

Introduction to PSPICE

PSPICE is a circuit analysis tool that allows the user to simulate a circuit and extract key voltages and currents. Information is entered into PSPICE via one of two methods; they are a typed 'Net List' or by designing a visual a schematic which is transformed into a netlist. In this class we will look at both the net list and the schematics. However, to to fully utilize the schematics we must first understand and become familiar with designing the net list.

Net List Format

The first line of the net list is the title line. This should contain pertinent information to the circuit and your name. The next lines are for circuit parameters – as many as needed. The next section is for output control statements. These will be discussed later. The file is closed with a <.END> statement. Below is the syntax for various elements. Each parameter name starts with a specific letter followed by a user-defined name (ie. R1, Cnew, Vout). The [] and the <> are not actually typed – they are for visual purposes only. Parameter components must be separated by spaces or tabs.

Parameter Syntax

Resistor:

R<name> [+ node] [- node] [value]

Capacitor:

C<name> [+ node] [- node] [value] [IC = <initial value>, optional]

Inductor

L<name> [+ node] [- node] [value] [IC = <initial value>, optional]

Independent Sources

I<name> [- node] [+ node] [value] [type] [transient spec] ;**Passive Load Convention!**

V<name> [+ node] [- node] [value] [type] [transient spec]

Dependant Sources

VCVS: E<name> [+ node] [- node] [+controlling node] [-controlling node] [gain]

CCCS: F<name> [+ node] [- node] [Vbranch] [gain]

VCCS: G<name> [+ node] [- node] [+controlling node] [-controlling node] [gain]

CCVS: H<name> [+ node] [- node] [Vbranch] [gain]

Example

Name - Lab # – Circuit Number (Header Line)

R1 0 1 1k ; 1000 ohm resistor named R1 from ground to node 1

V1 1 0 DC 0 ; Zero volt DC source from node 1 to ground

.END ; Required formal end statement

Line code after ' ; ' ignored by compiler and treated as a comment

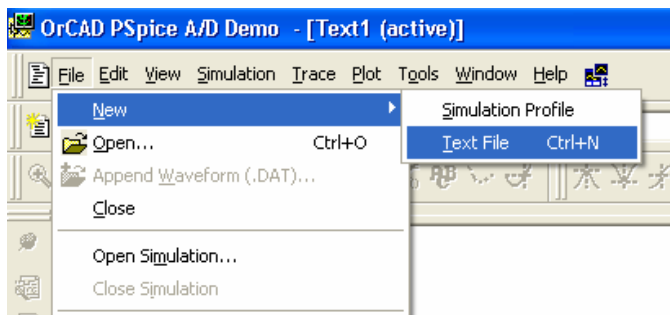
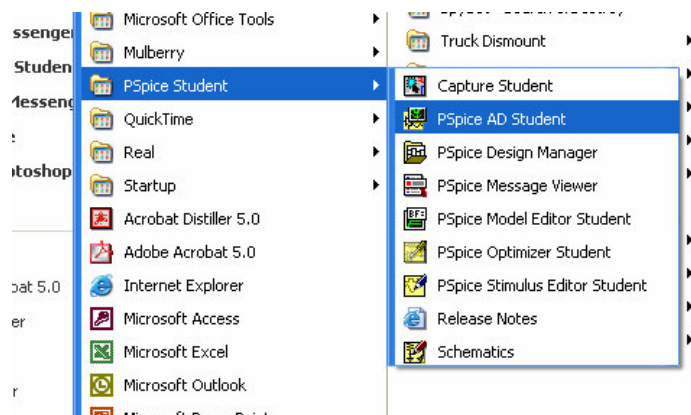
Parameter Suffixes

Tera	'T'	10^{12}
Giga	'G'	10^9
Mega	'Meg'	10^6
Kilo	'K'	10^3
Milli	'M'	10^{-3}
Micro	'U'	10^{-6}
Nano	'N'	10^{-9}
Pico	'P'	10^{-12}
Femto	'F'	10^{-15}

Writing a PSPICE Netlist Statement

Label every node on the circuit diagram. Start with ground (Labeled '0' by convention) and work clockwise around the circuit.

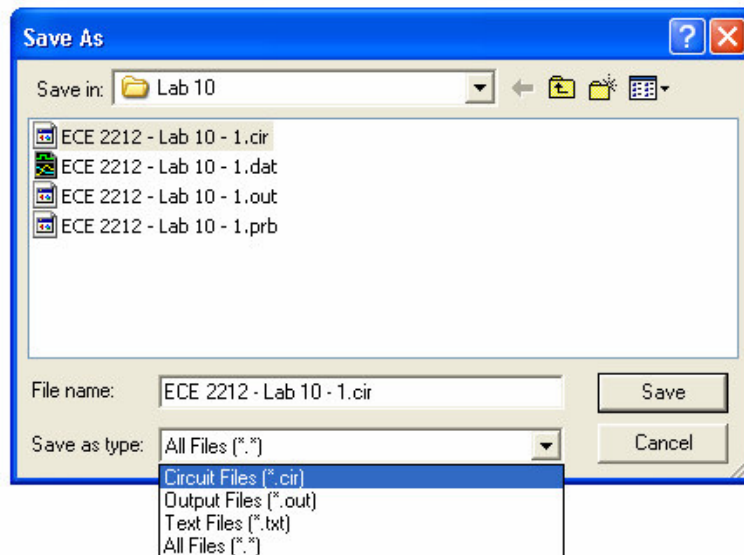
Open **PSPICE AD Student**



Goto **File -> New -> Text File** or
Click on the **New Icon** and then **New Text File**

Write your PSPICE circuit using above example and explanation or additional resources from textbooks or the internet. Make sure to have a label with a title, include all circuit elements and include a .END statement

Goto **File -> Save As** and save the file with a **.cir** circuit extension

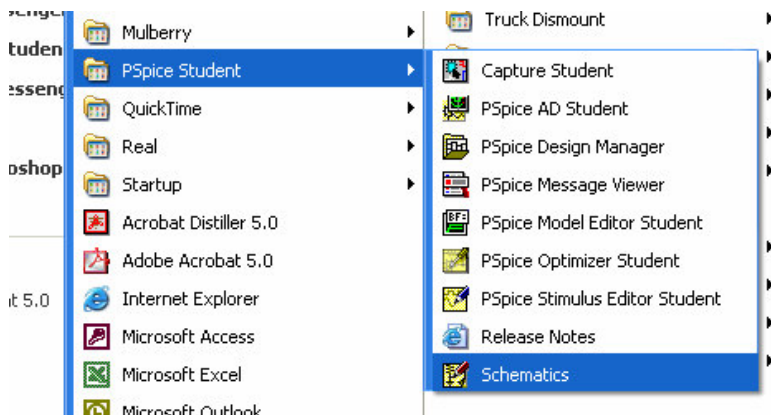


Goto **Simulation -> Run Filename**

If the option is not available you may need to close the file and reopen it.

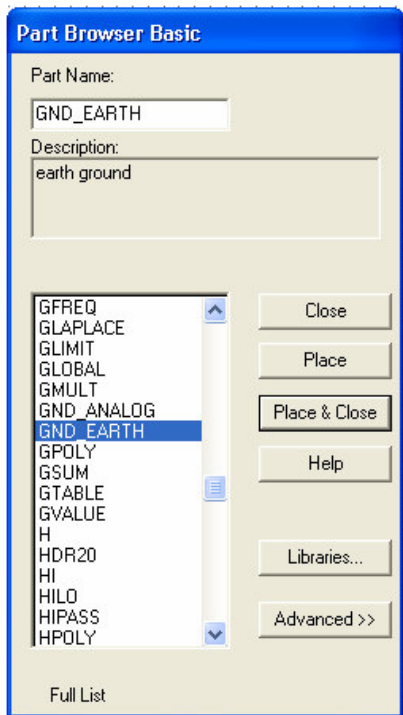
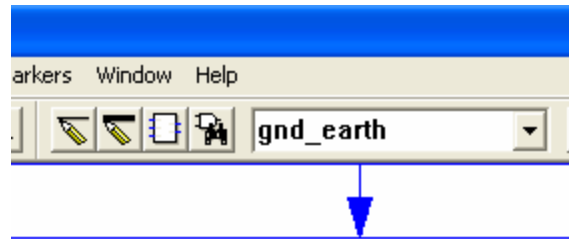
Once the program has run the output file may appear. If it doesn't you can go to **View -> Output File** or click on the **Output File Icon** on the left sidebar.

Writing a PSPICE Schematic



Open PSPICE Schematics

To place a part enter the name into the text box in the top button bar or **Click on the 'Get New Part' button .**



Select the desired part from the list.

Common Parts

GND_EARTH	Ground
R	Resistors
C	Capacitors
L	Inductors
VDC	DC Power Source
VAC	AC Power Source

Place Parts by left clicking the mouse button. Remove the part that trails the mouse pointer by right clicking.

Rotate Parts by pressing 'Ctrl-R' or double-right clicking.

Change Attribute Values such as voltage or resistance by double clicking the appropriate value on the placed part.

Connect Parts by Clicking on the 'Draw Wire' Button to the left of the 'Get New Part' Button. Make sure not to forget to include ground.

Set the Analysis Setup by going to Analysis -> Setup... For basic DC circuit analysis make sure 'Bias Point Detail' is checked.

Save the File as a schematic - filename.sch

Simulate by pressing F11 or clicking Analysis -> Simulate

The PSPICE AD Student Window should appear. **View the output** as done previously when creating a netlist.