Advanced Skills in EAGLE (Intermediate)

Brief:

The participants will be introduced to one of the most important skills that is used in the world of digital fabrication which is the PCB designing. The tool that is used is the professional EAGLE CAD soft that is used in designing PCBs. EAGLE is the most used cad soft in this field. This workshop will give the participants the basic skills in using the application, and then it will advance into more advanced skills that will help them to detect errors and take their PCB to the fabrication stage. The participants will be able to develop full double sided PCB that is most used in the world of fabrication.

Duration:

2 days

- each day with (*3 hours* session)

Workshop Agenda:

Duration (Minutes)	Activity	Remarks
	Day 1	
10	Agenda review Safety Instructions.	▲ Revising the agenda. ▲ To make sure no one runs any machine without the permission of the Fab lab facilitator.
10	Introducing the participants to each other	▲ Breaking the ice between participants
40	Introduction to the software	▲ Installation process. ▲ Discover the control panel of the software. ▲ Getting familiar with browsing the files and adding more files.
40	Schematic editor and drawing the circuit	▲ Over view. ▲ Introduction to the tools. ▲ Start the work and brows the parts. ▲ Connecting the circuit.
40	Layout editor and making final layouts	▲ Over view to the application. ▲ Placing the components and learning how to pick up the right packaging. ▲ Manual routing and basic information. ▲ Auto routing. ▲ Verifying the layout.
10	Q&A	▲ Giving participants chance to discuss the workshop and get more information from the trainer.

Day 2		
25	Exporting and manufacturing	▲ Ways to export schematics. ▲ Exporting layout. ▲ Exporting CAM files. ▲ Exporting bill of material. ▲ Start manufacturing.
50	Milling all the participants circuits	▲ This part will give the chance for all participants to mill their own circuit. ▲ This session will take time due to the long process of the milling; therefor the trainer should start explaining and soldering process and show a live practice.
30	Soldering session	▲ After the circuits are finished from the milling the participants will start this session which is the hands on experience. ▲ Introduce participants to soldering equipment. ▲ Soldering techniques. ▲ Using SMD components.
10	Q&A	▲ Giving participants chance to discuss the workshop and get more information from the trainer.

Requirements:

- Basic knowledge in reading circuits schematics.

Skills Learned:

▲ Designing schematics.
▲ Identifying different components packages.
▲ Finding components online.
▲ Designing double sided PCB.
▲ Exporting circuits files.
▲ Manufacturing double sided PCB.
▲ Using ERC and DRC in EAGLE to verify the manufacturability of the circuit.
▲ Tricks and professional way to develop PCB based on standards.
▲ Soldering through hole and SMD components.

Software:

▲ EAGLE CAD soft.

Materials:

▲ PCB.
▲ Components and soldering. (list of components is attached)

Outcome (Projects):

- The participants will be asked to draw simple circuit in the schematic editor.
- The participants will be asked to convert the schematic to layout and rout it.
- The participants will be able to follow the steps of manufacturing PCB and apply it.
- The participants will mill their own PCB and solder it.

P.S. The example that is going to be used in the learning process is the (Paperduino) circuit (http://paperduino.eu/doku.php), only they will make the design on PCB. This is a simple and useful circuit will give the participants the full practice of all the skills they need to make their own circuit. In addition, each participant is leaving with his own Arduino compatible micro-controller that can be used in the Arduino workshop.

Resources:

- http://www.cadsoftusa.com/index.php
- https://www.sparkfun.com
- http://www.adafruit.com/
- http://fabmodules.org/
- http://paperduino.eu/doku.php
- https://www.maharah.net/courses/inro2eagle
- https://github.com/MosabWadea/EAGLE-files
- https://github.com/MosabWadea/nodela

Attachments

1- The parts and components needed (*for each participant*) are found in the BOM (bill of material) file named (paperduino-customeParts_BOM.pdf)
2- The finished circuit and layout (*as EAGLE files*) are found in the folder (eagle-files).
3- The files that are used for manufacturing are placed in the folder (CAM-files).
4- Slides that are used in the first and second session are placed in the folder (slides).