Lasercutter

=> Use red button on top for emergency stop
=> Always be present when cutting is performed. There is a small risk of fire!

IMPORTANT ADVICE
When you use the Lasercutter for the first time with your own design: Start simple! Just draw a small rectangle or circle in Inkscape, and make sure you get it cut. When you know your workflow is good, go for the more complicated designs.

Setup:

- In the cupboard under the lasercutter on the left or right side you find the main switch that turns on the machines: the full spectrum laser (you must turn the red button to switch it on), compressor / airfilter (the large metal box, it has a switch at the back), cooler ("chiller"). It will make a lot of noise!
- Always check if the tube is well connected to the compressor. Check at the back of the lasercutter in the working area if it sucks air.
- Check if the small fan in the head of the laserhead blows.
- Connect the ethernet cable with the laptop/computer.
- Turn the computer on, password is MP021, turn off wifi (on the laptop)
- Start Retina Engrave software. This is the software that controls the lasercutter. This can take a while.
- The computer has to connect with lasercutter. This will fail the first time. The solution is to switch off the cutter (red button), wait a second or 10-20, and switch it on. Eventually repeat. You know the cutter is connected when on the small display at the right of the cutter an IP Address is shown in the upper line. If it stays 0.0.0.0 you are not connected.
- Connect Retina engrave to the cutter: on the screen bottom left you can connect it.
- Use the home icon of Retina engrave (in the buttonbar with a small house), it puts the laser in the upper right corner
- Test the laser with the laser button (red line and star) in the button bar (a few buttons to the right of the home button. It is next to a small nummer). When you push it you will see a small laser flash. If it doesn't, probably you forgot to switch some of the auxilliary equipment on.

Software:

- Use inkscape version 0.91 for design. There are rumours, and is our experience, version 0.92 doesn't work well with Retina. Some common errors by using inkscape:
  - After drawing an object, you must convert it to paths.
  - Set color opacity to 100% (so the last two hex numbers of RGBA will be ff)
  - As inkscape uses svg, it is impossible to start three lines or more from one point.
  - Set units to mm in File->Document properties->Page. It is very useful to use also the option "Resize to content" on that page with a 5mm margin.
  - Objects you want to vector cut should have a line color.

- You can use colors in your design to select different contours that need different settings of cutting. Always use this if you have forms within other forms. This makes it possible for you to first finish the inner forms and then the outer forms. If you first cut the outer forms, they are loose and might move, changing your dimensions/results.
- After making your design, export it to xps format (by using the print menu). **Select .xps as output format, NOT .oxps (won't work with Retina).** Some say you can export to .svg also, I did not get that to work.
- Import the design in Retina with File->Open
- You can install Retina Engrave 3D also on a windows laptop (from website manufacturer), so you can test the import on your own without a lasercutter connected.
- You can make a quick testdrawing in Retina engrave. Go to the design tab, draw a square, and click on the "render" button in the upper right corner of the sidebar.

Actual cutting:

- Lay some material in the machine (on the bed with heaxagonals). It must be 3mm acryl or 3mm wood. Other material or thickness is not supported. The maximum size is 500 x 300mm.
- Use the arrows in Retina engrave (jog left – right – up - down) to move the laser, to the point where the surface of your cutting will start. The laser will be positioned on the upper left corner of your cutting.
- After loading your design, you can use the buttons next to the home button to quickly see the outer contours of your piece.
- **Again: Use red button on top for emergency stop and be present during cutting!**
- To pause, just lift the lid
- Retina is very sensitive to errors in the drawing, but won't give a decent message
- To use 3D engraving, you have to enable the 3D tab in Retine. Also, you need to make a greyscale image. Every greyscale has its own engraving settings.
- It is said that pictures cannot contain groups, it gives the software hickups. Do "ctrl A" to select all and then several times "ctrl shift 9" until all groups are canceled.
- Let the protective cover on acryl plates during cutting and engraving (otherwise the debris of the lasercut will damage the surface)
- Laserpower and speed: this is something that you just have to try. My experiences (it depends on the kind and color of the material):
  - cutting in 3mm transparant acryl just 95% laserpower, and 25% laserspeed. 1 pass.
  - Engraving in acryl, get 2mm: Do 3 passes with 100% power, 20% speed. Just start by 1 pass.

Aftercare:
- After every session, clean the bed/bottom and the lid with isopropanol
- Remove debris
- If something is wrong: notify the OBA. Please don't leave it behind without telling someone.

3D-printer

Please add helpful comments

This is practical info in relation to Oba-Makerspace, checkout real tutorials online

Laptops are available with the necessary software to use the vinylcutter/lasercutter (has a dedicated laptop)/3Dprinter! Charge them while using

If you think necessary software is missing, let us know, it can be added to the standard image

You need to import your file in Cura before you can print it! Please calculate the printing time, it may not be over 40minutes. Do combine small prints with others so that they can be printed at the same time.

You can prepare your files at home an place them in the cloud!

Software: Cura; nozzle 0.4;

- You can find a lot of designs on Thingiverse and thinkercad
- Open it in Cura to select the right settings (support, fastness etc) and calculate the printing time. To do it yourself at the meetup if it takes less then an hour.
- Make sure you selected ultimaker 2+ (then you know it fits the bed 😌)
- Take an sd-card (there is one in the 3Dprinter)
- You need to save your file from within Cura to the removable sd card
- Choose eject card before taking it out (remove safely!)
- Put the sd card in the printer
- Select print by turning the round button, it will show the files on the sd card (check the printing time again, can you make it before 21h?)
- Select the right file and start printing
- After printing use the round button to delete the file from the sd-card.

Tips

- Slow is better 😊
- Don't touch the filament, we use what is inside (PLA)
- Other materials are possible, but we need to discuss it with OBA, they'll help us.
- If you have a big project to print that takes more than 1 hour, then they offered us that we can email the file and then they will print it for us.
- Nozzle 0.4