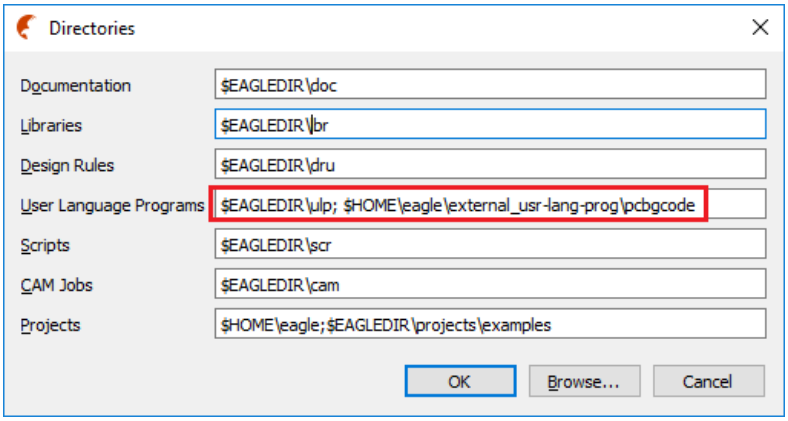
Create PCB using Cadsoft Eagle and Roland SRM-20

Installation

To start designing your PCB you need the Cadsoft Eagle software. It can be downloaded from here. In addition to Cadsoft Eagle you need a plug in to Eagle called PCB-GCODE. It can be downloaded from this GitHub repository. The “pcbgcode” folder path must be added to the directories of the “User Language Programs”. To do this go to the control panel and under “Options” select “Directories”. Add the path to the field “User Language Programs” by separating the existing and new directory with a semicolon (;).

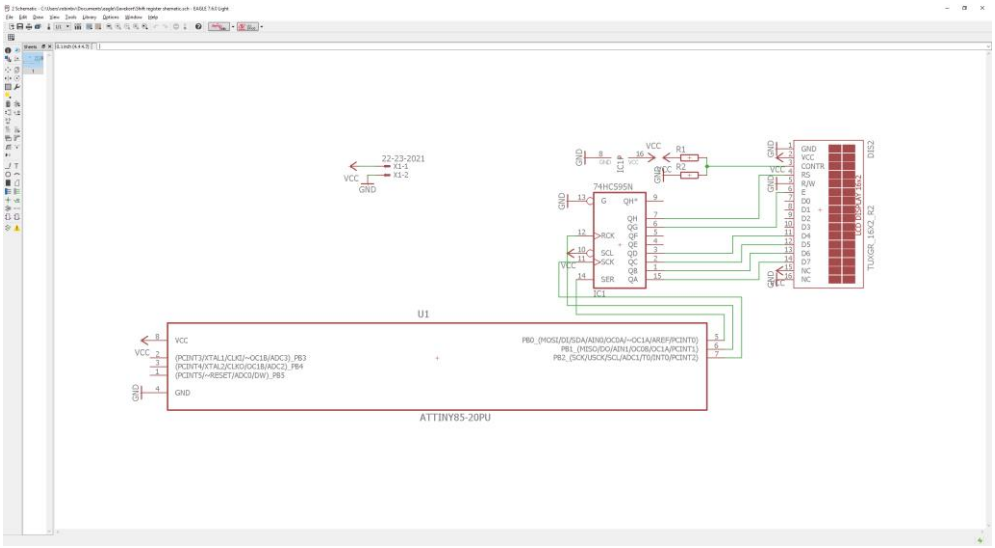


The last program needed is the “G-code correcter” program. It can be found in the GitHub repository.

This program does not need any installation.

Designing schematics

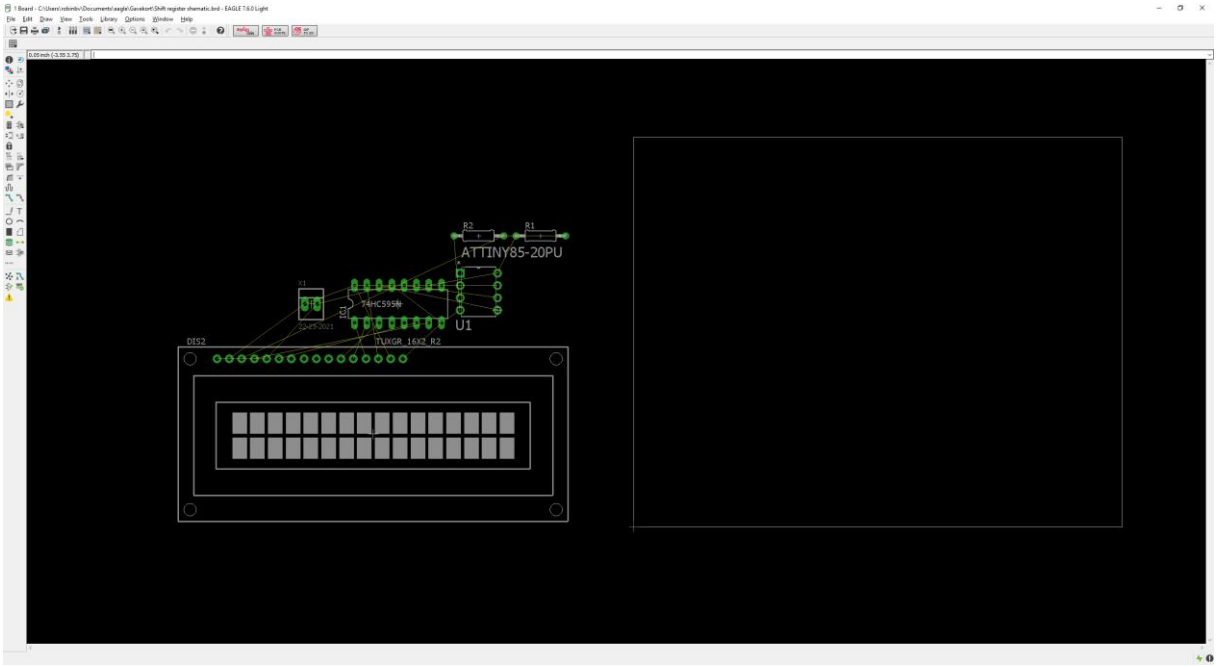
Create a new project from the “Option Panel” and right click on the project folder to create a new schematic. Add your components by using the “add” command, and the “net” command to connect the components together.

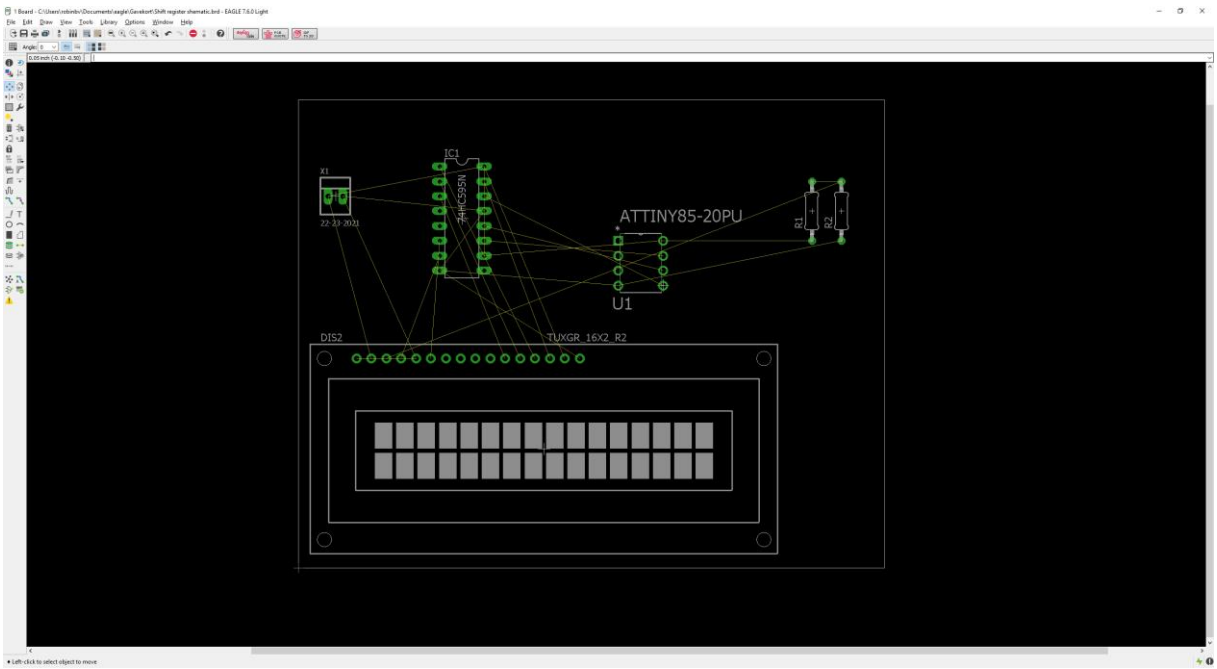


Designing board layout

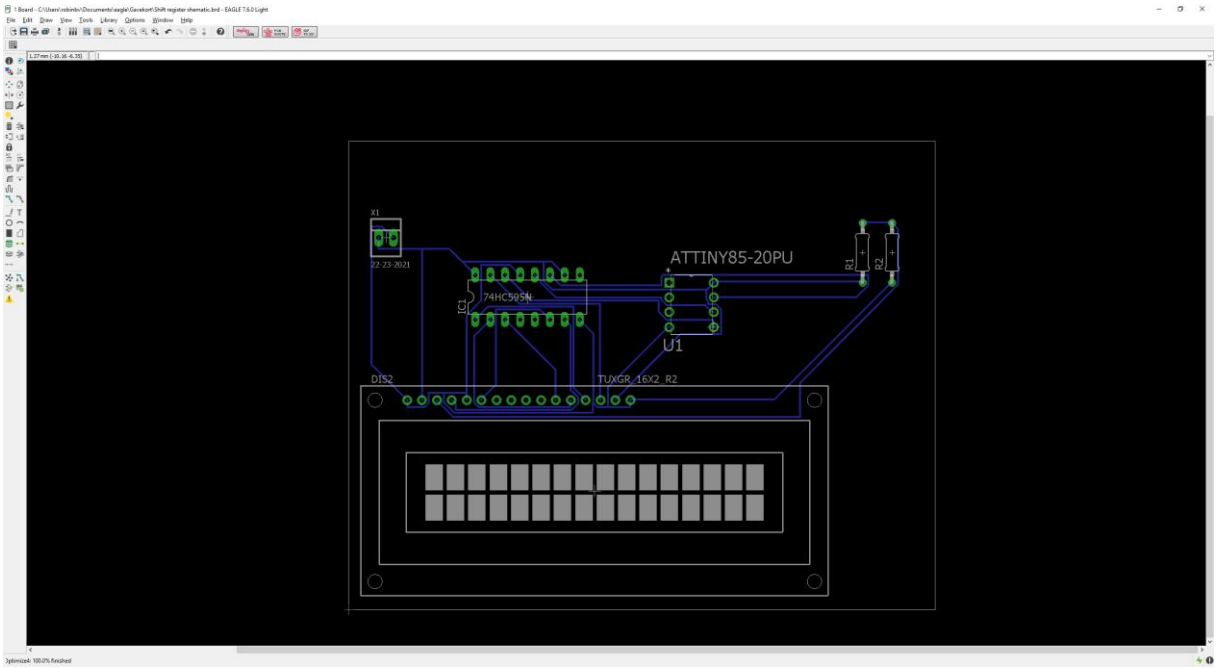
From the schematic go to “File” and select “Switch to board”. If you get a warning saying that the board does not exist. Select “Yes” to create the board file from the schematic.

All the components will be placed outside the area of the board, so use the “move” command to move the components to the board. While in the “move” command, right clicking will rotate the component.

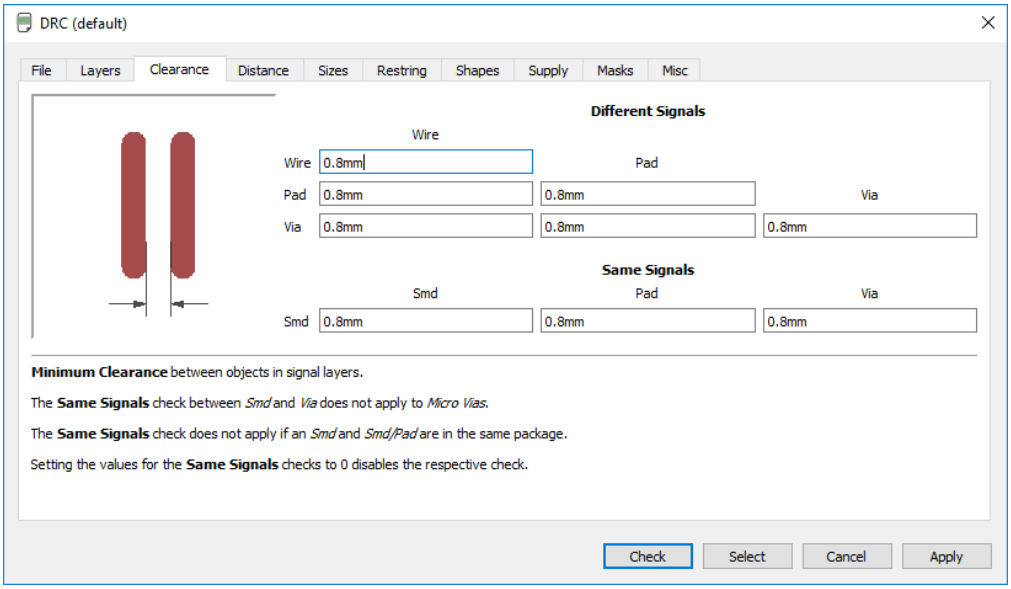




All the yellow lines indicate where we have to make connections. To do this we use the “route” command. Be aware of the layout you are routing on. For though hole components we need to route on the bottom layer and for surface mount components we need to route on the top layer. Choose the layer to route on in the top left corner when using the “route” command. Red routing lines indicates routing on the top layer and blue routing lines indicates routing on the bottom layer.



When routing your board, it can be helpful to use the “DRC” tool to check for errors. In the “Clearance” tab fill in the routing tool diameter to check for errors.

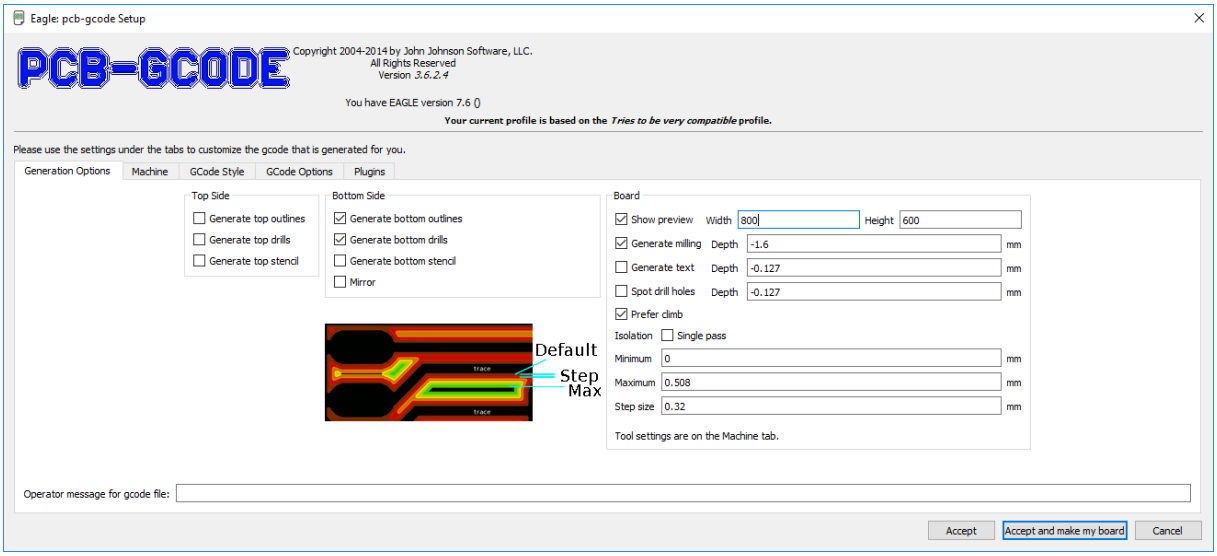


Fix the errors if you have any.

To generate the milling line, use the “wire” or the “circle” command and draw on the “Milling” layer.

Generate G-code

When you have routed your board it's time to generate the G-code for the Roland SRM-20. In "File" select "Run ULP" and locate the "pcbgcode" folder. Open the "pcb-gcode-setup.ulp" file.



The default settings are set to use an end mill with a diameter of 0.0312 inch and a board thickness of 1.6mm. It is also set to generate bottom outlines (trace milling), bottom drills and milling (cutting).

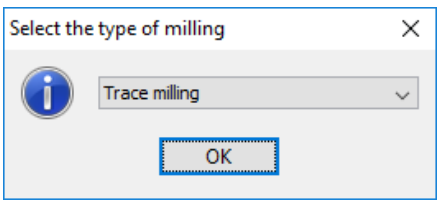
Under the "General Options" tab you can select what type of milling files (top and bottom) you want to generate and the settings for the board. The "Step size" should be about 40% of the tool diameter. Under the "Machine" tab you can select different tool sizes and change the feed rates.

When you have selected your settings click "Accept and make my board" a new window will be shown and this will show a preview if the milling sequence. Type "X" or "Q" to quit this window and click "OK" on the dialog. Several previews may show. This depends on the selections you made in the setup of the g-code.

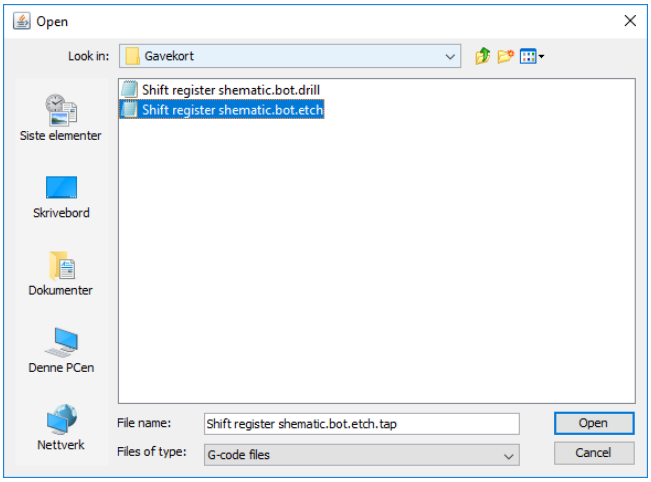
The generated g-code files will be placed in the project folder.

Correct the G-code

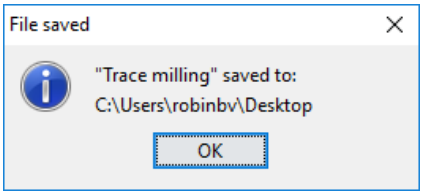
The g-code generated by "PCB-GCODE" is not compatible with the Roland SRM-20, so we have to correct the g-code files by using the "G-code correcter" program. When you run the program, the first thing you will be promoted to select is the type of milling you are going to do.



The next step is to find and open the file you want to correct. Files ending with ".etch" is for milling the traces, ".drill" is for hole drilling and ".mill" is for cutting out the board.



After selecting the file you want to correct you have to give the new file a name and select where to save the file. If everything went correct a dialog should show the name of the file and where it has been saved.



For the hole drill option, the program may generate several files. One file for each of the different drill sizes. The files will get the name: “name” + hole size.

Roland SRM-20 setup

Make sure the mill is turned on, and open “V-panel”.

Select the “G54” coordinate system and set the origin. If you are milling bottom traces the origin should be set to the right bottom corner of the PCB and to the left bottom corner for top traces.

When setting the origin of the Z-axis first set the bit all the way up in the collet and move the bit close to the PCB. Then open the set screw and let the bit drop on to the surface. Set this height to the Z-axis origin.

To start milling click on “Cut” and select the file you want to run.

If you can’t open the file open the settings and set the machine to NC files.