Delete Property Pivot Filter by:					
	Α				
	SCHEMATIC1 : PAGE1 : U1				
Color	Default				
DCVOLTAGE	\/////////////////////////////////////				
DCVTOL	10%				
Designator					
Graphic	My_DC_V_Source.Normal				
ID					
Implementation	Mv DC V Source				
Implementation Path	C:\Flowcad\My_Sources\MY_SOURCES.DSN				
Implementation Type	Schematic View				
Location X-Coordinate	520				
Location Y-Coordinate	280				
Name	INS412				
Part Reference	U1				
PCB Footprint					
Power Pins Visible	[
Primitive	NO				
PSpiceTemplate	X^@REFDES %+%-@MODEL				
Reference	U1				
Source Library	C:\FLOWCADWY_SOURCESWY_SOURCES.OLB				
Source Package	My_DC_V_Source				
Source Part	My_DC_V_Source.Normal				
Value	My_DC_V_Source				

10 Use of the Appended Files

Unzip the files to a folder called C:\Flowcad\My_Sources.

In **source_test.opj** you will find two simulation profiles one for the AC sources and one for the DC and sinus sources.

