Using WinSPICE

Why Use WinSPICE?
It's free.
It works…REALLY well.
No limitations on the number of nodes or transistors.
It can handle the EKV model (v2.6) well…this means you can simulate in subthreshold.

Setting Up WinSPICE
Download at
http://winspice.com

Get the program under the "Spice3F4" heading.
Also, obtain the latest version under the "Latest" section. Simply unzip this file and copy the 3 files (readme, SPICEMAN, and wspice3) into the same directory that contains the executable file.

Basic Operation
Currently, WinSPICE has no graphical schematic entry – everything is done with a text SPICE deck. Generally speaking, this is done the same way as in any other SPICE simulator.

There are two ways to get WinSPICE to analyze a circuit (in a .cir file).

1. Open WinSPICE and move to the directory where the file is located (ex. "cd c:\spice"). At the command prompt, type "source filename.cir".
2. Simply double click on the appropriate file. This will automatically cause WinSPICE to analyze the circuit.

You can use your favorite text editor to change the SPICE deck. If you just analyzed a SPICE deck (and the WinSPICE window is still open), then making changes to the SPICE deck and saving it will automatically cause WinSPICE to analyze the circuit again.

Saving Data to the Workspace
You must save data to the workspace in order to plot from the command line or save to an external file.

Use the ".SAVE" command within the SPICE deck to save important data. You can either save only specific parameters (".SAVE parameter_1 parameter_2 …") or all the parameters (".SAVE ALL"). However, if you want to see the current through a transistor, you must specifically save that data. For example, if you want the current through MOSFET 1, then use the command ".SAVE @M1[ID]".

Finding Out What Data is Available in the Workspace
To determine what data is available to manipulate, plot, or write to a file, you can type "let" or "display" at the WinSPICE command line. Consult the WinSPICE manual to determine the nuances.

Plotting
You can plot the dependent variables in WinSPICE automatically from the SPICE deck, or you can plot it from the command line.

To plot from the SPICE deck, you need to use the .PLOT command. Type ".PLOT analysis_type parameter_1 parameter_2 …" where analysis_type is the type of analysis you are running. For example, this could be "AC" if you are running an AC analysis.

To plot from the command line, type "plot parameter_1 parameter_2 …". This time, you do not include the analysis type.

You can zoom in on any plot by simply right clicking and dragging with the mouse. This will open up a new window with the zoomed-in plot.

Saving Data to a File
If you simply want all the data in a text file, type “write myfile.raw <specific parameters>”. Without any specific parameters, it simply saves all the data.

If you want the data in a fashion that can be loaded into a spreadsheet or into MATLAB, save it to a comma-separated-value (.csv) file. Type “write myfile.csv <specific parameters>”. Again, without any specific parameters, all the data will be saved.

The specific parameters can be written like “v(1)” “@m1[id]” for the voltage at node 1 and the current through MOSFET 1, respectively. All the parameters that are saved to a file must have been saved to the workspace through the “.SAVE” command in the SPICE deck.

Interpreting the .csv File
A .csv file should be automatically opened up in a spreadsheet program like Excel. The first row tells what is contained in each column. The first column is always the independent variable.

For most types of simulations, there is only one column for each voltage or current. However, for an AC analysis, there are two columns for each variable. The first corresponds to the real part, and the second corresponds to the imaginary part. Do not be confused by the labeling of the columns only going halfway over. Each label gets two columns. The first two columns are for the frequency. The third and fourth columns are for the real and imaginary part of the first variable that was written to file, and it continues until the end.

To load these .csv files into MATLAB, you can use the csvread function. In order to get all the data excluding the first row (that only contains the column names), type “variable = csvread(‘myfile.csv’,1,0);”. Each column in the MATLAB matrix corresponds to each column in the spreadsheet.

SPICE3 Functionality
WinSPICE uses SPICE3, which has all sorts of fun new toys to play with in SPICE. You can do things like sweep parameters and other such things. In order to do it, you start a line with “*#”. This is for backwards compatibility with earlier versions of SPICE. Typically, the “*” comments out a line, but the addition of the “#” directly afterwards makes it a normal line of operation within WinSPICE (or any other program that can run SPICE3). Actually, any of these lines will be operated on first when the circuit is analyzed. Consult the WinSPICE manual for more details.

Using the EKV Model
The EKV model (v2.6) works over the entire operating range of a MOSFET. However, it seems to be the best model for simulating MOSFETs in subthreshold. Use “LEVEL=44” in the .MODEL statement to implement the EKV model. Some EKV parameters can be found at http://legwww.epfl.ch/ekv/index.html.

Using BSIM3
BSIM3 can be used by stating “LEVEL=8” or “LEVEL=49” in the .MODEL statement. Both work.