

Module

---

# Electrical Engineering 1

---

Degree programme:  
Bachelor Course Information Engineering

---

## Introduction to PSpice

---

# Content

<b>Content</b>	<b>2</b>
<b>Scope</b>	<b>3</b>
<b>1 Introduction</b>	<b>3</b>
<b>2 Links and Literature</b>	<b>4</b>
<b>3 Structure of Program</b>	<b>5</b>
<b>4 Creating Circuits in <i>Schematics</i></b>	<b>6</b>
<b>5 Simulation set up</b>	<b>12</b>
<b>6 Simulation</b>	<b>14</b>

## Scope

This manual is dedicated to our students to become familiar with PSpice, the classical CAE tool for the development and simulation of electronic circuits. This guideline supports you to solve the examples in the PSpice lab being part of the module Electrical Engineering.

The following chapters show how to handle the most important features of the *schematic* front end.

## 1 Introduction

The test and development of electronic circuits is supported by powerful computer tools. These tools help the developer to draw the schematics, to design the layout and to simulate the function of the electronic circuit even before a prototype has been build.

The university Berkeley has developed the first analogue simulation program with an to that date unreached performance and accuracy of simulating electronic circuits. In the 70s the so called program SPICE (Simulation Program with Integrated Circuit Emphasis) has been presented and is considered to be the breakthrough in the development of electronic circuits and resulted in a dramatic increase of efficiency.

While SPICE was designed for high-performance mainframe computers the company MicroSim ported the program to the upcoming PC technology in 1984. Since then this powerful program became the de-facto standard tool for electronic circuit development. There have been added powerful tools resulting in a circuit design environment called *Design Lab*. There are other programs belonging to this package like *Stimulus Editor* for generating waveforms for independent voltage and current sources or *PARTS* for creating PSpice models from a measured curve, which describes the behavior of elements. In 1998 the company MicroSim has been acquired by Orcad and finally Orcad has been acquired by Cadence Design Systems one year later.

Even though Cadence decided to discontinue the PSpice to streamline their product portfolio the software PSpice is highly adequate to become familiar with Computer Aided Engineering and contains all relevant functions.

The student version 9.1 is freeware and can be used to solve all tests within the course electrical engineering despite some limitations:

- Only one *schematic* page ( A-Size or A4 ).
- Maximum of 25 symbols placed on *schematics*.
- Configuration of maximal 9 symbol libraries.
- Maximal 20 symbols allowed in a user-created symbol library.
- Circuit simulation limited to circuits with up to 64 nodes, 10 transistors, 65 passive elements, two operational amplifiers or a combination thereof, and 10 ideal transmission lines with not more than four non-ideal line and four coupled lines.

## 2 Links and Literature

### Program

Even though Cadence discontinued the distribution of PSpice it floats around the web. Google for „PSpice 9.1“:

<http://www.google.de/search?hl=de&q=%22PSpice+9.1%22>

### Introduction to PSpice in the Web

There are a lot of resources in the web offering an introduction into PSpice, e.g.:

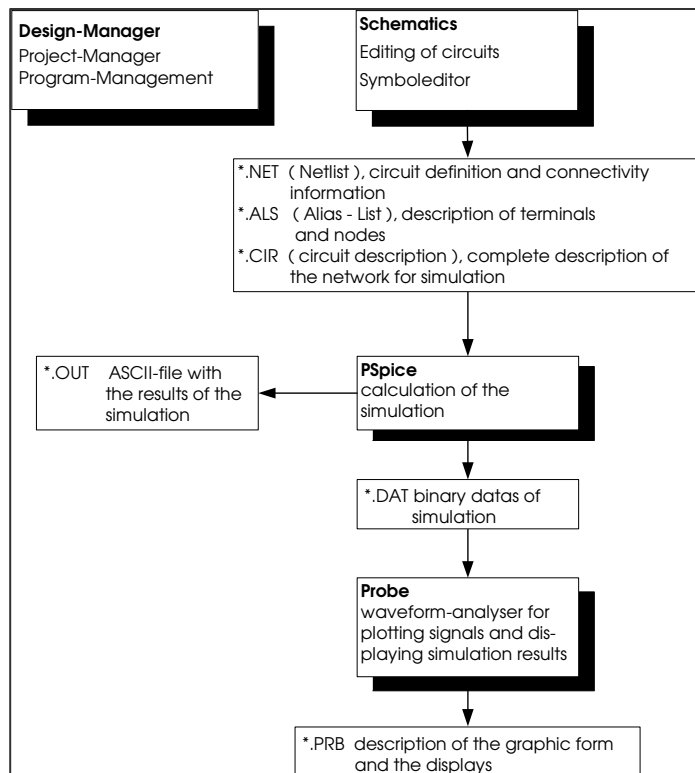
- Wolfgang Maichen  
Slide presentation of the most important topics with screen shots  
[http://www.pdamusician.com/testtechniques/tools/PSpice\\_Free\\_Student\\_Version/Introduction\\_to\\_PSpice.pdf](http://www.pdamusician.com/testtechniques/tools/PSpice_Free_Student_Version/Introduction_to_PSpice.pdf)
- University of Toledo  
Very short 8 page introduction with screen shots  
<http://www.eecs.utoledo.edu/~rking/circuits/lab/IntroToPSPICE.pdf>
- San José State University  
HTML online tutorial with screen shots and practical hints  
(written for version 8.0 but applicable)  
<http://www.engr.sjsu.edu/ee124/p-intro.html>

### English books

- Recommendation for beginners:  
Marc E. Herniter: “Schematic Capture with Cadence PSpice”  
Prentice Hall, 2.Edition 2003
- John Keown: “PSPICE and Circuit Analysis”  
Macmillan Publishing Company , 3. Edition 1998
- Franz Monssen: “OrCAD PSpice with Circuit Analysis”  
Verlag Prentice Hall , 3. Edition 2001

### 3 Structure of Program

Central part of the integrated package is the *design manager* which manages the different programs and the existing projects. The most important part is the graphical circuit editor *schematics* where you can draw circuits and from where you can start the simulator *pspice* and after simulation the program *probe* to draw the simulation results.



#### PSpice libraries

- **Model libraries**

They have the extension *\*.lib* and describe the electrical characteristics of the used elements and subcircuits. In the evaluation version all model libraries together are stored in the file *nom.lib*. New created model libraries will be configured by opening the pull-down menu → **Analysis/Library** and **Includes Files/Add Library**.

- **Symbol libraries**

They have the extension *\*.slib* and contain the graphic description of the symbols. In the evaluation version you get 11 symbol libraries with limited number of elements. Configuring of maximal 9 symbol libraries happens by selecting the menu **Options/Editor Configuration/Library Settings/Add Library**.

New symbol libraries will be created by clicking on the menu **File/Edit Library** or by clicking on the symbol to be editing and adding the command **Edit/Symbol**.

Necessary for the simulation are only the model library and the symbol library.

- **Package libraries**

They have the extension *\*.plib* and contain the values, which are necessary for creating board lay-outs with the program *PCBoard*, which belongs to the program package. In the lab the definition of this library is not necessary.

## 4 Creating Circuits in *Schematics*

### General setup

---

- Create the directories **C:\pspice** and **C:\pspice\lib** on your local hard disk or in your home directory on the net.
- Store the new created circuits under *schematics* in **C:\pspice** and new model- and symbol libraries in **C:\pspice\lib** .
- If you get a blue screen after starting *schematics* then there is a mistake in your file *msim\_ev.ini* .

Therefore copy the file *msim\_ev.ini* from the directory  
**G:\public\msim\_ev80\snapshot\** in the directory **H:\msim\_ev80\**

### Definitions

---

The following symbols will be used to describe the user's action:

- general sign to show different explanations, e.g. to show how to use the mouse
  - symbol for a command, e.g. for selecting the pull-down-menu **Edit / Model** including additional submenus
- a dialog box or a symbol appears, e.g. it appears **Edit Instance Model (Text)** and it can be opened by clicking
- ⇒ you have to fill in and to click on **OK**. to save the new attributes.

### Using the mouse

---

- move the cursor with the mouse on an object and click on the **left** button, to select it
- click the **right** mouse button to stop a function, e.g. the drawing of wires or placing parts
- double click the **right** button to repeat the function, then you can draw e.g. a new wire

### Placing Parts

---

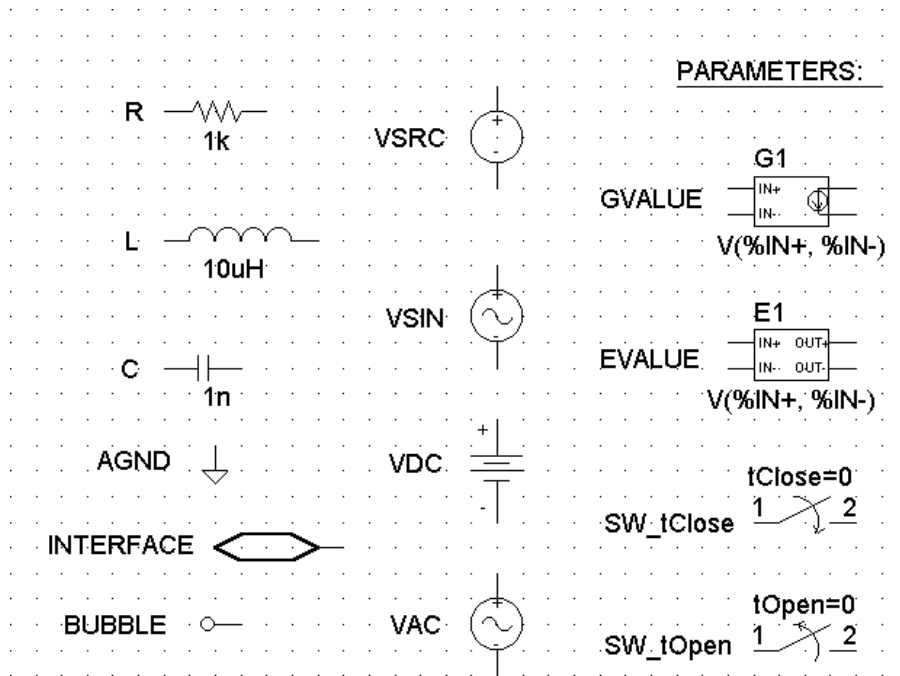
- draw under *schematics* all components you need
  - select In the menu **Draw / Get new part / Libraries**
- ⇒ searching for new parts in the symbol libraries, e.g.
- analog.slb
  - source.slb
  - port.slb

#### Note:

It is possible to open directly the **Part Browser** dialog box with the short cut *Str+G* and/or to place the name of the part you want, e.g. R (for resistance)

- place the selected part with the **left** mouse button and stop placing with the **right** button
- back to the menu and repeat the different steps until you have placed all necessary parts

**Important graphical symbols**



Symbol	Part	Library
R, L, C	Ohmic resistance, ideal inductance and ideal capacitance.	<i>analog.slb</i>
SW_tClose, SW_tOpen	Switches to open or close at defined times Attributes: Rclosed : "on" resistance Ropen : "off" resistance ttran : time of transition tClose , tOpen : moment of closing, opening	<i>eval.slb</i>
VAC	AC voltage source for AC sweep analysis Attributes: AC: RMS value of the voltage in the unit Volt	<i>source.slb</i>
VDC	DC voltage source for DC sweep analysis Attributes: DC: value of the voltage in the unit Volt	<i>source.slb</i>
VSIN	Sinusoidal voltage source with damping ratio for transient analysis generates a voltage with the following function: $u(t) = voff + v ampl \cdot e^{-df \cdot t} \cdot \sin(\omega(t - td) + phase)$ Attributes: voff: offset of the voltage in the unit Volt v ampl: amplitude of the alternating voltage in V td: time delay in the unit second df: damping ratio phase: phase angle in the unit grad	<i>source.slb</i>
VSRC	Voltage source for DC- , AC – or transient analysis Attributes: DC: value of the voltage in the unit Volt AC: RMS value of the voltage in the unit Volt tran: definition of the transient function e.g. tran = sin ( 1V 2V 1kHz 2ms 500 90 ) (units are not necessary) This will generate a voltage characteristic as the source VSIN. The assigned values must be filled in the same sequence as for the attributes of VSIN.	<i>source.slb</i>
IAC, IDC, ISIN, ISRC	as before but ideal independent <b>current</b> sources	<i>source.slb</i>
PARAM	Pseudocomponent up to three parameters can be defined fill in the attributes box: Name1... Name3 : name of a parameter Value1 ... Value3 : assigned value	<i>spezial.slb</i>
EVALUE , GVALUE	Controlled voltage and current source Attributes: The output source gets a value, which is a combination of arithmetic operators and input designators ( V%+ , V%- ) defined in the expression attributes	<i>abm.slb</i>
Interface	Interface port, provides connection between the pins of a hierarchical block or symbol and the underlying schematic	
Bubble	Global port, can be connected to another labeled port of the same name without using wires (easier to read)	



## Moving parts

---

Click the **left** mouse button on the part until it becomes highlighted in red, press and hold it. Without releasing the button move the mouse to move the part graphic to the desired spot in the schematic.

## Rotating and flipping parts

---

Click the **left** mouse button on the part until it becomes highlighted in red, then press:

<Strg><R> to rotate 90 degrees

<Strg><F> to flip at a vertical axis

## Deleting parts and circuits

---

- click the **left** mouse button on the part until it becomes highlighted in red, press the **Delete** key, or select **Edit / Delete**

or:

- **Cut** deletes the part and stores it in the clipboard where you can select it again with the command **Paste**

or

- press and hold the **left** mouse button and draw a box around a part graphic or a complete circuit. Everything inside this box will be selected and appears in red when you release the mouse button. With **Delete** or **Cut** you can delete it.

## Connecting parts

---

- select the menu item **Draw / Wire** or use the Shortcut: Ctrl + W  
→ It appears a pencil and the parts can be now connected
- move the pencil to the terminal, press the **left** mouse button.  
→ A dashed line appears
- move the pencil to the terminal of another part and click the **left** mouse button to make the connection
- to stop drawing click the **right** mouse button
- connect all parts until you get the whole circuit

### Note:

Your schematic may have bits of garbage floating around due to the editing process. To get a fresh copy select:

- **View / Fit**  
→ the whole drawing will be redrawn.
- **View / Redraw**  
→ the selected area will be redrawn.

**Note:**

The terminals of the parts and the wires should not be drawn overlapped. In this case a dot will be drawn. Sometimes you will get a message during creation of the netlist.

**Editing parts**

- double click the left mouse button on the part graphic ( e.g. VDC )
- the dialog box below appears
- ⇒ click on the parameter to be changed ( e.g. DC = )
- ⇒ change the default value to the new value

**Note:** **No** space between value and unit !

⇒ click on **save Attr** and the changed value will be stored

- double click the left mouse button on **change display**.
  - ⇒ select an option
  - ⇒ press **OK**
- ⇒ press **OK**

or:

- double click the left mouse button on the default value in the circuit.
- The dialog box appears
- ⇒ change the value
- ⇒ press **OK**

**Note:** Use the following short forms for the SI prefix (either as lower or upper case):

Shortcut	Meaning	pwr
f	Femto	$10^{-15}$
p	Pico	$10^{-12}$
n	Nano	$10^{-9}$
u	Micro	$10^{-6}$
m	Milli	$10^{-3}$
k	Kilo	$10^3$
meg	Mega	$10^6$
g	Giga	$10^9$

## Defining parameters

---

- select the menu item: **Draw / Get new part / Libraries**
  - ⇒ select the library *spezial.slb*
  - ⇒ click on the part *PARAM*
  - It appears the symbol PARAMETERS
  
- Double click on the symbol or select the menu **Edit / Attributes**
  - ⇒ edit the field **Name 1** fill in e.g. „*Rvar*“.
  - ⇒ press **OK**
  - ⇒ Under **Value 1** fill in e.g. „*1*“.
  - ⇒ press **OK**
  
- Double click on the part e.g. *R1*
  - ⇒ edit the field **Value** fill in : *{Rvar}*
  - ⇒ press **Save Attr** or press the **Enter** key
  - ⇒ press **OK**
  - *{Rvar}* appears below the resistor *R1*.

During simulation the value of R1 will be replaced by the values of the sweep range, which is defined in the DC sweep analysis setup (select the range for the **Global parameter Rvar**).

## Drawing text

---

- select the menu item **Draw / Text**
  - ⇒ Type the text directly.
  - ⇒ press **OK**
- or:
- select the menu item **Draw / Text Box**
  - ⇒ press and hold the **left** mouse button and move the mouse down and to the right
  - ⇒ a black box appears where you can write the additional text  
By pressing the mouse button the text box can be moved or placed.

## Ground

---

**Note:** Each circuit must have at last one ground connection

- select the menu **Draw / Get new part / Libraries**
  - ⇒ select *port.slb* and the part **AGND** ( Analog Ground ).
  - ⇒ click on **Place & Close** to accept the part
  - ⇒ move the symbol **AGND** to a suitable place in the circuit
  - ⇒ click the **right** mouse button to terminate drawing ground

## 5 Simulation set up

Before a simulation runs the following settings have to be made.

### Set markers

---

- **Set Attribute Value**  
This dialog box will be opened if you double click on a wire. This gives the possibility to label the connection line and generates for this wire a voltage with the name  $V(\text{label})$  and you can find it after simulation in the graphic analyzer *probe*. This is easier to read than the standard name of a voltage which is named according to the next terminator e.g.  $V(R2:2)$ .
- **Menu Markers / Mark Voltage/Level**  
Placing this marker at the wire automatically runs probe after simulation and displays the marked voltage between the marked point and analog ground.
- **Menu Markers / Mark Voltage Differential**  
Placing this marker at the wire automatically runs probe after simulation and displays the marked voltage between the marked plus and minus points.
- **Menu Markers / Mark Current into Pin**  
Placing this marker at a terminal of a component automatically runs probe after simulation and displays the marked current into the marked terminal.

### Set type of analysis

---

Selecting the menu **Analysis / Setup** allows to select the different simulations.

- Enable the required analysis by ticking the box and click the assigned button

The following types of analysis are of practical use in our laboratory:

- **AC Sweep**  
Analyze circuits with multiple frequencies in steady state.
- **DC Sweep**  
Changing DC-voltages or currents within a specified range.
- **Transient**  
Used to look for waveforms versus time and requires a time-domain voltage or current stimulus.
- **Parametric**  
In conjunction with other analyses Parametric analysis performs multiple iterations of a specified standard analysis while varying global parameter, model parameter or operational temperature. It generates a family of curves.  
  
(Similar is Nested Sweep within DC Sweep . But the plotting of the produced traces is different.)
- **Bias Point Detail**  
By default, the Bias Point Detail analysis is set to run automatically. It calculates e.g. the transistor operating point corresponding to the basic adjustment of the sources and components and also the start values for a following transient analysis.

## Print Step and Step Ceiling

---

When *PSpice* simulates a circuit it chooses the time between simulation points to be as large as possible. The larger the time step the less time the simulation takes to complete. By default the maximum of the time between simulation points is the value *final time/50*. In some cases *PSpice* chooses large time steps and the resulting graph constructed by these points may not look aesthetically pleasing.

For example a sinusoidal curve will be displayed only by several points and the plot shows linear connections between them. To increase the number of points *PSpice* provides the argument *Step Ceiling* for the transient analysis.

**Note:** The value of *Step Ceiling* is the maximum time between simulation points.

Reducing *Step Ceiling* increases the number of simulation points. To keep simulation time as short as possible leave the *Step Ceiling* value blank.

The argument *Print Step* is used for printing text output with the print part. After the time interval defined by *Print Step* in seconds the specified values will be sent to the the output file \*.out if in *schematics* the pseudocomponents *VPRINT* and *IPRINT*, or *VPLOT* and *IPLOT* (library *port.slb*) are placed. Additionally it is used to plot traces in *probe*. Since the simulation results are computed at a different time a 2<sup>nd</sup> – order polynomial interpolation is used to obtain the printed or plotted values.

The value *Step Ceiling* has a second meaning for the simulation. If *PSpice* exceeds the limit of reducing the time step the simulation fails to converge and stops. Then the *PSpice* window displays the message “Unable to finish transient Analysis - See output File for Details”.

The accuracy of 12 significant digits limits the minimal allowable step size which can not be less *Final Time/10<sup>12</sup>*. You can choose how often the time step can be reduced in the menu **Analysis / Setup / Options** .

Smaller values of *Step Ceiling* avoid too many automatic tries in reducing the time step. The *Step Ceiling* is the largest value for the time step. Since the step size can start with a small value of *Step Ceiling*, after a number of reductions within the limit the step size will become small enough to make the error acceptable.

Sometimes a second method to solve convergence problems is to increase the number of time step reductions. But this increases the simulation time.

## 6 Simulation

The following menu items are import for the simulation and the display of the results.

### Analysis

---

The simulation starts by selecting the menu Analysis. There are different submenus:

- **Analysis / Electrical Rule Check**  
checks the connections in the existing circuit. In case of errors a error list can be opened by the key < F10 >
- **Analysis / Create Netlist**  
creates a netlist, suitable for simulation
- **Simulate**  
starts the „Rule Check“, creates the netlist and starts the simulation

Depending on the default setting of the menu item **Analysis / Probe Setup Options** the graphic analyzer *probe* starts immediately after simulation. It shows an empty diagram with the selected variable shown on the the x-axis.

The following menu items are required to adjust the display to show the relevant data. All menu items are set in the probe window which is generated with the simulation run.

### Trace / Add

---

- Signals can be added to the active window. After selecting the menu *Add Traces* the currents through all components and all voltages versus the placed AGND are shown. By clicking the left mouse button the signals will be copied into the trace command text field *Trace Expression*.
- Voltages across simple components or parts of circuits can be constructed as difference between voltages in the line *Trace Expression*.
- You can use mathematical expressions in *Trace Expression*, e.g. the product of a voltage with a current to calculate the power function.

In the field *Functions or Macros* you find mathematical expressions and functions, e.g.:

R(x) = Real part of x

IMG (x) = Imaginary part of x

AVG (x) = moving average of x over the range of the x-axis variable

AVGX (x, d) = moving average of x from x-axis-value(x) minus the constant value d

### Plot / X Axis and Plot / Y Axis

---

- These menus provide the settings of the x-axis and y-axis. The limits of the axes can be set automatically or manually by pressing the button *User Defined*.
- In the menu **Plot / X Axis** and the submenu *X Axis Settings* the variable shown on the X axis can be changed.

### Menu Plot / Add Y Axis

---

- Selecting this menu you can add a second Y axis with a different scaling from the first one.

### Menu Add Plot

---

- A second plot is generated and displayed simultaneously.

### Tools / Cursor

---

- An active cursor can be used to obtain numerical values from traces. With the right mouse button you can move a second cursor. The dialog box displays the coordinates of each cursor and difference between the cursor values.

### Tools / Label

---

- The cursors are used to find numerical values. Once important points are found, the coordinates of those points can be placed on the plot. To label these coordinates – only in case of a active cursor - select the menu **Tools / Label / Mark**.

### Tools / Display Control

---

- After closing *probe* the last display with all traces, scale ranges and labeled points will be stored. When selecting this menu at a later time a dialog box with the name *Save/Restore Display* appears. By clicking on *Last Session* and *Restore* the last display with all curves will be created. This is helpful to get a fast display reconstruction after a new simulation with changed parameters.

### File / Log Command and File / Run Command

---

- Selecting the menu *File/Log Command* allows to record all changes you make to the display settings. Later the commands can be restored by selecting the menu item **File / Run Command**.