

Introduction to Pspice Orcad Capture

Nik Din Muhamad
Jabatan ENCON
Fakulti Kejuruteraan Elektrik, UTM.
E-mel:nikd@fke.utm.my

About Pspice

SPICE, the acronym for a **S**imulation **P**rogram with **I**ntegrated **C**ircuit **E**mphasis, was developed at the University of California Berkeley. Pspice is a commercial version, developed by Microsim Corporation. In 1997 Pspice was bought by Orcad, Inc., then in 1998 Cadence bought Orcad. There are two schematic editor available: Capture and Schematic. The Capture is inherited from Orcad, whilst the Schematic is inherited from Microsim. Microsim's Schematic Editor is still available now and for the next few years until the Pspice's users become more familiar with Orcad Capture.

A general procedure to use Pspice Orcad Capture

Circuit simulation in Pspice is a four-step process:

1. Open and draw a circuit file under Orcad Capture
2. Simulate
3. Plot results using probe
4. Analyse the results

A general procedure to use Pspice can be given as follows:

1. *Open the Orcad Capture*

- Choose the menu option **Start:Programs:Pspice Student: Capture Student**
- In the Create New Project dialog box, tick *Create a Blank Project*.
- Click **OK**.

2. *Create a new Project or open an existing Project*

- Choose the menu option **File: New: Project**
Give your project a name in the dialog box.
- *Select **Analog or mixed A/D Wizard***
- Click **OK**

Or

- Choose the menu option **File: Open: Project**
- *Use the browse button in the dialog to locate the folder containing your Pspice files.*
- Click **OK**

3. *Load Libraries of Parts*

- *Choose the menu option **Place: Part: Add library***
We recommend adding all of the available part libraries.
- Click **OK**

4. *Choose the Parts for your schematic*

- Choose the menu option **Place: Part**
- In the Place Part dialog box, choose the specific part by selecting *Libraries* and *Part list*.
- Click **OK**.
- Locate the part on drawing with mouse, then *right click* on the mouse and select **End Mode**.
- To rotate a part, select part and type r.
- To flip a part vertically, select part and type v.
- To flip a part horizontally, select part and type h.

5. *Specify the attributes of the part*

- Double click on the value of the part
- In the Display Properties dialog box, change the value.

Or

- Double click on the part.
- In the Orcad Capture Property Editor, change the attribute values.

The Orcad Capture Property Editor is a spread sheet type, like Excell. We can add a new column, if needed.

A different part requires a different attribute values. Some parts (such as Vdc, R, L, and C) require only one attribute , usually the value of the part, that we must specify; and the other parts (such as VSIN, VPULSE and VPWL) require a few attribute values. Mastering this is an important thing to mastering Pspice. Appendix A gives a list for the frequent used parts and their attribute values.

The process of locating a part, placing it on the layout grid and setting the attribute values is repeated for each part in your circuit.

6. *Wire the schematic*

- Choose the menu option **Place: Wire**
- Locate wire on drawing with mouse, then *right click* on the mouse and select **End Mode**.

7. *Grounding the circuit*

The **Ground** is always required for any circuits that we want to simulate using Pspice.

- Choose the menu option **Place: Ground**
- In the Place Ground dialog box, choose Add Libraries. In the **Browse File** dialog box select **Source**, then click **Open**.
- In the **Place Ground** dialog box again, select *Source* in the Libraries and select **0**.

8. *Place Voltage and/or current probes (optional)*

- Choose the menu option **Pspice: Marker: Voltage level** if needed
- Choose the menu option **Pspice: Marker: Voltage differential** if needed
- Choose the menu option **Pspice: Marker: Current into pin** if needed

9. *Setup the simulation*

- Choose the menu option **Pspice: New Simulation Profile**

Give your simulation profile a name in the dialog box.

- In the simulation setting dialog box, select **Analysis Type**
The available analysis types are *Time domain (transient)*, *DC sweep*, *AC Sweep/Noise* and *Bias point*.
- Set Run to Time (TSTOP)

For a typical Time domain analysis, one to ten cycles of the input source is often required. (e.g. for 50 Hz source, Run to time (TSTOP) of 60 ms (three cycles) is enough)

10. Run simulation and display the results.

- Choose the menu option **Pspice: Run**
- The probe window will be prompted automatically. The results for the marked current/voltage probes will be displayed.
- In the Probe Window choose the menu option **Trace: Add trace** if needed.
- In the Probe Window choose the menu option **Trace: Eval goal function** if needed
- In the Probe Window choose the menu option **Plot: Add Plot to window** if needed
- In the Probe Window choose the menu option **Plot: Axis setting** if needed

11. Analyse the results

- Using the cursors. In the Probe Window, choose **Cursor: Display**. Then, you will be given access to two cursors. One of the cursors is controlled by the left mouse button and one is controlled by the right mouse button. A small window labelled Probe Cursor give the values at each of the cursors as well as the difference between them.
- Marking a specific voltage level. In the Probe Window, choose **Label: Mark**. This will label the specific point on the graph which is pointed to by the cursor that was last moved.
- Evaluating the goal function. In the Probe Window, choose **Trace: Eval goal function**. This will invoke the Evaluate Goal Function(s) dialog box. Select the specific function that you want to evaluate.

Appendix A

Attribute values for the parts that we must specify:

Vsin (VOFF AMPL FREQ PHASE(optional))
Vpulse (V1 V2 TD TR TF PW PER)
VPWL (T1 V1 T2 V2 T3 V3...)
L (value IC(optional))
C (value IC(optional))
VDC (value)
VAC(value)
R (value)