

## Simulation Hints

SWCADIII is recommended for simulation. This is fairly intuitive to use and also has extensive on-line help. This document just mentions some things relevant to gEDA and Spice simulation.

Information about Spice can also be found in the Spice User's Guide on the H: drive in gEDA/doc/Spice3F5Manual.html. There is also a (rather old) gEDA Spice Howto document there.

## SPICE values

- Note the following from the "Spice User's Guide" relating to scale factors:

A number field may be an integer field (12, -44), a floating point field (3.14159), either an integer or floating point number followed by an integer exponent (1e-14, 2.65e3), or either an integer or a floating point number followed by one of the following scale factors:

12	9	6	3	-6
T = 10	G = 10	Meg = 10	K = 10	mil = 25.4

-3	-6	-9	-12	-15
m = 10	u (or M) = 10	n = 10	p = 10	f = 10

- Because spice is not case sensitive, note that 1M means 1 milli, not 1 Mega (which you would have to explicitly put as 1Meg).

- And also:

Letters immediately following a number that are not scale factors are ignored, and letters immediately following a scale factor are ignored. Hence, 10, 10V, 10Volts, and 10Hz all represent the same number, and M, MA, MSec, and MMhos all represent the same scale factor. Note that 1000, 1000.0, 1000Hz, 1e3, 1.0e3, 1KHz, and 1K all represent the same number.

The context defines whether it's a voltage or something else you are specifying.

There's no harm in putting, eg 10V to indicate 10 volts though - at least it's a reminder to you what it represents. Spice will just ignore the "V".

## Gschem SPICE symbols

- The "value" attribute of spice source symbols contains the spice source type (eg DC or pulse) followed by its parameters.
- VDC - change the parameter in the "value" attribute to the DC voltage you want.
- VPULSE - this has 7 parameters in the "value" attribute in the following order:
  - V1 - initial voltage value
  - V2 - pulsed value
  - TD - delay time
  - TR - rise time
  - TF - fall time
  - PW - pulse width
  - PER - period

eg pulse 0 5 0 0 0 1m 1

would be a 5 volt pulse of 1 millisecond length repeated every second.
- Spice Directive. This is a spice command you want included in the spice netlist. Typically you would set the "value" attribute to the type of analysis you want with any parameters it needs (eg .tran 500u to do a transient analysis for 500 microseconds).

## Gschem Reference Designators

- Reference designators ("refdes" attribute) appear unchanged in the spice netlist.
- Spice makes assumptions about what model is required based on the first letter of the refdes (eg R101, R6, Ronaldo are all resistors, C8, CONN1 are both capacitors etc), so you MUST use appropriate refdes values.
- Complex circuits such as opamps are modelled as "sub-circuits" - a kind of spice function. The spice code for each sub-circuit in the design is included by the netlist generator, which therefore must be told where to find this code.
- Spice models for many components are available from manufacturer's web sites.
- For these kind of components, in gschem set the "model-name" attribute of the symbol to the name used in the relevant .SUBCKT line of the spice model file (there may be more than one device modelled in a single file). Set the "file" attribute to the path to the model file. (If you are in cygwin, remember that, for example, the C: drive is accessed by /cygdrive/c .)

## Gschem to Spice Netlist

- Use the "gnetlist" netlist generator:

```
gnetlist -g spice-sdb yourpage1.sch yourpage2.sch ...
```

The spice netlist will be in output.net, which you can load into swcadiii.

## SWCADIII Simulation

- When you load the spice netlist you will see that it is a simple text file.
- A line such as:

```
R12 1 foo 10k
```

signifies a resistor (R12) connected from net "1" to net "foo" and having a value of 10,000 ohms.

- If nets are given names in gschem, these names will appear in the spice netlist, otherwise numeric names are assigned. It's a good idea to give names to any nets you might want to look at so that you can easily choose to monitor them in swcadiii.
- You can edit the netlist within swcadiii, for example to change component values or analysis options (but this won't feed back into your schematic diagram).