

## Exercise 2.

# PCB Design

### Aim of the measurement

- Introducing to the PCB design
- Creating a schematic of an analog circuit, making simulations on it and designing a Printed circuit board for it.

### Keywords

Printed circuit board, Schematic, OrCAD

### References

[http://en.wikipedia.org/wiki/Printed\\_circuit\\_board](http://en.wikipedia.org/wiki/Printed_circuit_board)

<http://www.ee.siue.edu/faculty/adgrego/classes/ece326/PSPICEtutorial.pdf>

<http://www.ece.rutgers.edu/~ece428/PCBs/Gramiccioni,%20T.%20-%20A%20Tutorial%20on%20OrCAD%20Layout%20and%20Printed%20Circuit%20Board%20Fabrication.pdf>

<http://www.pcb.electrosoft-engineering.com/04-articles-custom-system-design-and-pcb/01-printed-circuit-board-concepts/printed-circuit-board-pcb-concepts.html>

[http://server.ibfriedrich.com/wiki/ibfwikien/images/d/da/PCB\\_Layout\\_Tutorial\\_e.pdf](http://server.ibfriedrich.com/wiki/ibfwikien/images/d/da/PCB_Layout_Tutorial_e.pdf)

### Measurement instruments:

PC with a Demo version of OrCAD 9.1

### Important terms

#### PCB

PCB is a thin (usually 0,5-3 mm) board made of fiberglass, composite epoxy, or other laminate material. Conductive pathways are etched or printed onto board, connecting different electronic components on the PCB. There are single sided (one copper layer), double sided (two copper layers) or multi-layer up to 10+ layers PCBs. Beside the conductive layer most PCBs contains solder mask layers (a thin lacquer-like layer of polymer that is usually applied to prevent solder bridges between closely spaced solder pads) and silk screen layers (displaying text information for soldering or application).

## Circuit Schematic or Schematic

The Schematic is the representation of the electronic system using abstract, graphic symbols. Drawing a Schematic is always the first step of designing a PCB. Usually PCB design softwares contain separate tools for Schematic and PCB design. Often a PCB is represented by multiple Schematic pages separated by functions, like power domain, analog signal domain, inputs/outputs.

## Components

In a PCB designing system a component is described by two different representation. The first is a Schematic symbol, an abstract representation of the inputs and outputs of the component. The second is the a Layout part which contains information about the component's mechanical size and the required copper footprint to solder the component onto the PCB. In many PCB design software more than one Schematic symbol can be connected to one Layout part. For example an integrated circuit represented by a Layout part can contain 8 pieces of inverters, which can be used as individual Schematic symbols. Usually there is also an opportunity to use a Schematic symbol with multiple Layout representation. For example the integrated IC used in the previous example can have different footprints (a surface mount version and a through-hole mounted version) described by its data sheet.

## Footprint

The Footprint is a pattern and space on a PCB taken up by a component. Every Integrated circuits has one or more packaging type. Data sheets describes the possible packages with their dimensions, and recommended PCB footprint dimensions. Usually PCB design softwares contain a huge library with hundreds of footprints representing the PCB layout of the most widespread standard packages.

## Netlist

Many PCB design software use a different component for PCB and Schematic design. The netlist connect these two software components describing the used circuits and their interconnects.

## PCB Layers

Most of the PCBs are multilayer ones. Sort description of the most widespread layers:

*Mechanical layers:* There can be multiple mechanical layers containing the mechanical dimensions of the PCB board and/or the dimensions of the IC packages. Mechanical layers can contain information about the drillings used on the PCB for through hole packages or for inter layer connections: VIAs (VIA is a conductive drill hole connecting two routing layers).

*Routing layers:* Routing layers are used for representing component footprints and interconnects. Routing layers can be located on the top or bottom of the PCB or also can be an internal layer (without footprints of course).

*Ground Planes & Power Planes:* Ground planes and power planes (in 4+ layer PCBs) are solid copper layers shorted to a fixed potential. These layers are very important for power distribution and solid grounding.

*Silkscreen Layers:* Used for displaying human readable information to help the soldering process or application of the PCB. Can be used on both the top layer and on the bottom layer.

*Solder Mask Layers:* Solder masks are thin lacquer-like layers of polymer those are usually applied to prevent solder bridges between closely spaced solder pads. Solder mask layers are used for SMD (surface mount) components, where the soldering is not manually performed.

## Back annotation

Back annotation is used to propagate back the changes made on the PCB to the Schematic. For example if a component's name is changed on the PCB design to sort the components and help the soldering process, then the result must be propagated back to the Schematic to maintain the integrity of the design.

### **Gerber file**

The Gerber format is an vector image file format which is the de-facto standard used by printed circuit board (PCB) industry to describe the printed circuit board images.

### **PCB design rules**

There are many rules and advices to design a good PCB. During the design it is also important to keep the rules given by the future manufacturer of the PCB. Some practices are given below:

*Check the precision limits of the selected manufacturer:* keep the minimum routing width (usually 6 mils, but there are manufacturers with better precision), minimum distance between routes etc. In most of the PCB design softwares there are a design rule tool to check these limits.

*Use wider routs for power supply and ground:* routing width effects the resistance of the track. Therefore tracks conducting significant amount of current should be wider to minimalise the voltage drop.

*Do not use 90 degree wire breaks:* rout wires with obtuse angles (more than 135 degrees) to avoid reflections and manufacturing problems.

*Regularize the drill table:* PCB designed by using standard library footprints can have many different types of drill holes. These drill holes many times differs only in 1 or 2 mils, therefore they should be aggregated which lowers the manufacturing time and cost.

*Solder mask:* keeping outs should be 2-4 mil larger than the copper pads, because of the manufacturing process.

*Board outline:* Do not place wires or holes very near to the board outline, because it can be damaged during the cutting process.

*Drill Holes:* Always use a bigger hole than the diameter of the pin, because the hole galvanization process will result in a smaller hole. Data sheets usually contains the recommended drill hole size.

*Routing:* For a two layer PCB try to rout horizontally on one of the layers and vertically on the other side.

## **Laboratory exercises**

During this laboratory exercise a schematic will be created for the inverting amplifier examined at measurement 1. Using this schematic a time domain and a frequency domain simulations will be performed. After the successful simulations a PCB will be designed for the circuit. Before the laboratory exercises read

[http://en.wikipedia.org/wiki/Printed\\_circuit\\_board](http://en.wikipedia.org/wiki/Printed_circuit_board)

as a background.

### **1. Creating a schematic in OrCAD Capture CIS**

A first step of creating a PCB is to draw circuit schematics. Start the Capture CIS program from the desktop.

As a first step create a new project: File->New->Project. Choose the Analog or Mixed-Signal Circuit Wizard option (Figure 1.).

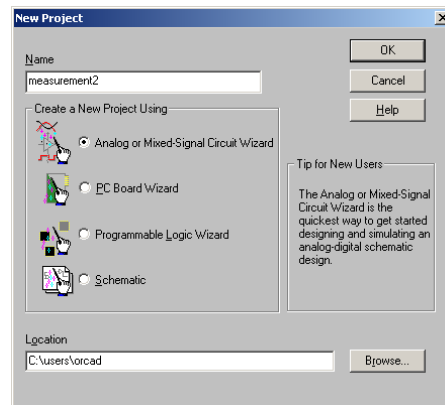


Figure 1. Creating a new project

After creating a project two file is generated: .opj file witch contains the project setting, and a .dsn file containing the schematic circuit and its properties.

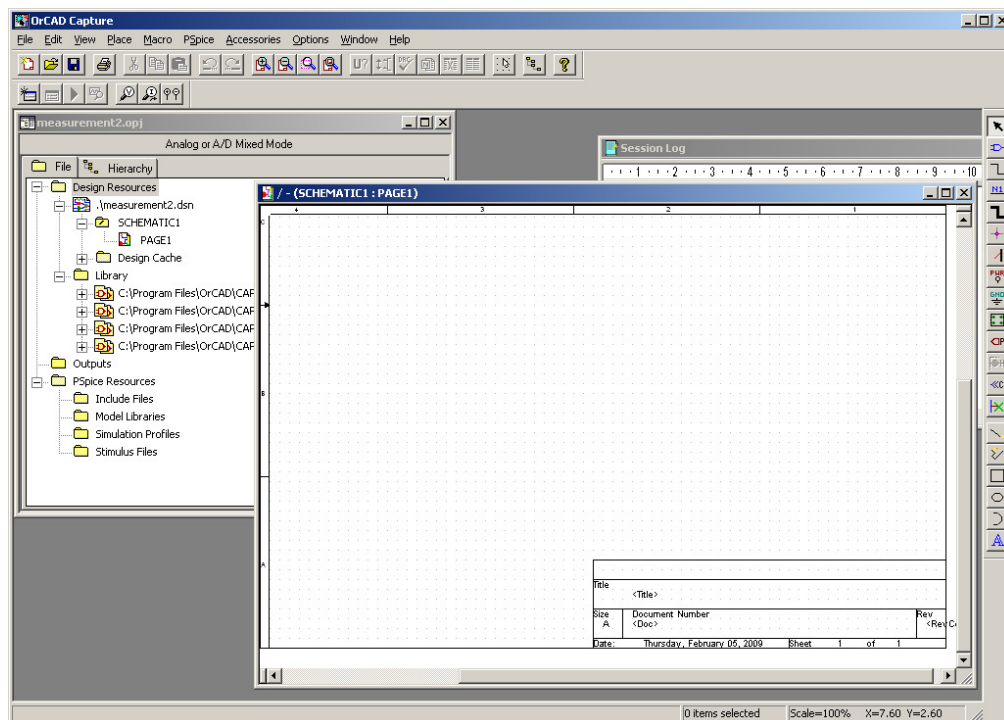


Figure 2. A new project in OrCAD

In a new project there are two main windows. On the left corner there is the project window containing the files of the project. In this window under the .dsn file the schematic pages of the project can be seen (a project can contain multiple schematic pages). This window also contains some information about simulation setups and part libraries.

The other page is a schematic page. The schematic of the inverting amplifier will be drawn here. First exercise is to draw the schematic shown on figure 3.

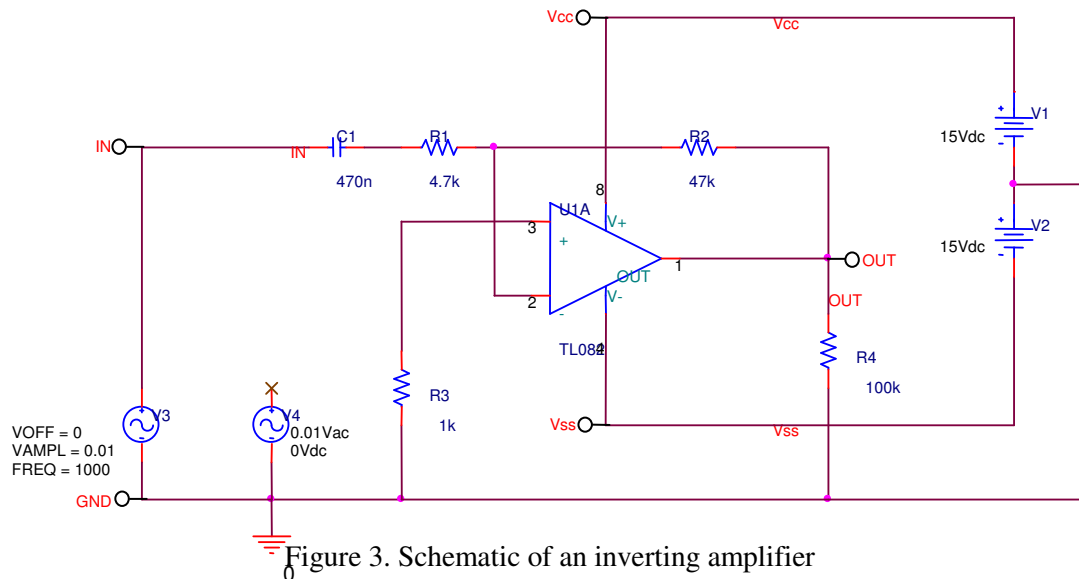



Figure 3. Schematic of an inverting amplifier

To draw the schematic above, circuits should be added to the page. Use the Place->Part menu or the  icon at the right toolbar to do this. Selecting the part to place is done using the pop up menu.

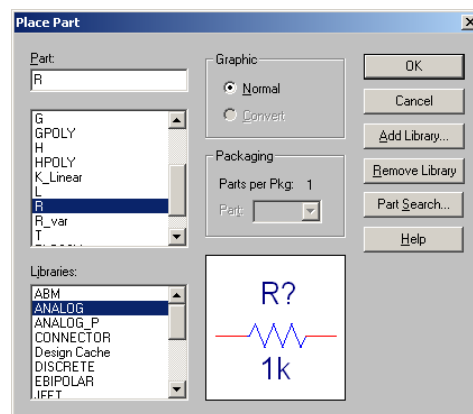





Figure 4. Selecting a part to place

The selected part can be placed into the schematic page. It can be rotated (key r), mirrored (by using the right mouse button pop up menu) etc. The place part command can be aborted by hitting the Esc key. Wires can be added to the schematic page by using Place->Wire menu or  icon on the right toolbar. Instead of drawing very long wires a Place->Netalias () tool can be used to logically connect two wire fragment (wires with the same net alias are considered as connected). The net alias is also very useful to help drawing simulation results, so place a name IN to the input wire and OUT to the output wire of the operation amplifier.

Place and connect all the parts shown on Figure 3. Not connected parts should be terminated by using  icon on the right toolbar.

Ground references can be added to the project by using Place->Ground or the appropriate icon on the toolbar. Always use the 0/Source ground, other ground icons could cause problems during simulations.

The last step is to set the proper value of the schematic circuit parts. It can be done by double clicking on the values or by selecting the part and using the “Edit Properties” option from the right mouse click pop up menu.

## 2. Making simulations

Simulation setups can be created by using the PSpice->New Simulation Profil menu (note that the .dsn file should be selected on the left project window to access this menu).

As a first step lets create a frequency domain simulation (Make sure that the Vac source is connected to the operational amplifier as an input). Use the PSpice->New Simulation Profil menu. Give a name to the simulation and set the parameters shown on Figure 5.

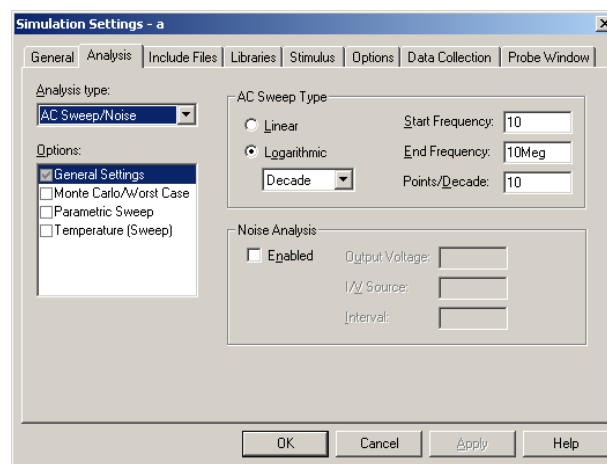


Figure 5. Frequency domain simulation

Start the simulation by using the PSpice->Run menu or the play button on the toolbar. A new empty result window will be opened Figure 6.

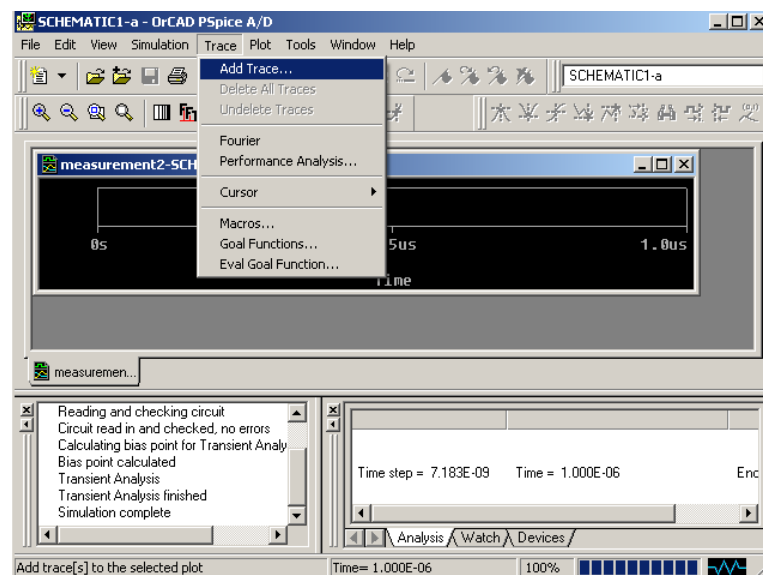


Figure 6. Empty simulation result.

Use the Trace->Add Trace to display results. Draw the amplitude and phase diagram use  $DB(V(OUT)/V(IN))$  for the amplitude and  $P(V(OUT)/V(IN))$  for the phase.

As a second step create a time domain simulation and draw the V(OUT) and V(IN) signals.

### 3. Creating a netlist for the Layout editor

After checking the schematic with simulation, time to create a PCB for the circuit. First step of PCB creation is adding footprints to schematic parts. Footprints can be look up from the Library manager of OrCAD Layout. Start OrCAD Layout Demo and use the Tools->Library manager menu. Footprints can be selected in this manager, and the name of the footprint can be simply copied out from here (Figure 7).

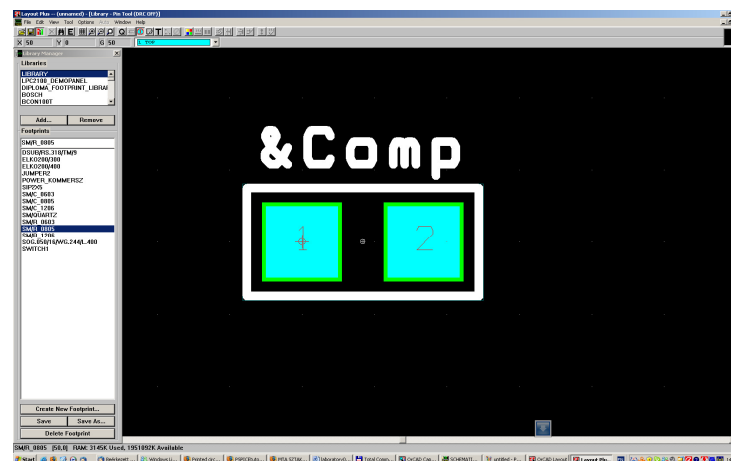


Figure 7. Library manager.

Selected footprints can be added to the schematic parts by using the part properties menu. Select the whole design and by using the right mouse click select the “Edit properties” options from the pop up menu (use the Parts sub title). In the properties window the PCB Footprint property contains the real physical footprint of the schematic part (Figure 8.).

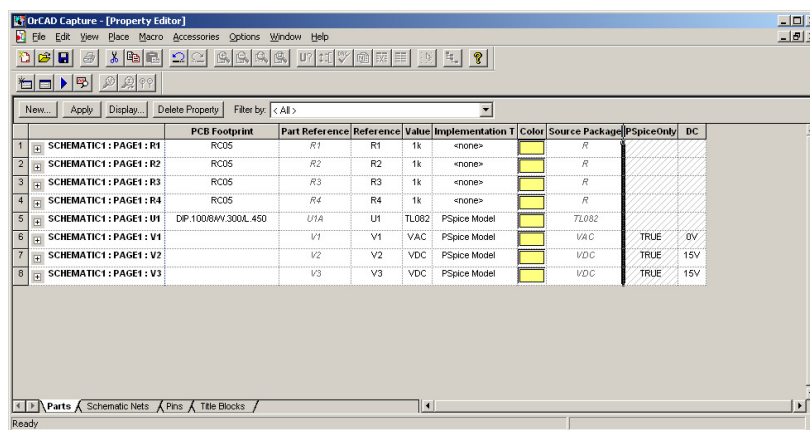


Figure 8. Adding PCB footprint properties.

Use the following footprints: SM/R\_0805 for resistors, SM/C\_0805 for capacitors, and DIP.100/8/W.300/L.450 for the TL082 operation amplifier. Add connectors to the schematics for the Input, Output and for the Vcc, Vss, and GND. Add footprints to this connectors too. Parts only represented in the simulations has a PSpiceOnly TRUE property. Make this property TRUE for the load resistor of the operating amplifier.


After adding all the footprints use the Tools->Create Netlist menu and create a Layout netlist. This will create a netlist file witch make connection between the schematic editor and the Layout designer.


#### 4. PCB Design

Start the Layout Plus Demo. New project is created by using File->New menu. A new project is asking for the default configuration of the PCB to be designed. This default configuration contains default wire wide parameters default VIA drills, default system settings etc. Select the DEFAULT.TCH from Program File\OrCAD\Layout\_Plus\Data. After selecting the default configuration select the .mnl file created by the schematic editor. The next is a .max file, which is the new generated PCB design.

The Layout designer will start with all of the footprint placed onto the design sheet.

As a next step select the system parameters by using Options->System Settings. The Display unit: mils, a Visible grid 100, other parameters 25.

Place the components on the design sheet in a way to help the routing process (yellow lines indicate which components should be connected). For component placement use the  icon from the toolbar or the Tools->Components->Select Tool from the main menu.

Connect the components by using the  icon from the toolbar. Try to make an optimal routing. When connecting Power or GND nets use wider wires comparing to the signal nets. A possible solutions is shown on

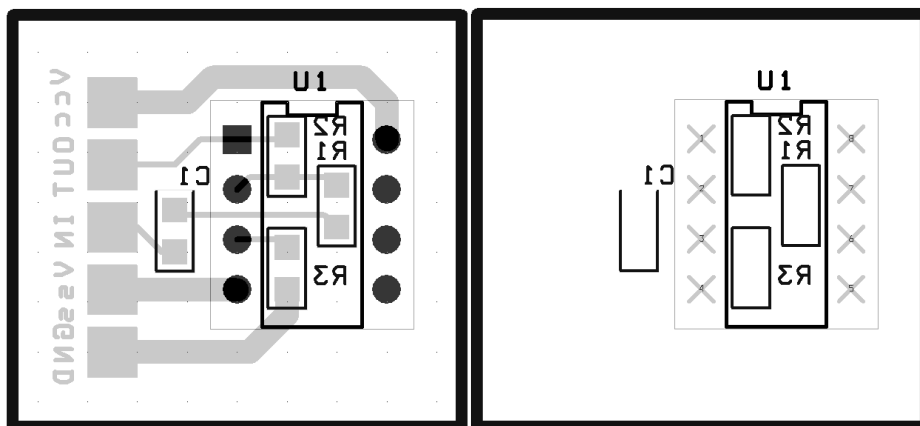


Figure 9. An example layout configuration.

After creating a suitable layout the next step is to prepare the PCB for manufacturing. The manufacturers use the so called Gerber files in their process. To creation of the Gerber files starts by the specification of the used layers. Usually for this design requires the following layers TOP, BOT (bottom), DRD (drill drawing) and SMB (soldermask bottom), SMT (soldermask top). The layers to be used for Gerber file creation can be specified at Options->Post Process Settings menu. After the specification, the Gerber file generation is a straight forward automatic process, which can be started by using the Auto->Run Post Process menu.