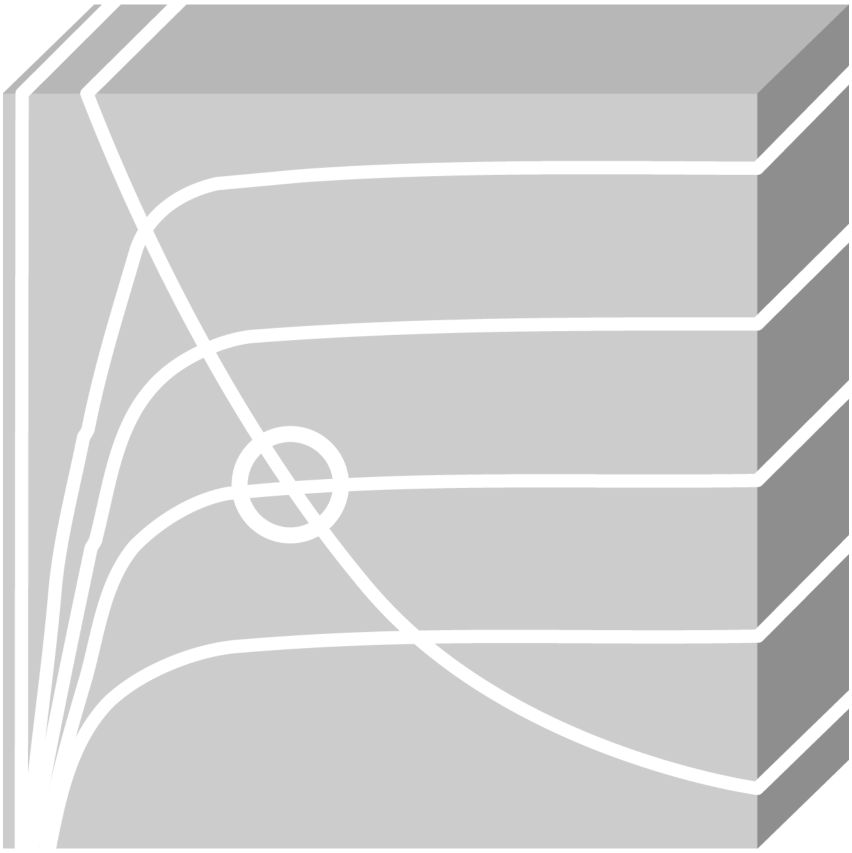
FlowCAD

Import and Export of Sources



PSpice

Application Note | V1

FlowCAD

Table of Contents

1 Introduction

It is known that you can define many voltage and current sources for your design. But sometimes you have measured data from the real word and you want to use them as a source for your PSpice simulation. You can achieve this using stimulus files along with a schematic symbol like a VPWL_FILE for voltage source or IPWL_FILE for current source.

A stimulus file contains time-based definitions for analog and / or digital input waveforms. You can create a stimulus file either:

- Manually by using the Model Text View of the Model Editor (or a standard text editor) to create the definition (a typical file extension is .STM)
- Automatically by using the Stimulus Editor (which generates a .STL file extension).

2 How to Define Stimulus for Simulation

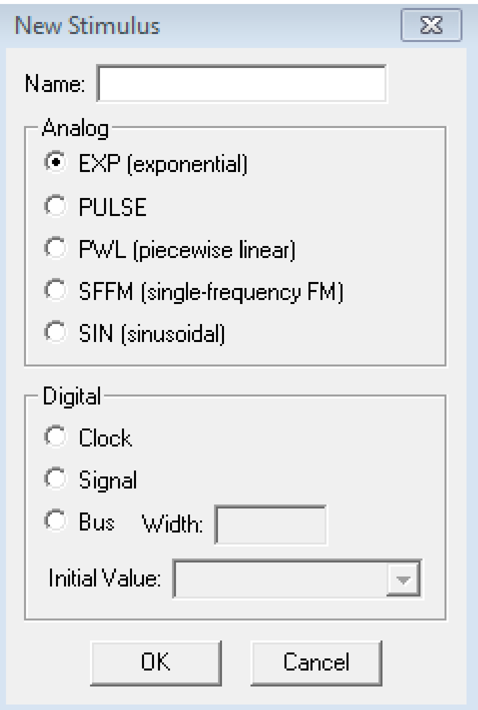
There are many options (manually and automatically) to define stimulus for a simulation and also many extensions that you could use (.stl, .stm, .txt, .csv, etc.).

2.1 Stimulus Editor

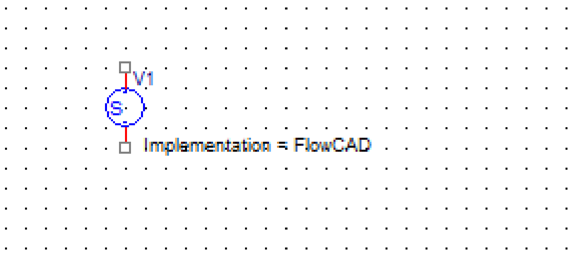
It is a PSpice utility which allows you to create (semi-automatically) different kind of analog and digital signals. After you have defined the signal(s) you need, a .stl file is created.

Start > All Programs > Cadence Release 17.2-2016 > Product Utilities > PSpice Utilities > Stimulus Editor

Just click on **File > New** and then **Stimulus > New** in order to create the source you need:

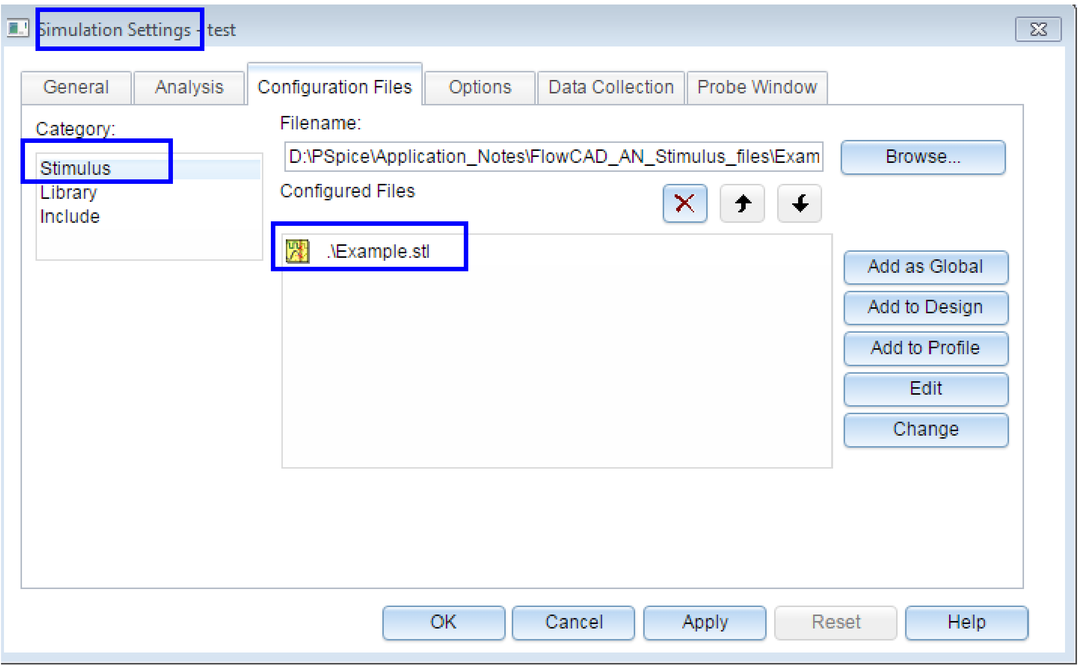


In order to be able to use this stimulus in a simulation you have to connect such file within PSpice using a particular symbol like VSTIM or ISTIM for analog signals and DigStim for digital one, both from the library SOURCSTM.olb. Moreover, you have to add such generated file in the simulation profile.



Note

The value of the property **Implementation** is the name of the source you have created in the Stimulus Editor. In one stimulus file there can be more than one stimulus signal.

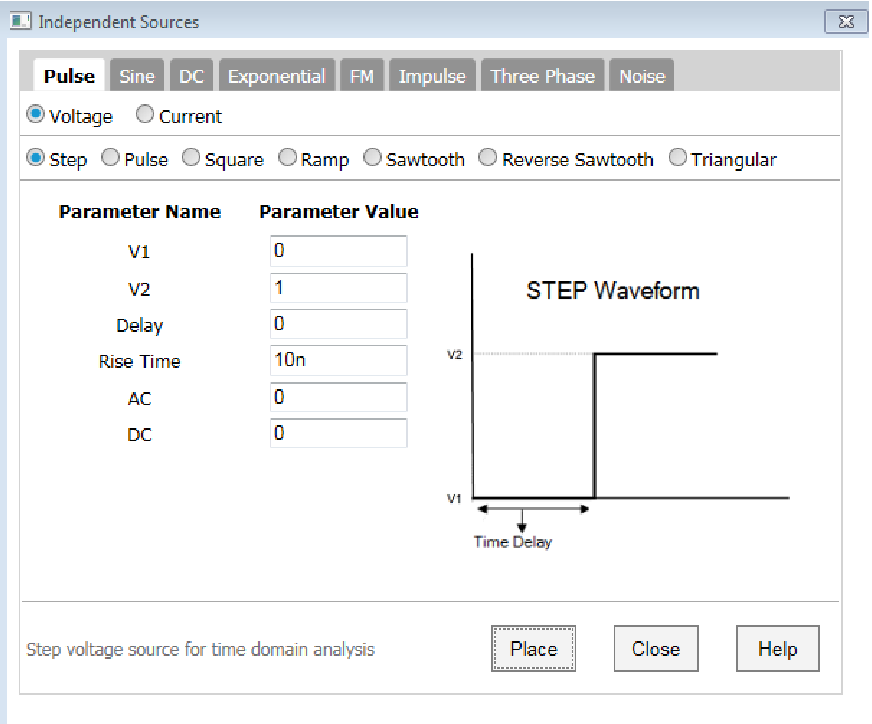


2.2 PSpice Modeling Application

It is a Graphical User Interface placed directly in OrCAD Capture which allows you to define and set up quickly sources, components, etc. You can access to it clicking in Capture on **Place > PSpice Components > Modeling Application > Sources.**

2.2.1 Independent Sources

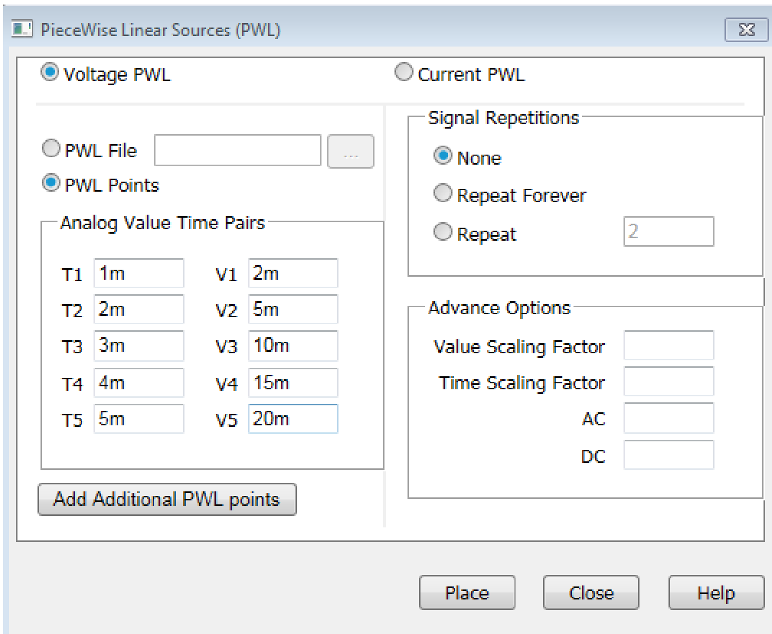
You can define the sources directly from this interface. For these sources you do not need any external source file. For example, if you need a Pulse, you could use this way where you also have a description of each parameters or you could use VPULSE component from the library source.olb.



2.2.2 PWL Sources

Piece Wise Linear Sources (PWL) allows you to define voltage and current stimulus including also a repeat option. This automatically generates a source symbol with your desired source. You have two possibilities to proceed:

- Including manually point by point the different pair points (time – voltage or time – current).



- There is another option. You can directly include a txt or a csv file with the values extracted from an external tool, an oscilloscope, or an user defined. All you have to do is to select PWL File and load such file, selecting if it is voltage or current:

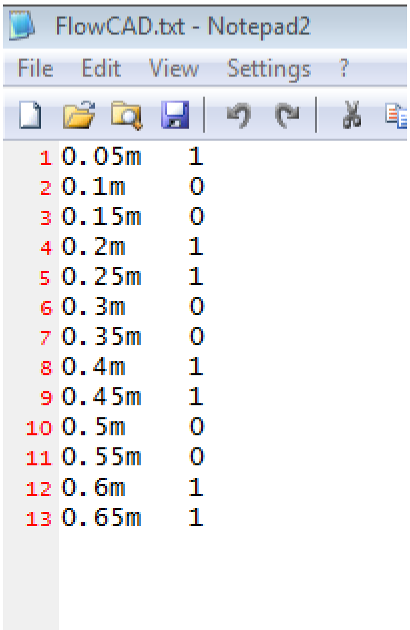
  Note

  In the advanced Options you find two important properties, Value Scaling Factor and Time Scaling Factor. Imagine that the first one is set to 0.1 and the second one to 1. If the voltage at 1us is 416.40625V in my_data.txt file, for the PSpice simulation the voltage at 1us is 41.640625V.

  Once you place the symbol, the file is already attached to it with the whole location of the text file:

	A
	SCHEMATIC1 : PAGE1 : V2
AC	
AppProp	No_reps$FILE_TYPE_PWL$0$0
BiasValue Power	0W
Color	Default
DC	
Designator	
FILE	D:\PSpice\Application_Notes\FlowCAD_AN_Stimulus_files\FlowCAD.txt
FIRST_NPAIRS	
Graphic	VPWL_GENERIC.Normal
ID	
Implementation	
Implementation Path	
Implementation Type	PSpice Model
Location X-Coordinate	470
Location Y-Coordinate	120
Name	INS1003
Part Reference	V2
PCB Footprint	
Power Pins Visible	
Primitive	DEFAULT
PSpiceOnly	TRUE
PSpiceTemplate	V^@REFDES %+ %- ?DC|DC @DC| ?AC|AC @AC| PWL ?TSF|TIME_SCALE_FACT
Reference	V2
REPEAT_VALUE	
SECOND_NPAIRS	
Source Library	C:\CADENCE\SPB_17.2\TOOLS\CAPTURE\TCLSCRIPTS\CAPPSPICESOU
Source Package	VPWL_GENERIC
Source Part	VPWL_GENERIC.Normal
THIRD_NPAIRS	
TSF	
Value	VPWL_GENERIC
VSF	

This option can only be used for analog signals. The format for the text or csv file is this one:



FlowCAD.txt - Notepad2

File Edit View Settings ?

```
0.05m   1
0.1m    0
0.15m   0
0.2m    1
0.25m   1
0.3m    0
0.35m   0
0.4m    1
0.45m   1
0.5m    0
0.55m   0
0.6m    1
0.65m   1
```

In this case we have first column for time (in milliseconds) and the second one for voltage (in this case volts).

2.3 Symbols from the Default Cadence Library

- **VPWL:** This symbol comes from the library source.olb and is used to define time and voltage pairs manually.

  Note
  If you need more pair points, you should have to modify the template for this component or to use the PSpice Modeling Application Option.

	A
	SCHEMATIC1 : PAGE1
AC	
Color	Default
DC	
Designator	
Graphic	VPWL.Normal
ID	
Implementation	
Implementation Path	
Implementation Type	PSpice Model
Location X-Coordinate	470
Location Y-Coordinate	180
Name	INS772
Part Reference	V1
PCB Footprint	
Power Pins Visible	
Primitive	DEFAULT
PSpiceOnly	TRUE
PSpiceTemplate	V^@REFDES %+ %- ?DC|D
Reference	V1
Source Library	C:\CADENCE\SPB_17.2
Source Package	VPWL
Source Part	VPWL.Normal
T1	
T2	
T3	
T4	
T5	
T6	
T7	
T8	
V1	
V2	
V3	
V4	
V5	
V6	
V7	
V8	
Value	VPWL

- **IPWL:** This symbol comes from the library source.olb and is used to define time and current pairs manually.

	A
	SCHEMATIC1 : PAGE1
AC	
Color	Default
DC	
Designator	
Graphic	IPWL.Normal
I1	
I2	
I3	
I4	
I5	
I6	
I7	
I8	
ID	
Implementation	
Implementation Path	
Implementation Type	PSpice Model
Location X-Coordinate	790
Location Y-Coordinate	170
Name	INS848
Part Reference	I1
PCB Footprint	
Power Pins Visible	
Primitive	DEFAULT
PSpiceOnly	TRUE
PSpiceTemplate	I^@REFDES %+ %- ?DC|DC
Reference	I1
Source Library	C:\CADENCE\SPB_17.2
Source Package	IPWL
Source Part	IPWL.Normal
T1	
T2	
T3	
T4	
T5	
T6	
T7	
T8	
Value	IPWL

- **Stim:** This symbol is used for digital sources and it is found in the library source.olb. There is a variation of it depending on the number of bits, stim1, stim4, stim8 and stim16. The data must be added manually.

	A
	SCHEMATIC1 : PAGE1
Color	Default
COMMAND1	0s 0
COMMAND2	
COMMAND3	
COMMAND4	
COMMAND5	
COMMAND6	
COMMAND7	
COMMAND8	
COMMAND9	
COMMAND10	
COMMAND11	
COMMAND12	
COMMAND13	
COMMAND14	
COMMAND15	
COMMAND16	
Designator	
DIG_GND	$G_DGND
DIG_PWR	$G_DPWR
FORMAT	1
Graphic	STIM1.Normal
ID	
Implementation	
Implementation Path	
Implementation Type	PSpice Model
IO_LEVEL	0
IO_MODEL	IO_STM
Location X-Coordinate	620
Location Y-Coordinate	190
Name	INS865
Part Reference	DSTM2
PCB Footprint	
Power Pins Visible	☐
Primitive	DEFAULT
PSpiceOnly	TRUE
PSpiceTemplate	U^@REFDES STIM(@WIDTH
Reference	DSTM2
Source Library	C:\CADENCE\SPB_17.2
Source Package	STIM1
Source Part	STIM1.Normal
TIMESTEP	
Value	STIM1
WIDTH	1

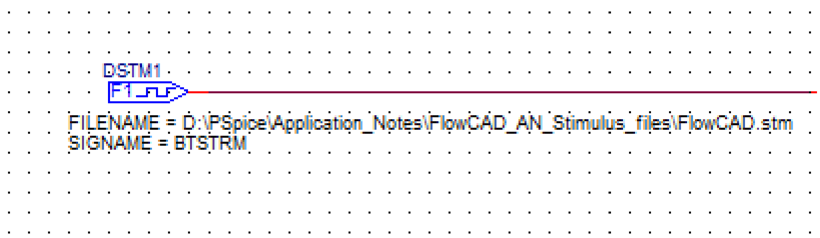
- **VPWL_FILE** and **IPWL_FILE:** These symbols are located in the library source.olb and allow you to use an external text or csv file as stimulus. All you have to do is to indicate where such file is located. Do not forget the possibility to use the properties TSF (Time Scaling Facter) and VSF (Value Scaling Factor).

	A
	SCHEMATIC1 : PAGE1 : V4
AC	
Color	Default
DC	
Designator	
FILE	
Graphic	VPWL_FILE.Normal
ID	
Implementation	
Implementation Path	
Implementation Type	PSpice Model
Location X-Coordinate	680
Location Y-Coordinate	110
Name	INS1218
Part Reference	V4
PCB Footprint	
Power Pins Visible	
Primitive	DEFAULT
PSpiceOnly	TRUE
PSpiceTemplate	V^@REFDES %+ %- ?DC|DC @DC| ?AC|AC @AC| PWL ?TSF|TIME_SCALE_
Reference	V4
REPEAT_VALUE	
Source Library	C:\CADENCE\SPB_17.2\TOOLS\CAPTURE\LIBRARY\PSPICE\SOUR
Source Package	VPWL_FILE
Source Part	VPWL_FILE.Normal
TSF	
Value	VPWL_FILE
VSF	

- **FileStim:** This symbol is used for digital sources and it is found in the library source.olb. There is a variation of it depending on the number of bits, FileStim1, FileStim2, FileStim4, FileStim8, FileStim16 and FileStim32.

  In this case you have to define where the txt, csv or stm is located and also the name of the signal.

	A
	SCHEMATIC1 : PAGE1 : DSTM1
Color	Default
Designator	
FILENAME	D:\PSpice\Application_Notes\FlowCAD_AN_Stimulus_files\FlowCAD.stm
Graphic	FileStim1.Normal
ID	
Implementation	
Implementation Path	
Implementation Type	PSpice Model
IO_LEVEL	0
IO_MODEL	IO_STM
Location X-Coordinate	*350*
Location Y-Coordinate	*310*
Name	*INS412*
Part Reference	DSTM1
PCB Footprint	
Power Pins Visible	
Primitive	DEFAULT
PSpiceOnly	TRUE
PSpiceTemplate	U^@REFDES FSTIM(1) %VCC %GND %PIN1 @IO_MODEL FILE="@FILENAME" IO
Reference	DSTM1
SIGNAME	BTSTRM
Source Library	*C:\CADENCE\SPB_17.2\TOOLS\CAPTURE\LIBRARY\PSPICE\SOURCE.O*
Source Package	*FileStim1*
Source Part	*FileStim1.Normal*
Value	FileStim1



The format for this file depends on your requirements, but for a simple one, it looks like this:

```
TIMESCALE=1ms
BTSTRM

0.05   1
0.1    0
0.15   0
0.2    1
0.25   1
0.3    0
0.35   0
0.4    1
0.45   1
```

The first column is the time (in milliseconds) and the second one the values.

Note
If you need more information about the FileStim options and properties, you will find it in the PSpice Reference Guide (p. 435 – 440).

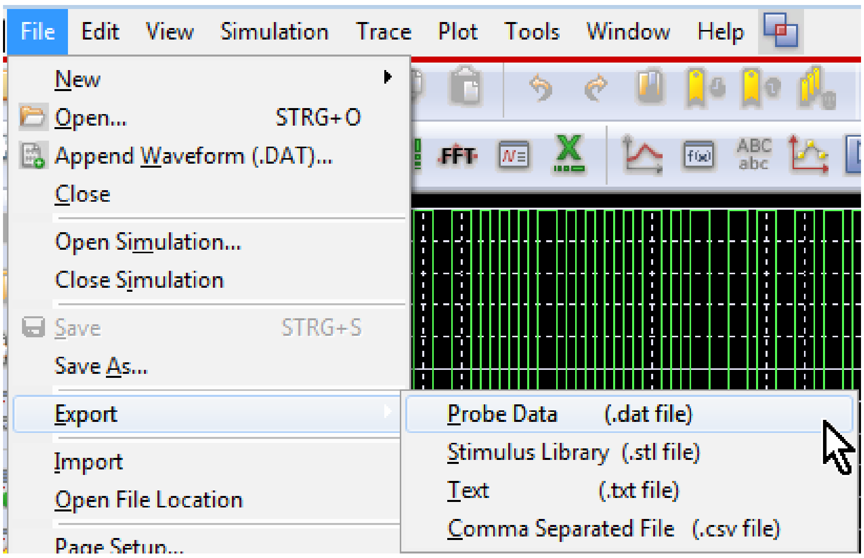
3 Export PSpice Results

There are many options to export PSpice results to be used in another software, as documentation or as source to be used in another PSpice simulation.

3.1 Analog Signals

For analog signals, there are two options to export data:

- From PSpice, click on **File > Export.** You can generate a dat, stl, txt or csv file.



- From the plotted signals in PSpice Probe window, you can select the signals you want to export, then you click on **Ctrl+C** and then in a text file **Ctrl+V.**

Note

It is also possible to export data directly from PSpice to MATLAB and then use it for post processing and visualization capabilities. But for this, it is necessary a particular license.

3.2 Digital Signals

For digital signals there is only a way to export data. You have to use the component vector1, vector2, vector4, vector8, vector16 or vector 32 from the library special.olb

You have to indicate the location where the text file (.vec) must be generated and the name for the signal. So, when you simulated automatically a file with the whole data is created in the directory you specified.