

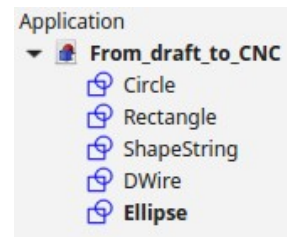
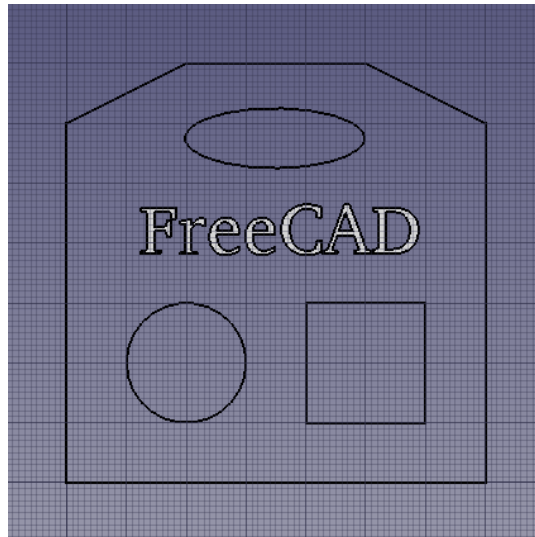
From draft to CNC mill

Workflow for FreeCAD 0.17 and LinuxCNC

Example: front plate

1. Draft

Test drawing:



How to do it: Tutorial: http://www.freecadweb.org/wiki/index.php?title=Draft_tutorial

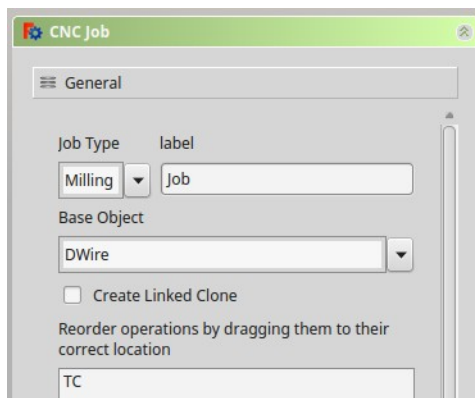
2. Creating the paths

- Switch to Path workbench



- Create a new path object

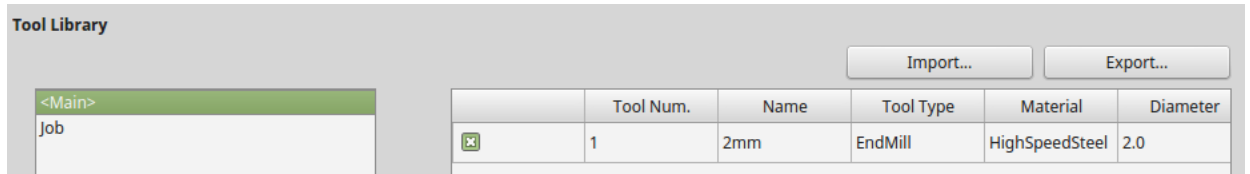
•



- Set base object to the object that represents the outer boundaries (in our case a Dwire). This will be used for the outer contour path.

-  Edit tool table

Create new tool e.g. 2mm end mill and add it to the job (with the blue arrow button in the middle)

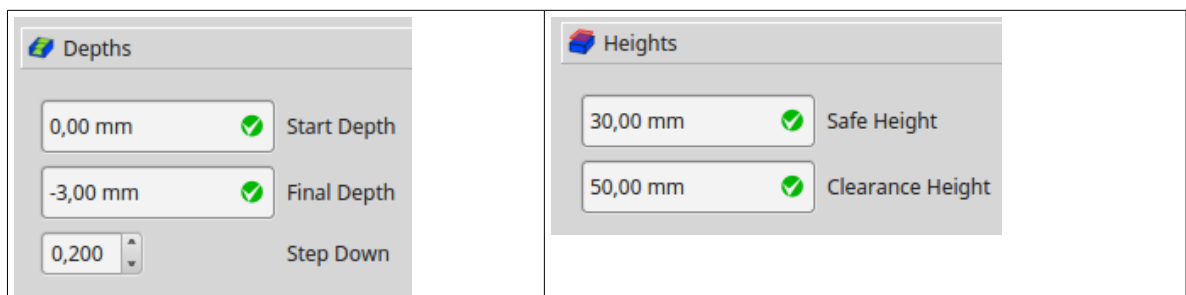


Now the new tool appears in the tree view.

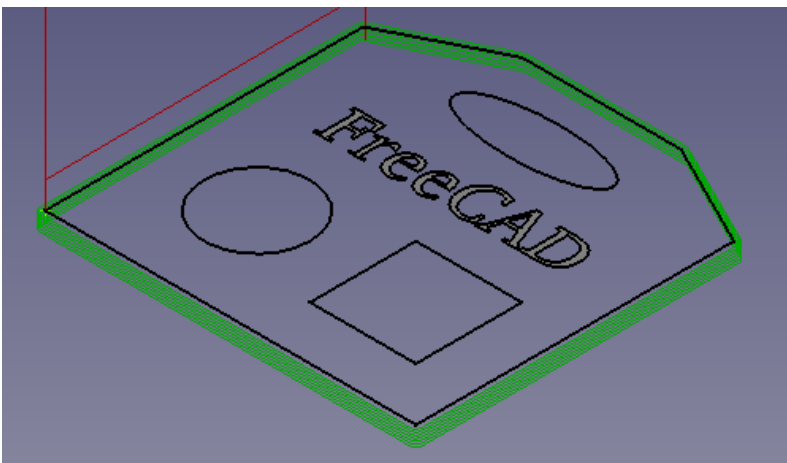
- In the tool properties box, set spindle speed and feed rates


-  Create **contour path for the base object.**

This does a contour path for the outer boundaries of the object.
Set depths and safe heights



The path can be verified in the orthographic view:

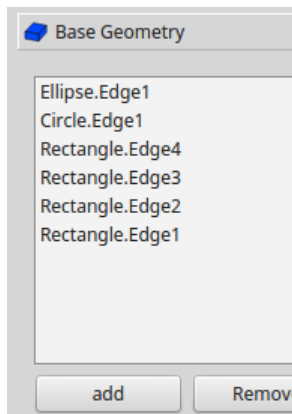


-
-  Profile based on edges

This does the path for any holes in the plate.

Select „Base Geometry“

In the drawing, select the desired elements (<Ctrl><LClick>) and press the „add“ button.

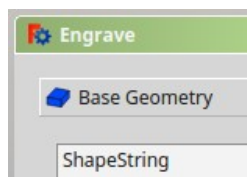


Set Depths and Heights as you have done for the contour path.

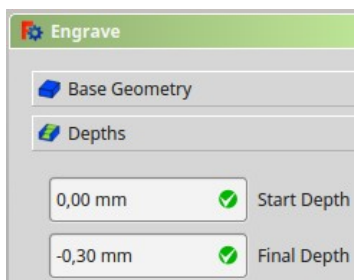
Engraving

Select „Base Geometry“

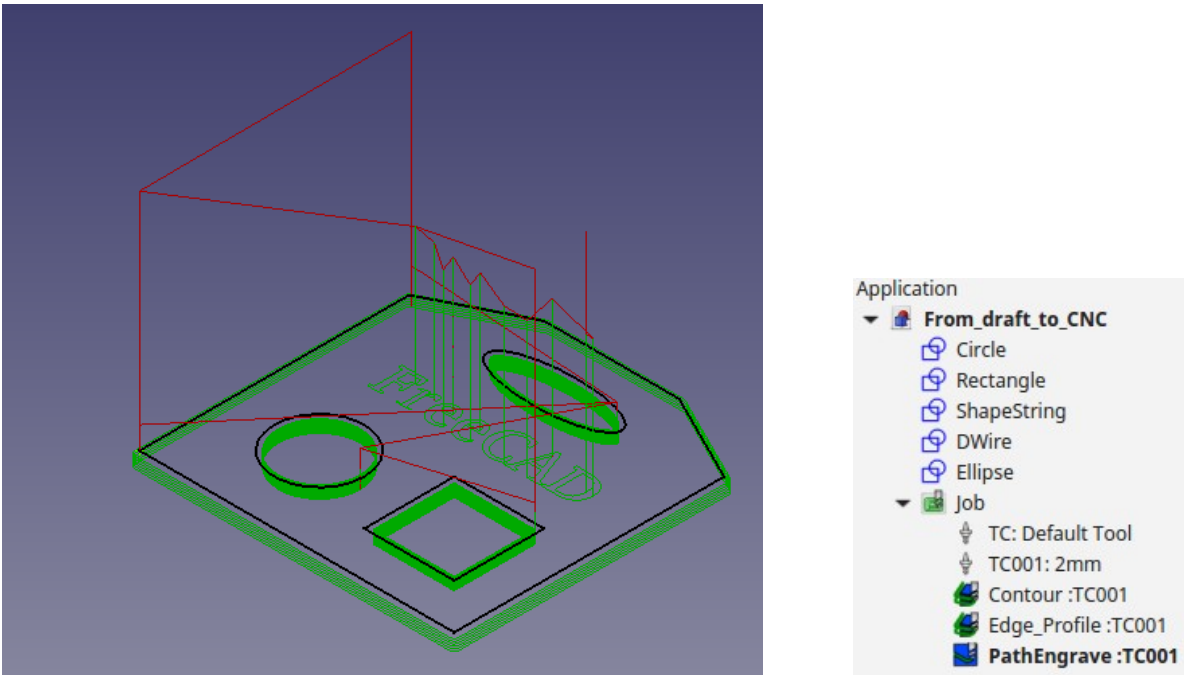
In the drawing, select the text to engrave and press the „add“ button.



Set the depth of the engraving, (and eventually the safe heights):



Now we see all paths in the tree view, and in the orthographic view of the drawing:



Note: to better see (or even to see at all) the engraving path, it is a good idea to turn off the visibility of the ShapeString (in the tree view, select and press <Space>).

3. Postprocessing the paths

This creates the G code.

It is a good idea to produce 3 different files for contour, profile and engraving.

In the tree view, select the paths and press the postprocessing button. Save the resulting files. The result is displayed in a text box:

```

freecad-daily
|(Exported by FreeCAD)
(Post Processor: PathScripts.linuxcnc_post)
(Output Time:2016-11-22 18:35:22.376627)
(begin preamble modified)
G90
M03 S1500
G21
(begin operation: Contour :TC001)
(Contour :TC001)
(Uncompensated Tool Path)
G0 Z50.0000
G00 X-0.4470 Y60.8940
G00 Z30.0000
G01 X-0.4470 Y60.8940 Z-0.5000 F30.00
G01 X19.5520 Y70.8940 Z-0.5000 F30.00
G02 X20.0000 Y71.0000 Z-0.5000 I0.4470 J-0.8940 F30.00
G01 X50.0000 Y71.0000 Z-0.5000 F30.00

```

Note: I have slightly adapted the postprocessor script for LinuxCNC, so as to switch on the spindle at the beginning and switch it off at the end. This is done in the PREAMBLE and POSTAMBLE variables. To do this, I have a copy of /usr/lib/freecad-daily/Mod/Path in the folder ~/ .FreeCAD/Mod/Path that I can modify without changing the original. FreeCAD always looks looks in the home folder first. The script to edit is linuxcnc.py in the PathScript folder.