

Robotics Essentials

Lesson 1

3D-design for 3D-printing

Do you want to create the Design and the Code for Robots?
Do you want to know how to 3D-design parts and then to 3D-print them?
Want to learn Sketchup and *STL files creation?
Ever heard about Robot Operating System (ROS) for Linux?
Do you want your kids to have fun, while learning cutting edge technology?
Take your Windows laptop, and come to our lessons "Robotics Essentials" on MakerPlace!
On this particular lesson you will learn, how to design and quickly 3D-print your robot parts, the skill, that every inventor must have!
During this lesson we will together launch your part to print on our Robo3D printer.

Part I

45 minutes

1.1. The Big Picture of Robotics

Why robotics?
Mechanical, Electrical, Wireless, Control, ROS, Code
Robotics today

1.2. Course overview

1.3. Preparing the workstation (laptop)

3D design software choice - what are the options?
What is STL format?
Why to choose Sketchup?
Sketchup installation - to quickly design part
Export STL plugin installation - to export part in STL format
Netfabb installation - to check the STL file for errors and correct them
Arduino and Taurino drivers installation - to install drivers
MatterControl installation - to 3D print the part

Part 2

1 hour 45 minutes

2.1. Designing the part

Choosing the template: inches, mm
Scaling up 1:100
Drawing primitives: line, circle, rectangle, polygon
Push / Pull operation
Follow me operation
Making component
Connecting components
Scaling down 100:1
Solid inspector check

Exporting 3D part in STL file
Netfabb check and repair if needed
Avoiding mess: parts numbers, names and revisions

Everybody chooses 5 minutes what to design, and makes a pencil & paper sketch, then designs it in Sketchup.
Then all vote for the best and nice design, we print the part of the winner.

Part 3

30 minutes

Making the part

Part fabrication choice - what are the options?

- 3D printing
- 2D Laser cutting
- CNC machining

Why 3D-printing?

Where in San Diego I can print my parts? www.3Dhubs.com

Materials for 3D printing:

- What materials are easy to print?
- Mat or glossy
- Strength: PLA, ABS, Nylon, PET
- Hard or flexible
- Transparent or coloured
- Special effects composites: glow in the dark, wood, carbon fiber, bronze, conductive etc.
- 3D Printing Material Guides websites

3D-printing the part of PLA using Robo3D printer of MakerPlace

Printing launch procedure

Where to buy the filament? How much is it?

Filament spool installation

Preheating the printer

Loading STL file in MatterControl, orientation, rotation

Cleaning the bed, applying hairspray to it

General settings:

- Printing quality (layers height) - 0.2
- Outer surface perimeters - 2
- Top & bottom solid layers - 6
- Infill density - 0.3
- Infill type - triangles
- Skirt loops - 4 loops, 12 mm
- Support material - yes, not everywhere
- Center on bed - yes

Filament:

- Diameter 1.75
- Extruder temperature 220
- Bed temperature 60

Printer: Z axis offset
Options export and import

MatterControl 3D printing launch procedure for ROBO3D

GENERAL ACTIONS:

1. Bring your filament 1.75 mm thick. Tested and good brands are Hatchbox, 3D Solutech, 25 USD per 2 pounds (1 kg spool) on Amazon.com.
2. Turn 3d Printer on. Power button is on the back, where power cord is entering printer.
3. Launch "MatterControl". Press "Connect". It should say "Connected".
4. Press "Add" and add your 3D model in the printing queue. Model should be in *.STL format. You can check it for errors in Netfabb Basic (shortcut is on the laptop desktop) and repair there, if they exist.
5. Orient the model how you want it to be printed (on the bottom of the 3D rendering -> Edit, then rotate, scale if needed, and make sure it is aligned to the bed).
6. Clean the bed and apply hairspray on the printing area.
7. Settings & Controls -> Controls -> Preheat Extruder to 215 degrees, Preheat bed to 60 degrees.
8. Load the PLA filament. Make sure extruder is fully preheated to 220 degrees, and filament goes freely to the printing head and squeezes from it like a tiny spaghetti to the bed. If it does not do this, ask for help, do not continue.
9. Make sure the bed has the space to move to the back, where power cords are. Sometimes printers are too close to the wall, it will ruin your print if bed will hit the obstacle (stepper motor of the bed will loose the exact position).

MAIN SETTINGS:

10. Settings & Controls -> Printer -> Z Offset 0.6 mm (Jack printer), 0.9 mm (Jill printer).
11. Quality (Low, Medium, High), usually we print on Medium, it gives layer resolution 0.2 mm.
12. Material -> PLA

SECONDARY SETTINGS: (Settings & Controls -> General):

13. General -> Layers / Surface -> Top Solid Layers 8 counts.

14. Bottom Solid Layers - 8 counts. It will give you sturdy upper and bottom plastic plates of 1.6 mm thickness (0.063 inch).

15. Infill -> I usually use 0.3 or 0.4. 100% solid is 1.

16. Infill type - choose what you like. Structurally best are triangles and honeycomb, I usually use triangles or grid, it makes the life of the 3d printer longer, as it shakes less.

17. Support Material -> there are two little boxes with crosses. If you remove both, slicer will not generate supports at all, if you do check "Generate Support Material", it will generate it where it thinks it needed. If you do check both little boxes with crosses, it will generate support everywhere, where your 3D model has parts in the air.

18. Output Options -> "Center on the Bed" or not. In latter case, you will print your part exactly as you see it on the rendering on the right in MatterControl.

19. Filament -> Filament -> Extruder Temperature. For PLA should be 220 degrees Celsius.

20. Filament -> Filament -> Bed Temperature. For PLA should be 60 degrees.

21. Press "Print", and good luck with your invention :)

P.S. Don't forget to oil the printer rod and slides with engine oil from time to time. This will make the 3d printer to live very long time.

If you have further questions:

Nick Grigoryev

619-551-0828

nikgrigoryev@gmail.com

LESSON FEEDBACK FORM

Fields are to be marked from 1 to 10

1 = Completely No

10 = Absolutely Yes

and 8 shades in between

Was the lesson interesting for you?	
Do you feel the matter covered enough today?	
Do you want to learn more 3D design?	
Do you want to know more about 3D printing?	
Do you feel this lesson should be more short and quick?	
Do you feel this lesson should be more long and detailed?	
Was it fun?	
Was it boring and not interesting?	
Are you interested to know more about Robotics?	
Would you come again to learn in MakerPlace?	
How about value for money you paid?	
Do you like the teacher?	
What about the language of the teacher? Do you like it or not?	
How do you like the learning room?	

What would you like to learn more?

Please tell us anything else we should know, which was not in our questions: