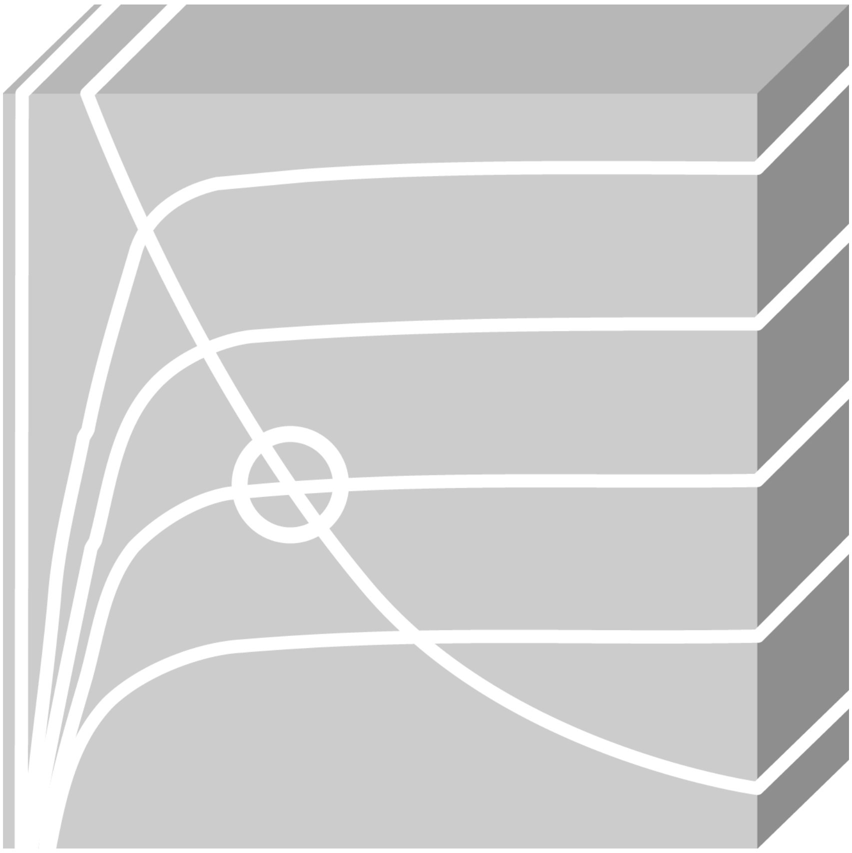
FlowCAD

Sources with Tolerances



PSpice

Application Note | V2

Table of Contents

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.

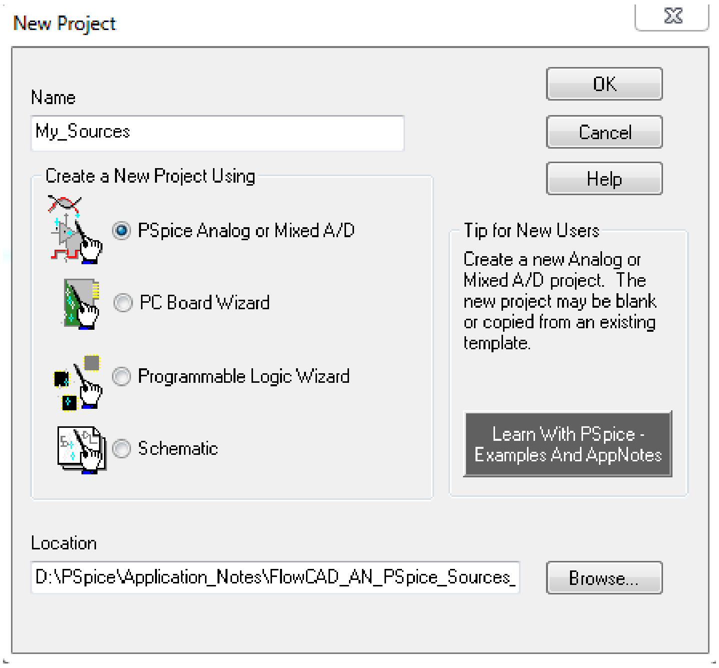
2 Detailed Explanation on a DC Voltage Source

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

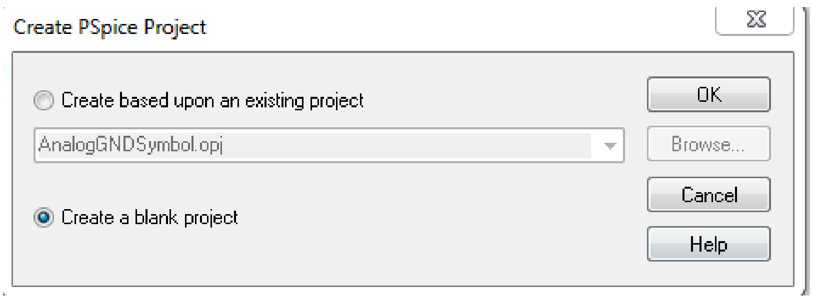
2.1 Creating a New Project

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

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

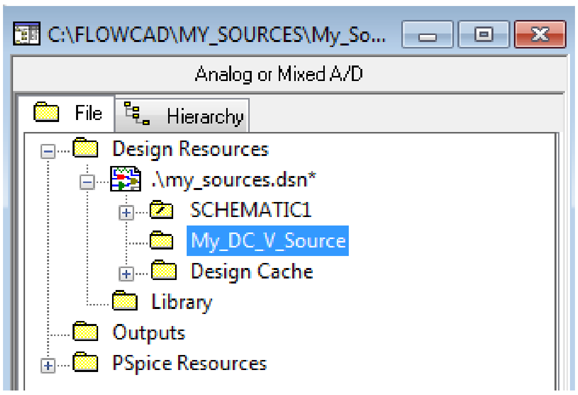


- Click **OK.**



- Choose **Create a blank project** and click **OK.**

- In the Project Manager right click on the .dsn **> New Schematic > Name it** My_DC_V_Source.

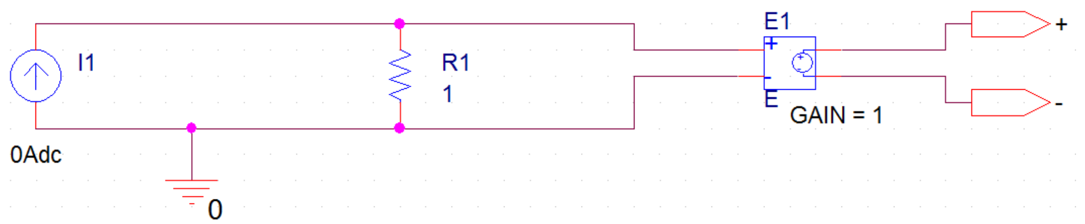


- Right-click on the **new Schematic Folder > New Page > OK.**

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.

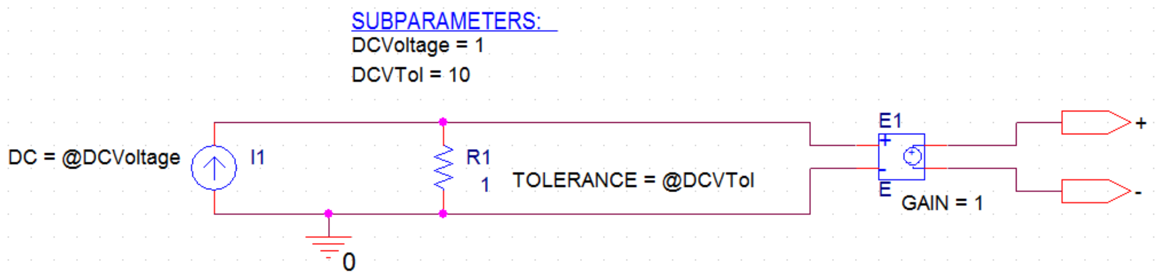


To simplify the schematic there is a current source. The voltage value will be interpreted as current value. [V] → [A]. As long as the Resistor R1 is 1Ω, this works fine. At the Input of E1 will be the voltage you define.

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.



- Double click on the source.
- Add the value **@DCVoltage** to the Property DC and set it to visible.
- DoubleClick on R1.
- Add the value **@DCVTol** 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.

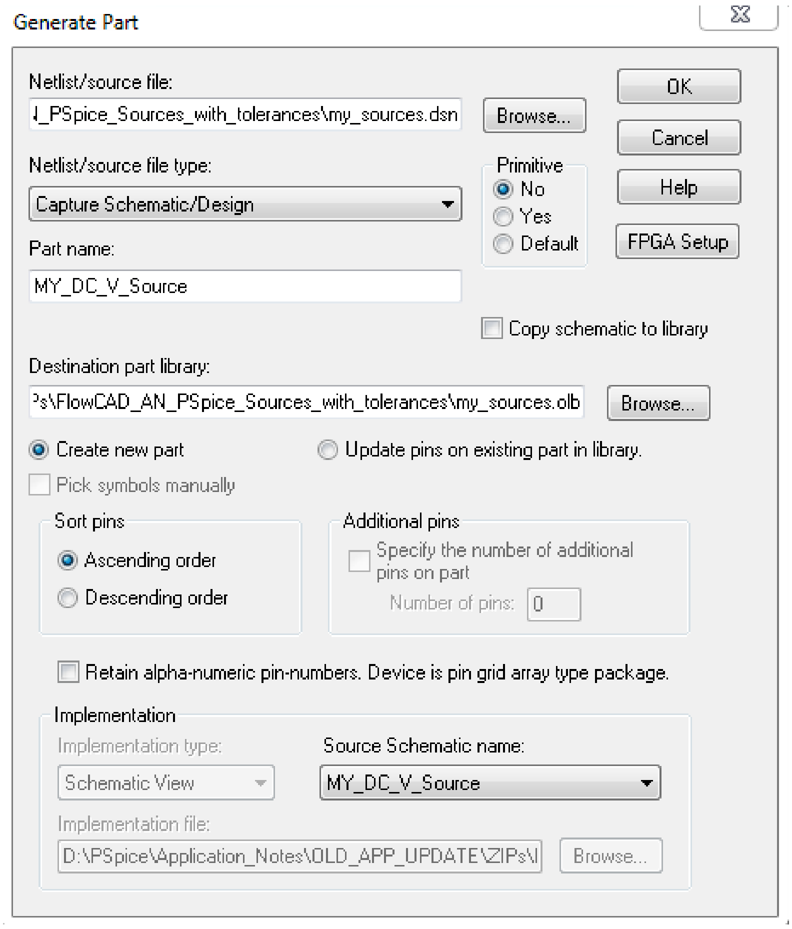
The percent sign % for the TOLERANCE Property shall only be placed ones from the main schematic to the subcircuit. If it is missing, or more as one time in the variable chain, you will get a similar error message:

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

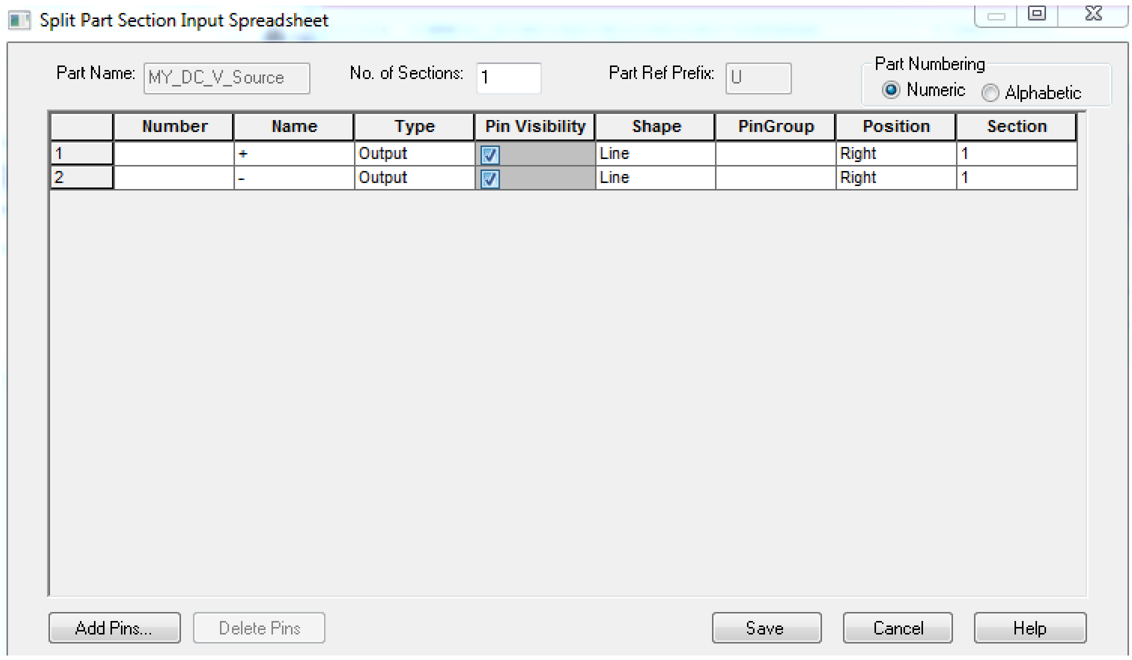
2.2 Creating the New Part

2.2.1 Creating Automatically the Symbol

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



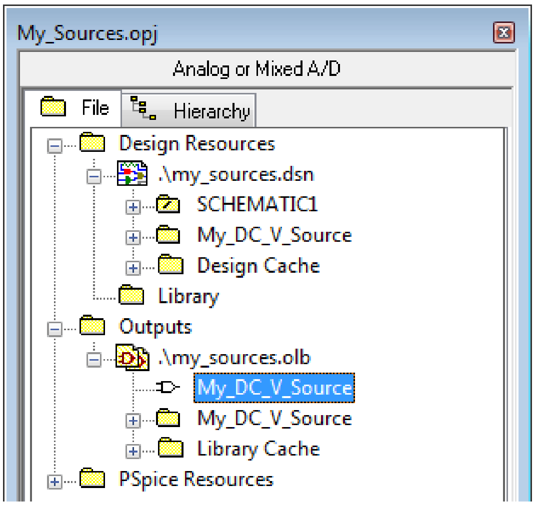
- Click on **OK.**
- The Split part window opens. Close it clicking on **Save.**



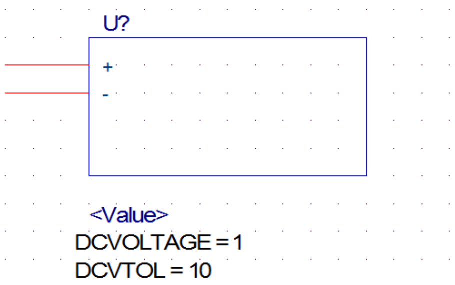
2.3 Designing the Source Symbol

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

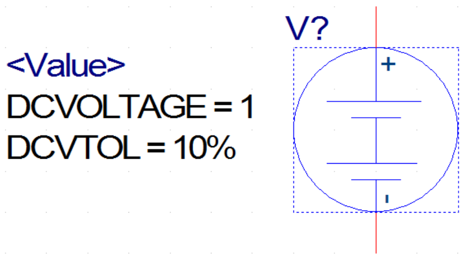
- Double click on the Symbol in the .olb.



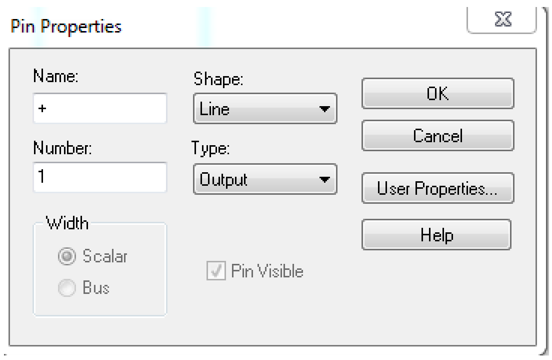
- Edit the Symbol as you like.



For example:



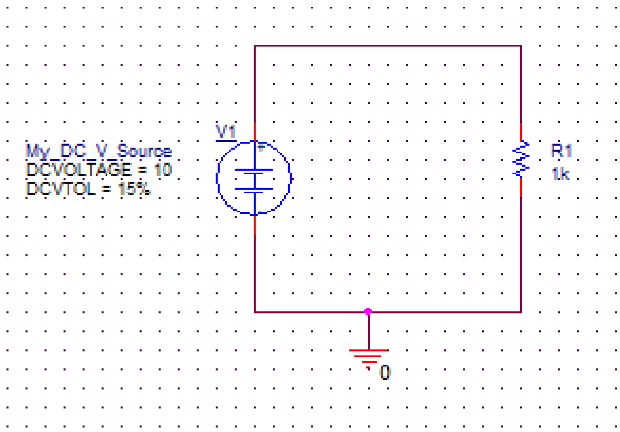
- To change the reference Designator or the part name click on **Options > Part Properties…**
- Add the percent sign in the DCVTol Property.
- Change the settings of the Pins clicking on the pins.



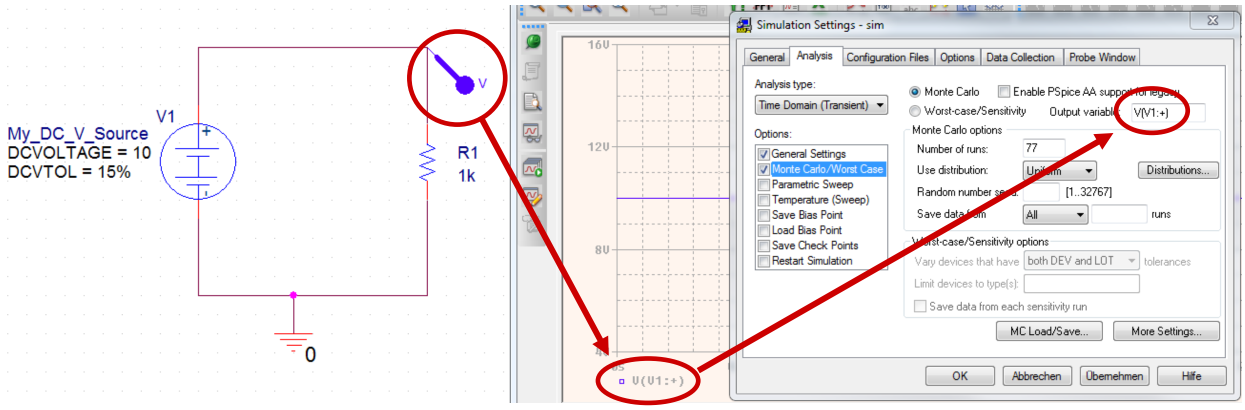
- Save your new symbol.

3 Using the New Source in a Design

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

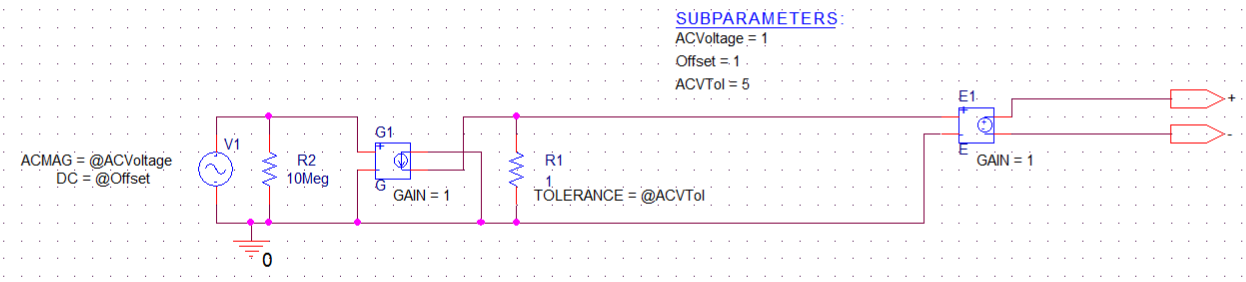


- Create a **new simulation Profile.**
- Add the Library in Simulation Profile.
- Add the Monte Carlo settings with the Output Variable.



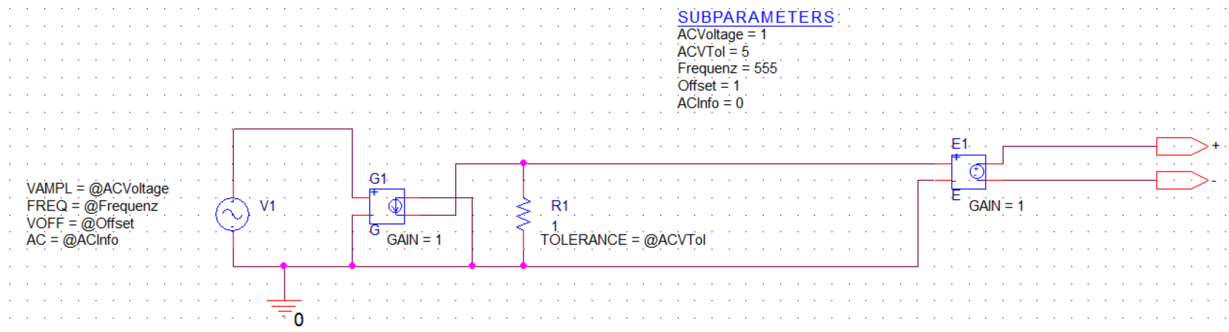
- **Run** the simulation.

4 AC Voltage Source



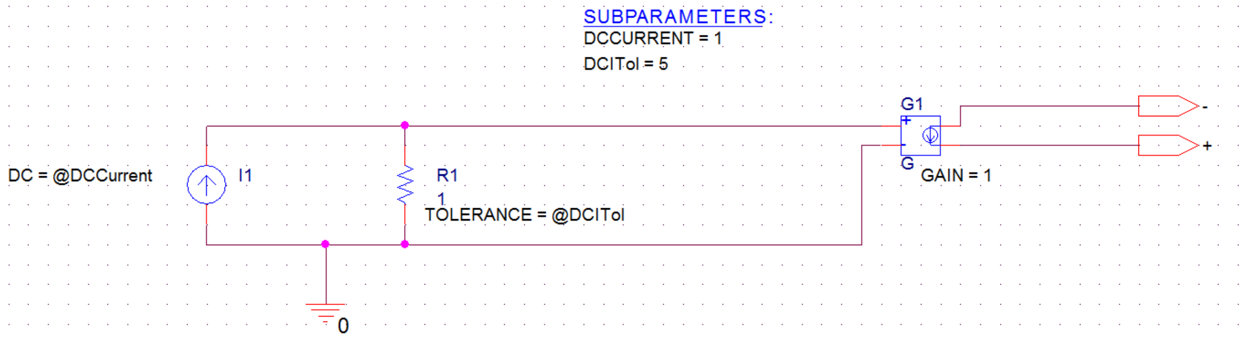
Equivalent Circuit Diagram

5 Sinus Voltage Source



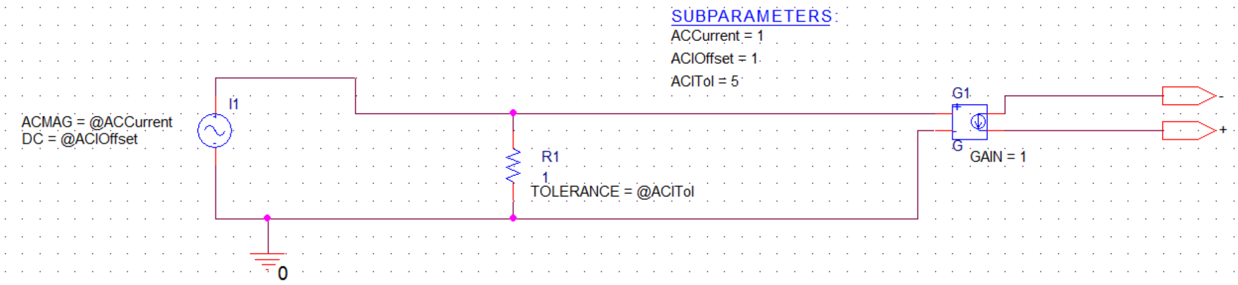
Equivalent Circuit Diagram

6 DC Current Source



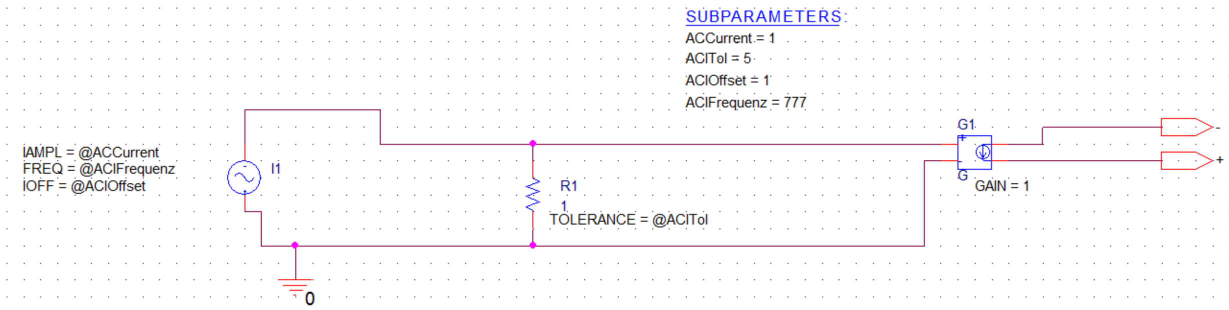
Equivalent Circuit Diagram

7 AC Current Source



Equivalent Circuit Diagram

8 Sinus Current Source



Equivalent Circuit Diagram

9 Reuse of the .olb

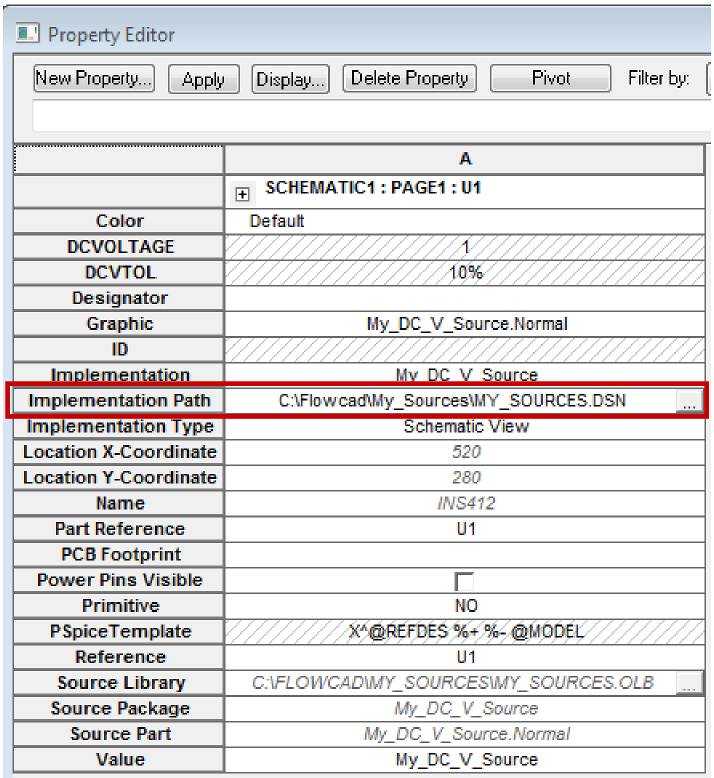
Even though you have not chosen **Copy schematic to library** in the **Generate Part** window, you can reuse the part for other designs. You just have to make sure the path for the schematic is correct in the property **Implementation Path.**

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.



	A
	SCHEMATIC1 : PAGE1 : U1
Color	Default
DCVOLTAGE	1
DCVTOL	10%
Designator	
Graphic	My_DC_V_Source.Normal
ID	
Implementation	My_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:\FLOWCAD\MY_SOURCES\MY_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.