

Complex of Technical Schools in Grudziądz



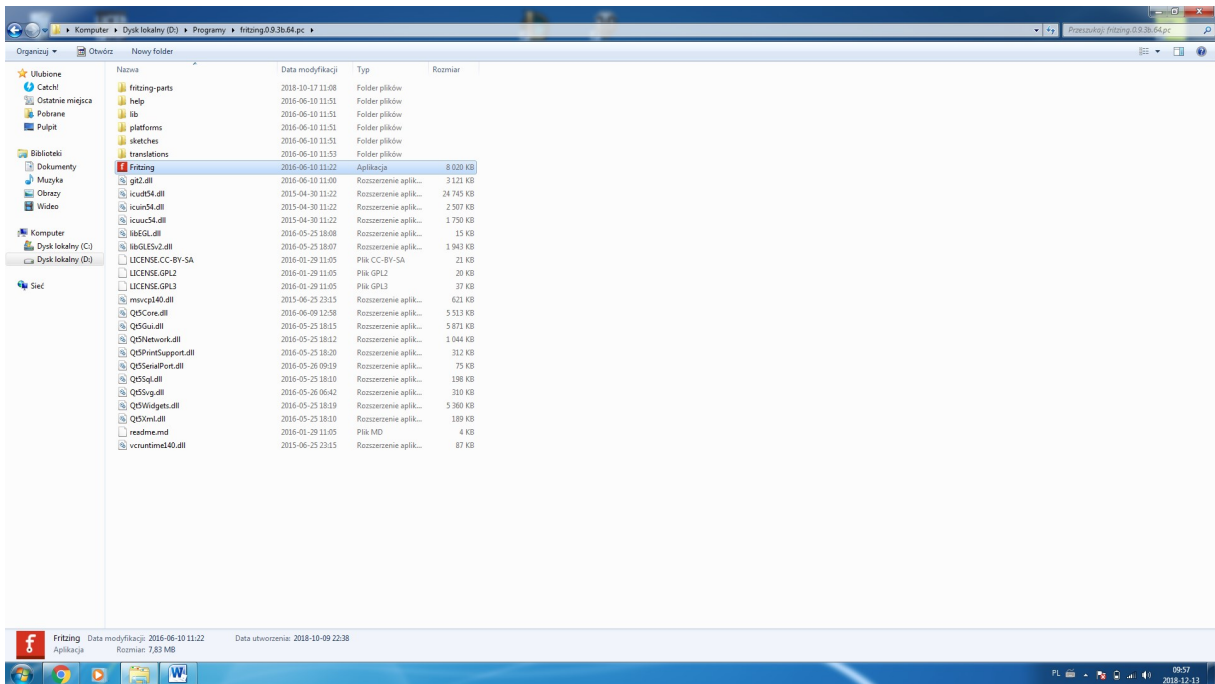
Automation & Robotics LAB (48)

Laboratory manual:

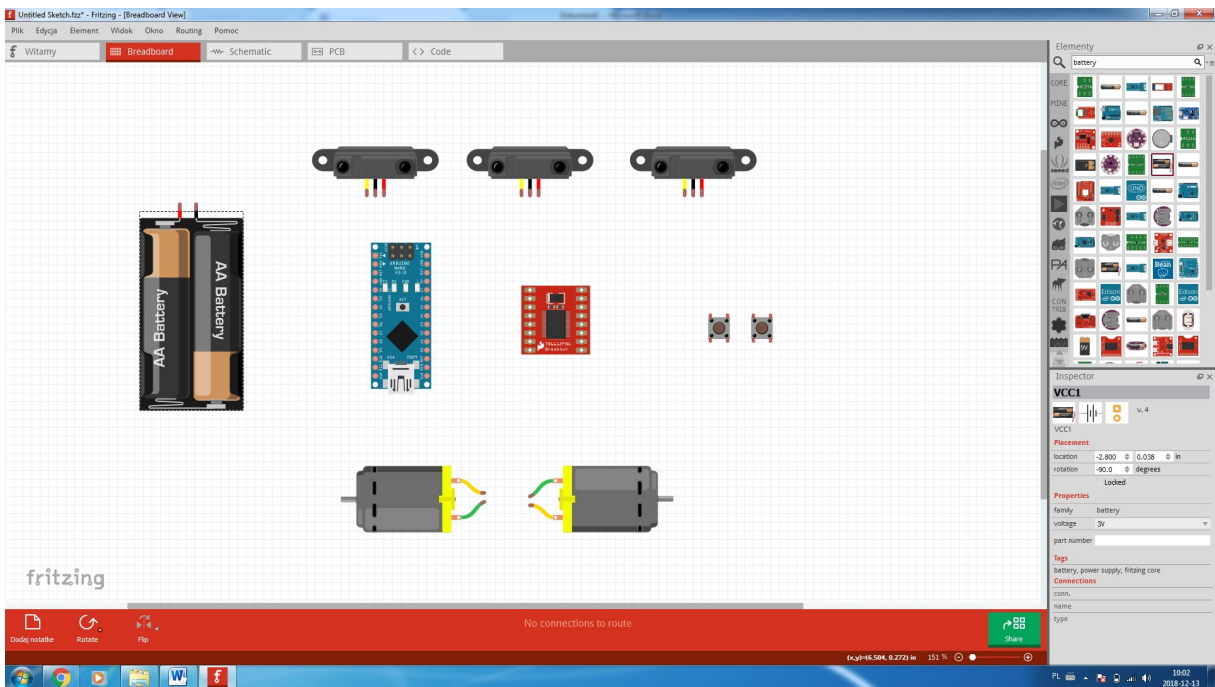
Basics of Robotics – PCB design using the „Fritzing” freeware
program

Prepared by: Marcin Jabłoński, MSc Eng.

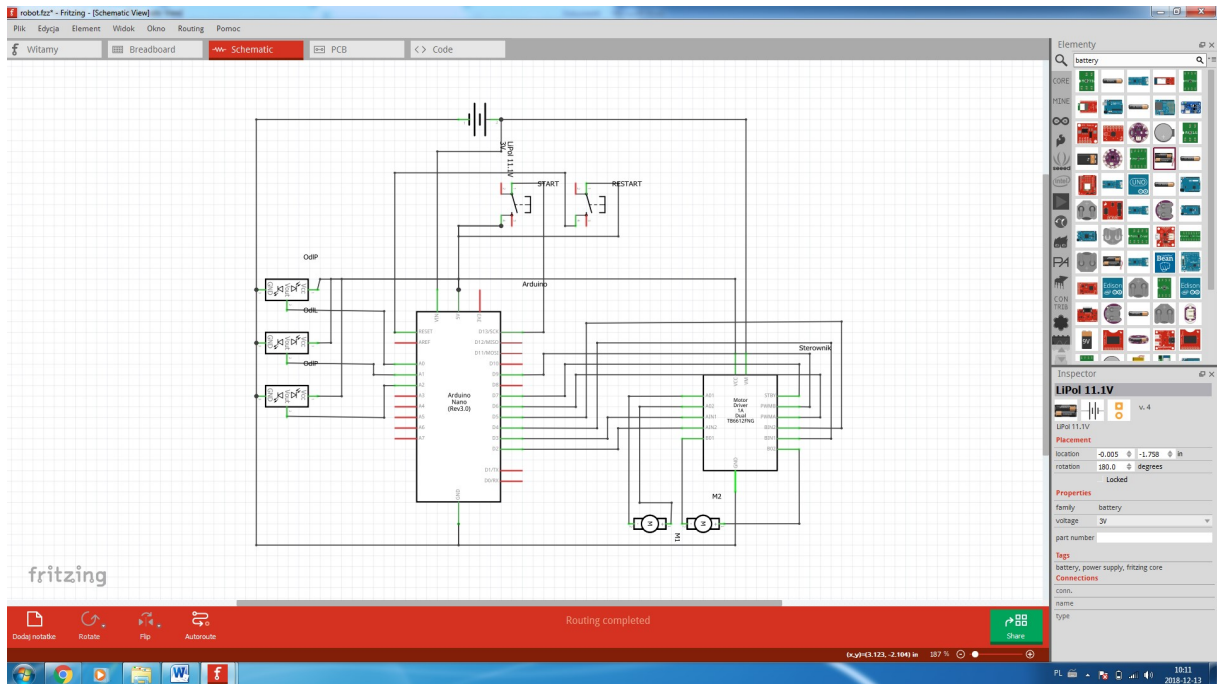
1. We run the free Fritzing program.



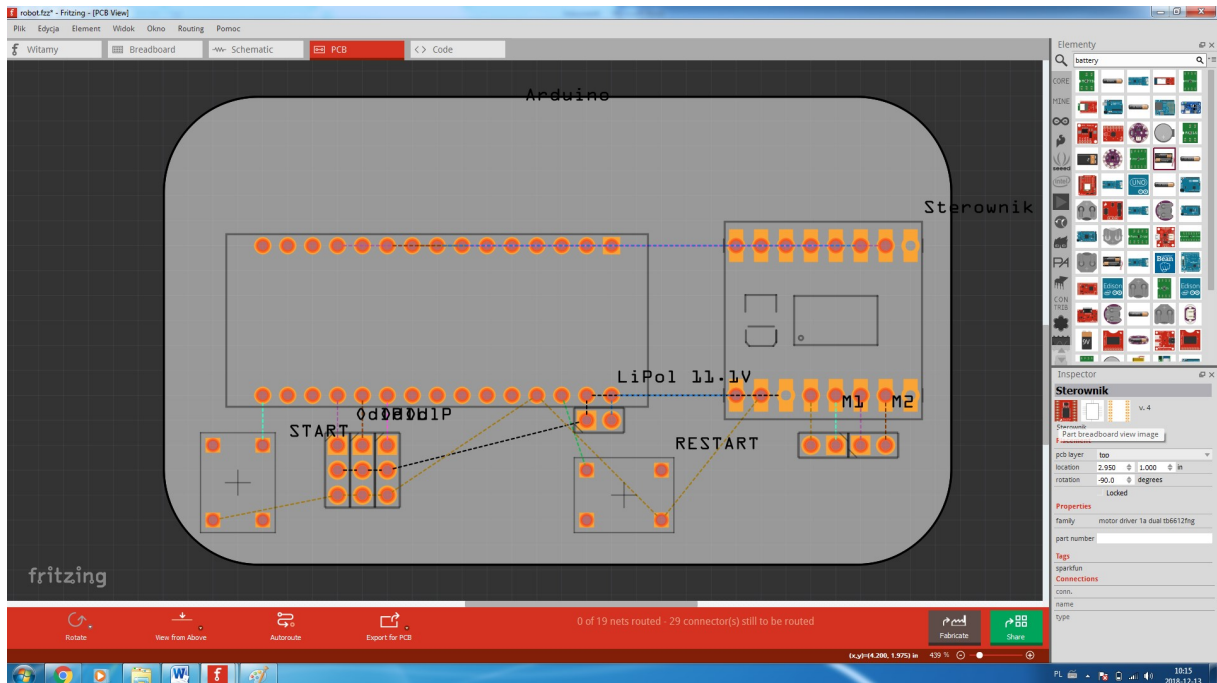
2. We choose the elements that we want to put on our PCB.



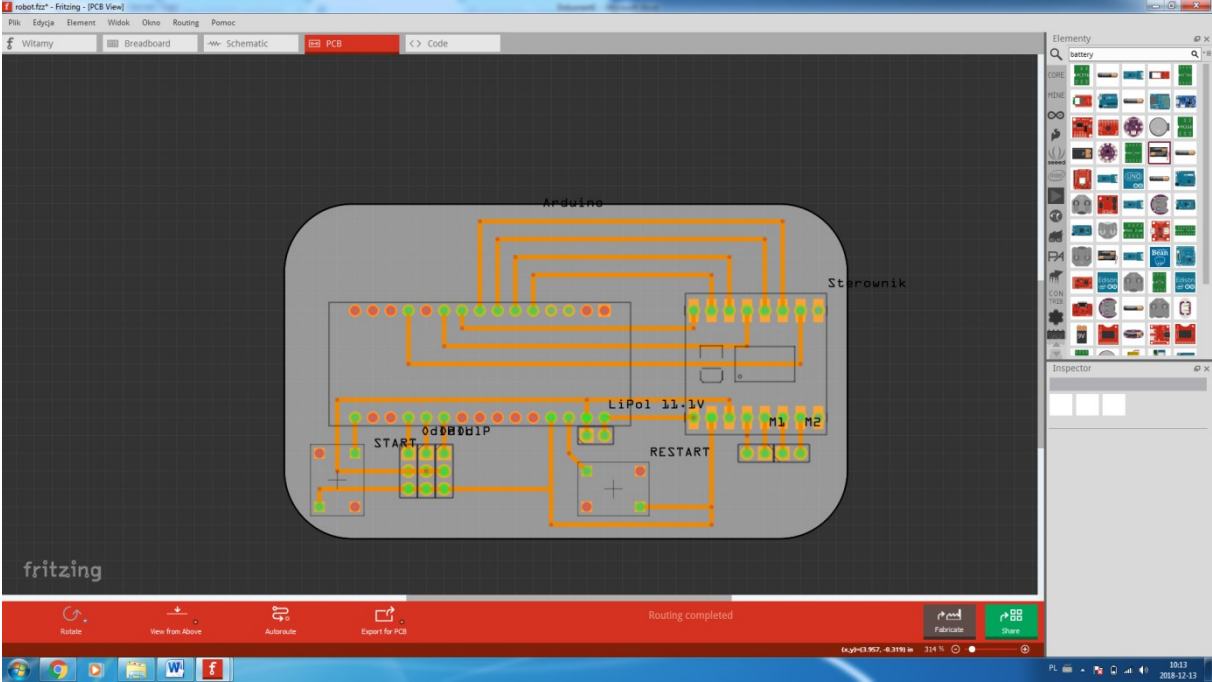
3. Go to the "Schematic" tab and create a connection scheme.



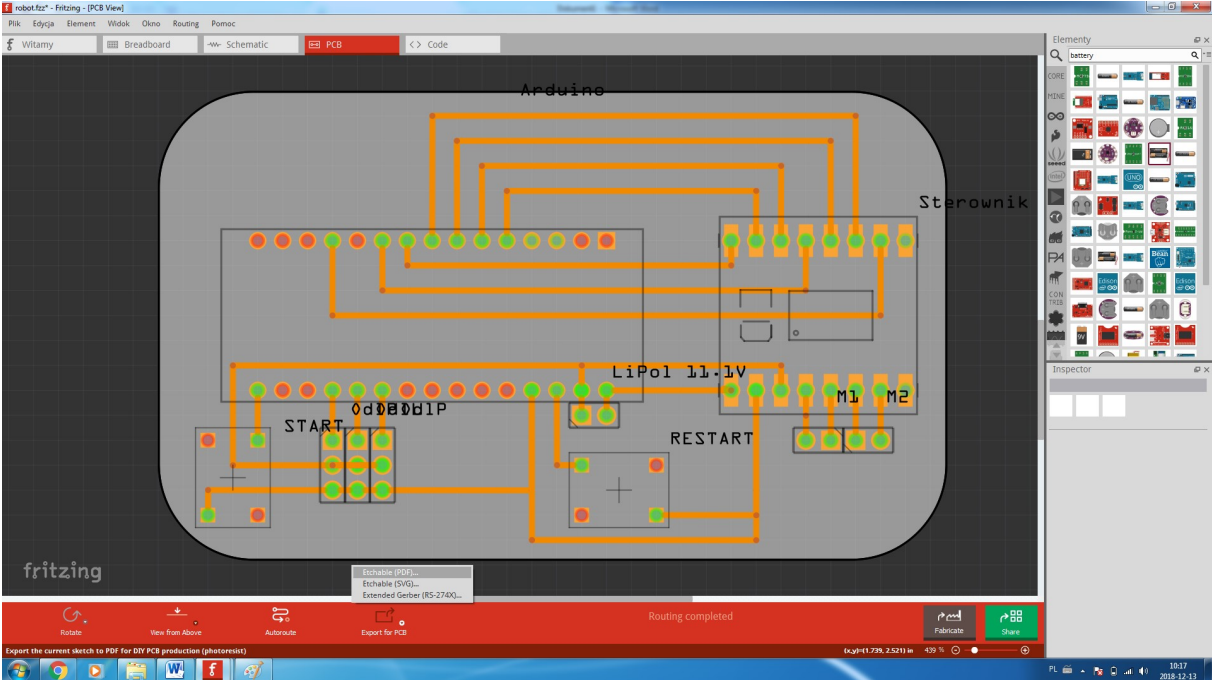
4. Go to the "PCB" tab and place the elements in the target position.



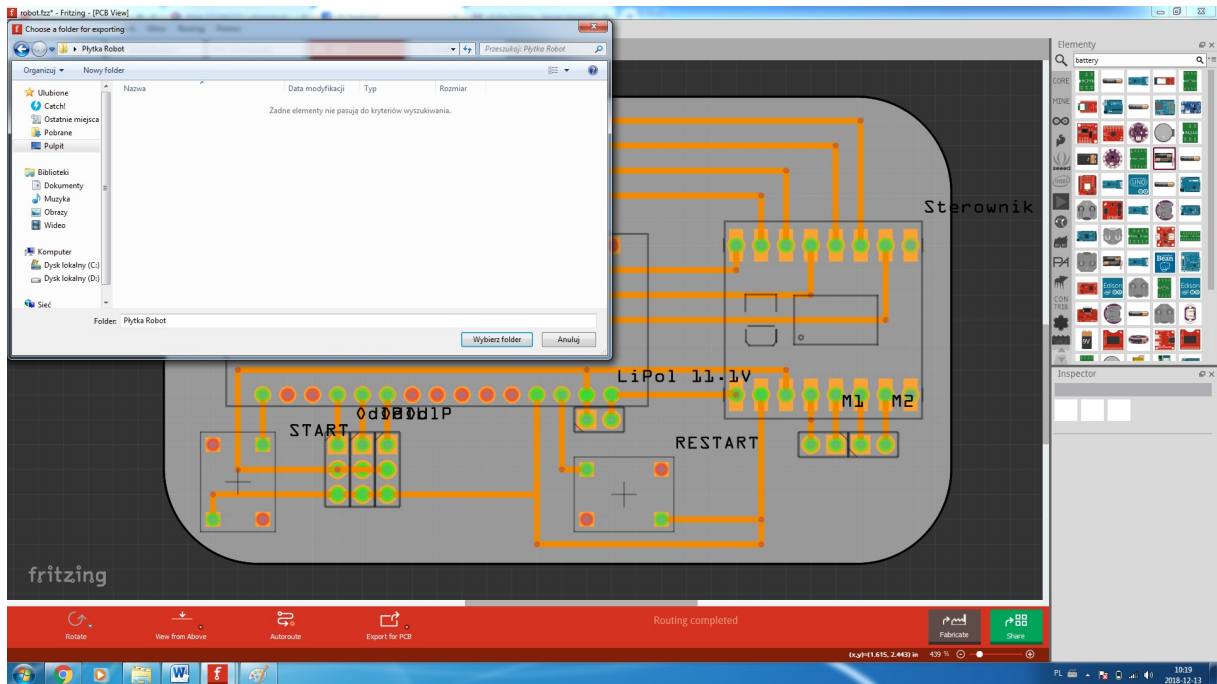
5. We arrange our paths so that they do not intersect, because it will be a one-sided etched PCB.



5. Click "Export for PCB" and select "Etchable (PDF) ..."



6. We select the destination folder for our prints and click "Select folder".



7. To correctly etch the plate, we select and print the paths in mirror image.

