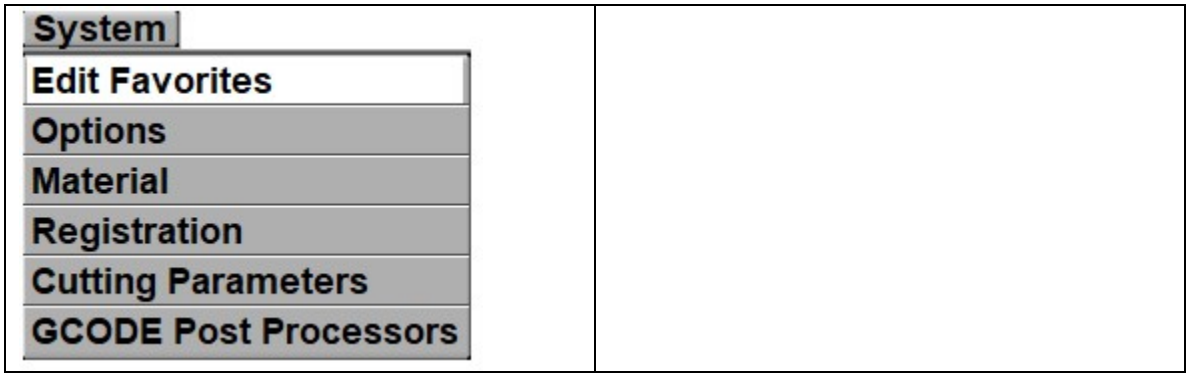
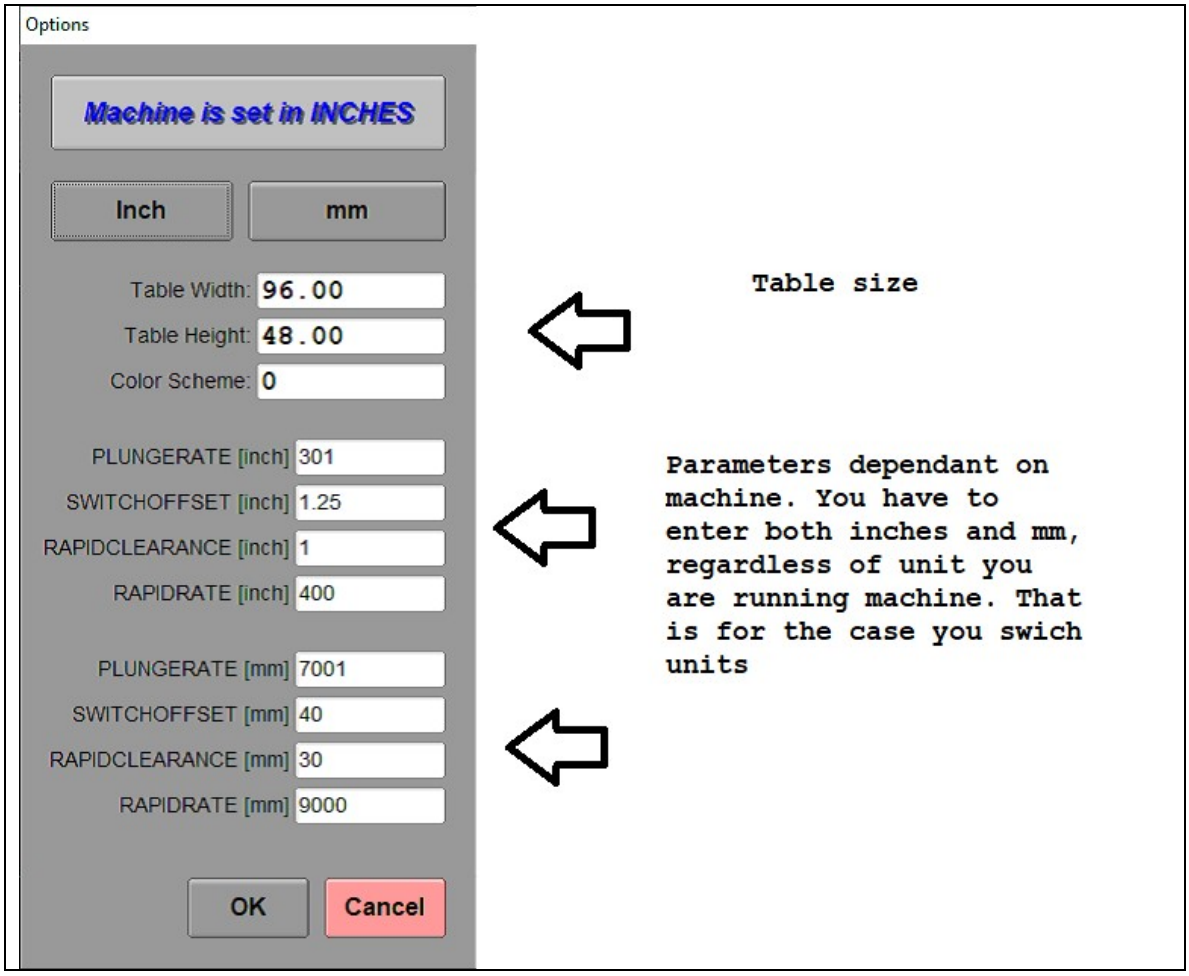
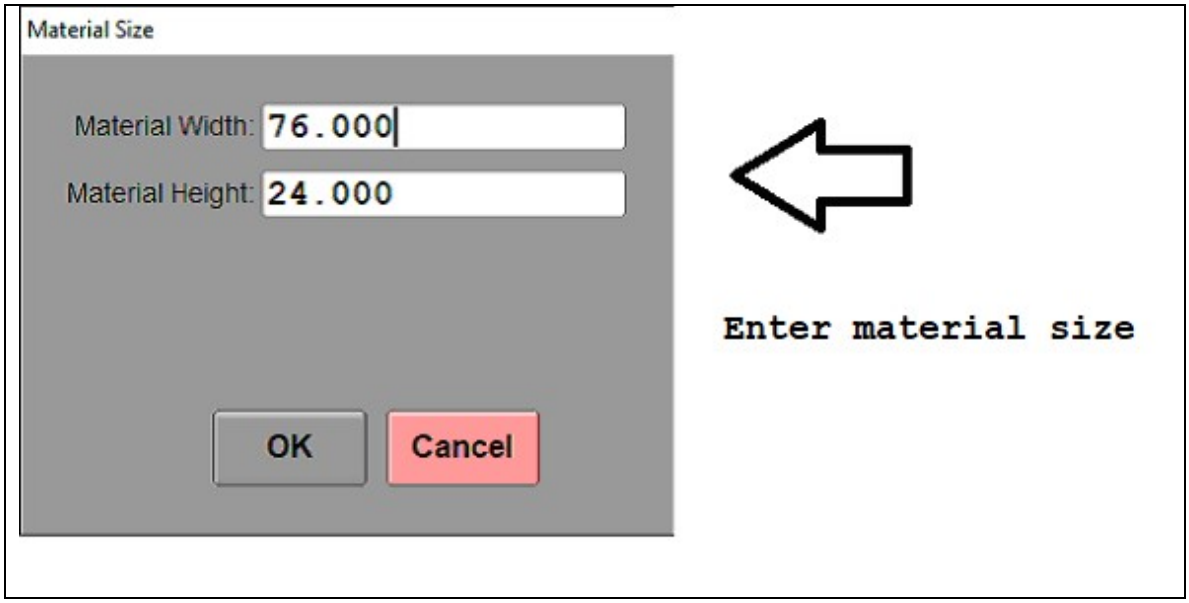
System


System
Edit Favorites
Options
Material
Registration
Cutting Parameters
GCODE Post Processors

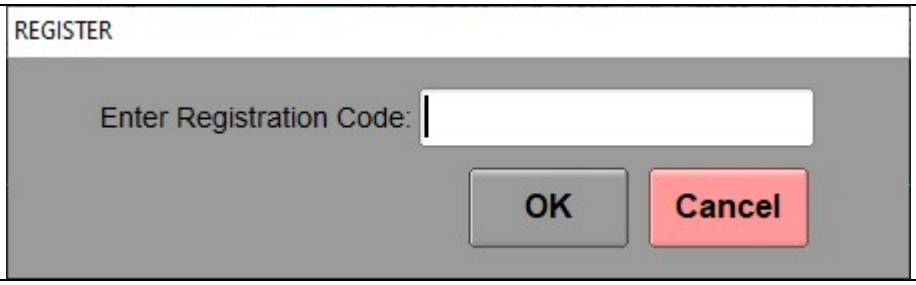
Options

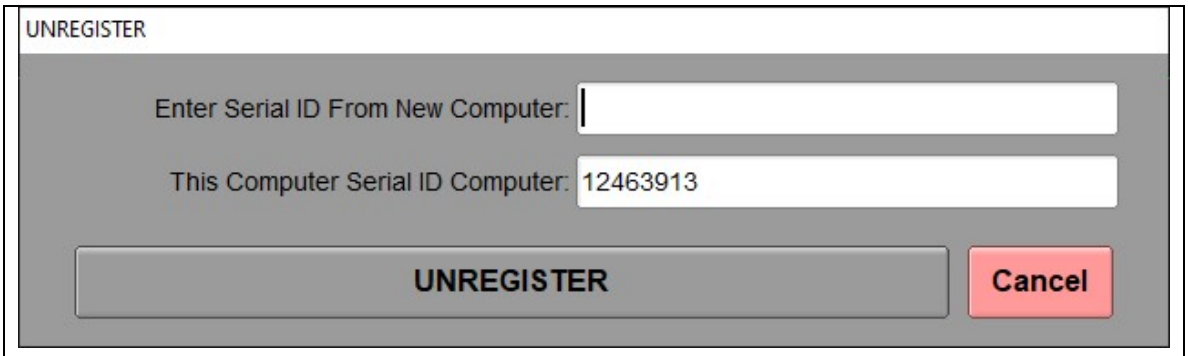

Options
Machine is set in INCHES
Inch
mm
Table Width: 96.00
Table Height: 48.00
Color Scheme: 0
PLUNGERATE [inch] 301
SWITCHOFFSET [inch] 1.25
RAPIDCLEARANCE [inch] 1
RAPIDRATE [inch] 400
PLUNGERATE [mm] 7001
SWITCHOFFSET [mm] 40
RAPIDCLEARANCE [mm] 30
RAPIDRATE [mm] 9000
OK
Cancel
Table size
Parameters dependant on machine. You have to enter both inches and mm, regardless of unit you are running machine. That is for the case you swich units

Material


Material Size
Material Width: 76.000
Material Height: 24.000
OK
Cancel
Enter material size

Register

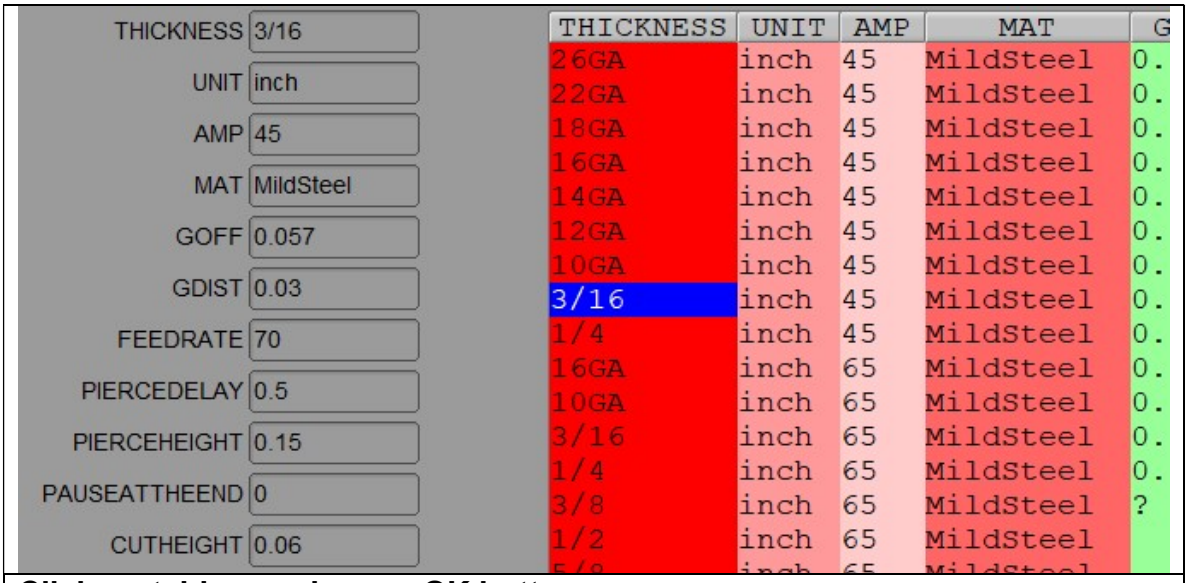




If you want to transfer license to new computer, first you have to have new computer with a demo version installed on it. Click About, and write down new computer serial number. Go back to old computer, go to menu System/UnRegistration, and enter serial number from new computer. Don't try to click UNREGISTER more than one time, and try to generate new license for more than one new computer. Once you press UNREGISTER, old computer will tell you password for new computer after 2 weeks, and you will able to register software to new computer. Old computer program will continue to work for another 2 months.
Warning: Don't try to press UNREGISTER twice, don't try to change time or date during this period, program may refuse to cooperate.

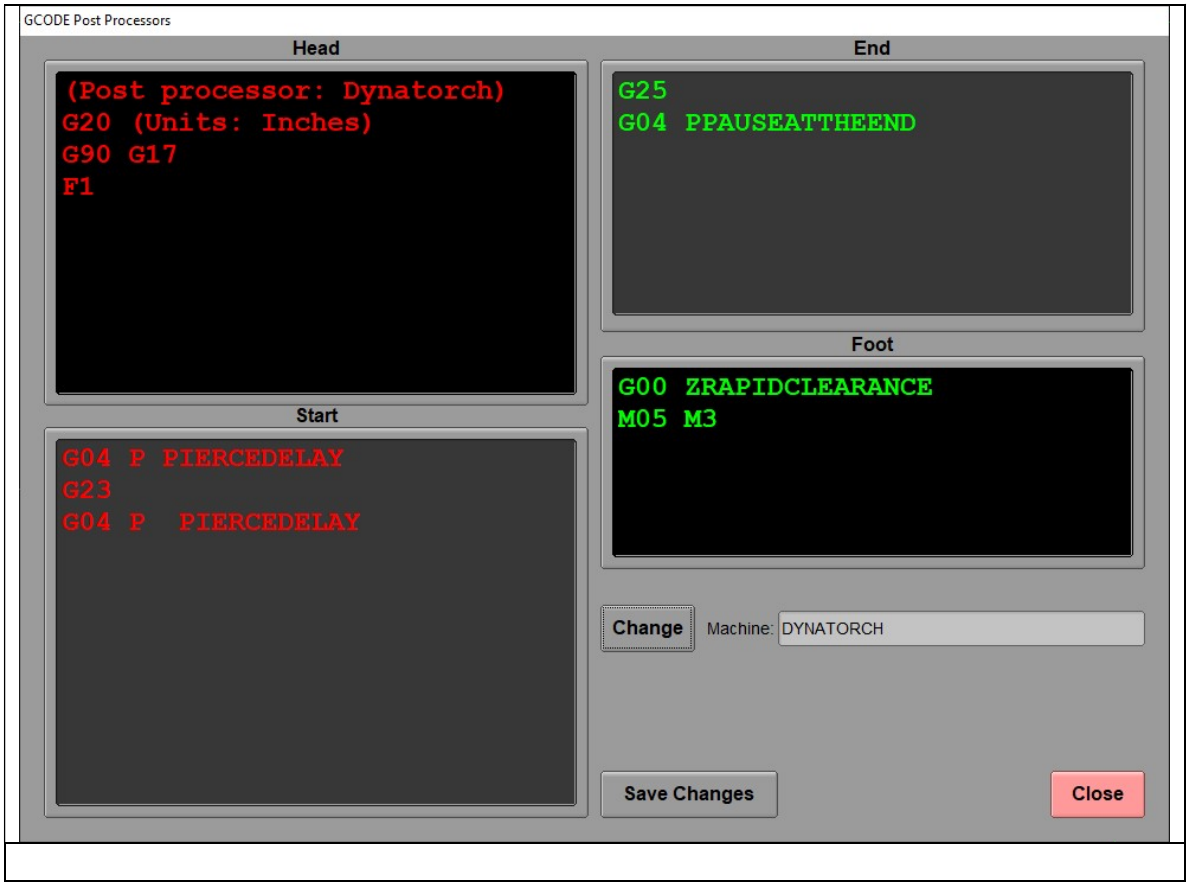
This option is not intended so yo can swich computers every day, every month or so. It is intended to provide password for new computer only.

Cutting Parameters



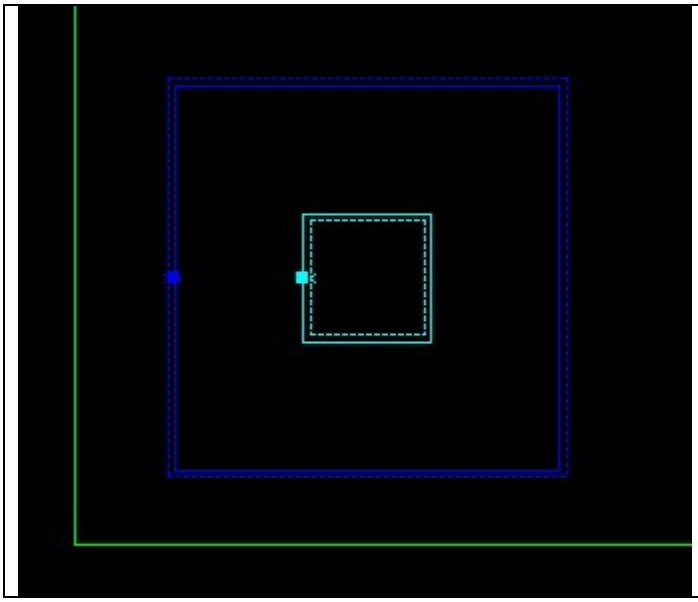
Click on table – and press OK button

G Code Setup


GCODE Post Processors
Head
(Post processor: Dynatorch)
G20 (Units: Inches)
G90 G17
F1
End
G25
G04 PPAUSEATTHEEND
Foot
G00 ZRAPIDCLEARANCE
M05 M3
Start
G04 P PIERCEDELAY
G23
G04 P PIERCEDELAY
Change
Machine: DYNATORCH
Save Changes
Close

GCODE – definition and generation

In this example we are going to show how gcode is created. Example part is very simple: a litlle square for inside cut and square for outside cut. You can see our cutting parameters GOFF, GDIST, FEEDRATE



```
GOFF,0.054
GDIST,0.03
FEEDRATE,270
PIERCEDELAY,0.2
PIERCEHEIGHT,0.15
PAUSEATTHEEND,0
CUTHEIGHT,0.06
PREHEATDELAY,0.2
PLUNGERATE,301
SWITCHOFFSET,1.25
RAPIDCLEARANCE,1
RAPIDRATE,400
```

Here is out postprocessor setup:

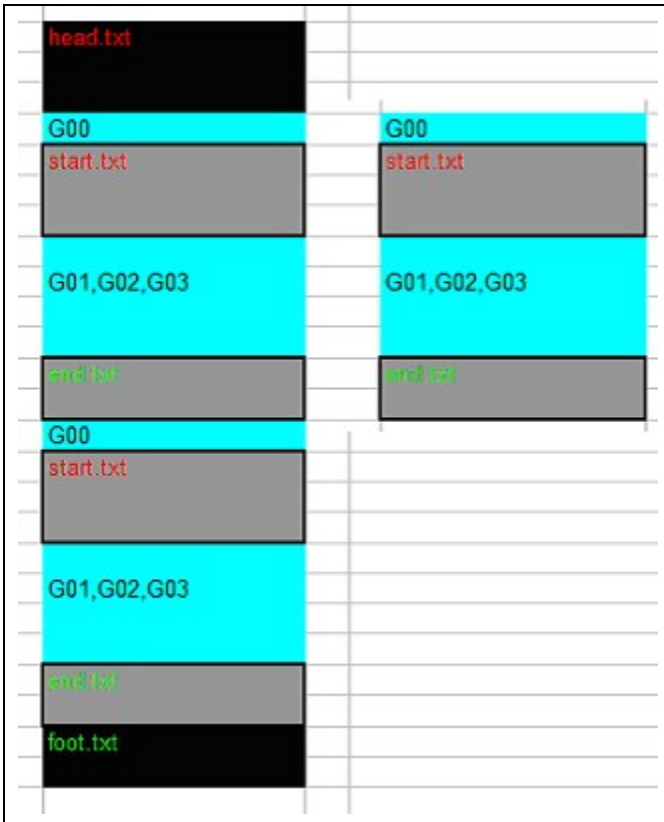
head.txt	foot.txt
G20 (Units: Inches) G92 X0 Y0 G40 G90	G00 ZRAPIDCLEARANCE M08 M30
start.txt	**end.txt**
G00 ZPIERCEHEIGHT G04 PPIERCEDELAY	M08 G04 PPAUSEATTHEEND

Look at foot.txt: `G00 ZRAPIDCLEARANCE` and `RAPIDCLEARANCE,1`
So it will be replaced like this `G00 Z1`

In first step gcode will look like this:

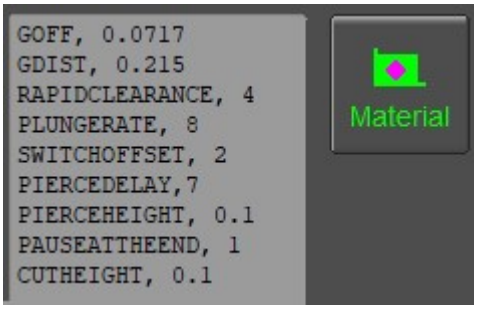
```
head.txt

G00 X1.8716 Y2.1101
start.txt
G01 X1.8416 Y2.0801
G01 X1.8416 Y1.6341
G01 X2.7336 Y1.6341
G01 X2.7336 Y2.5261
G01 X1.8416 Y2.5261
G01 X1.8416 Y2.0801
G01 X1.8716 Y2.0501
end.txt

G00 X0.7036 Y2.0501
start.txt
G01 X0.7336 Y2.0801
G01 X0.7336 Y3.6341
G01 X3.8416 Y3.6341
G01 X3.8416 Y0.5261
G01 X0.7336 Y0.5261
G01 X0.7336 Y2.0801
G01 X0.7036 Y2.1101
end.txt

foot.txt
```

For each polyline G code will be created this way

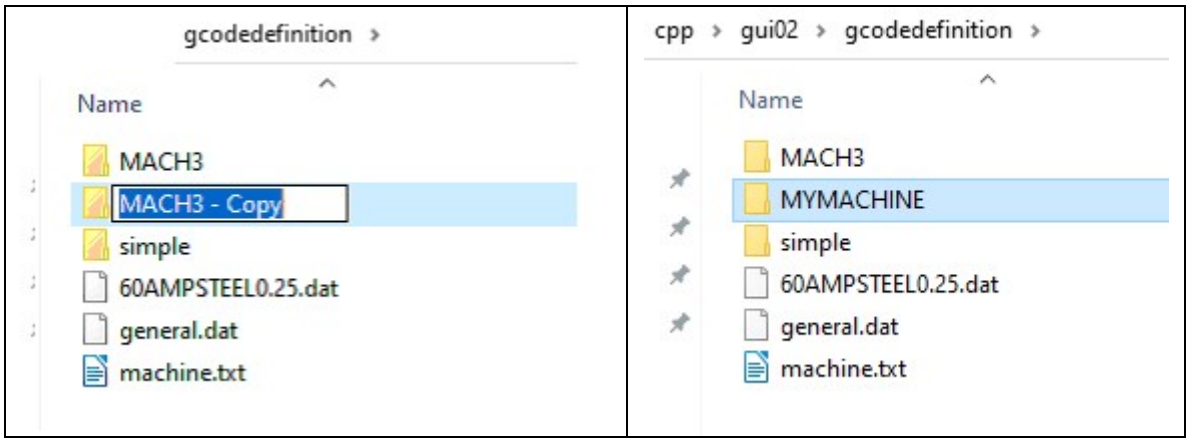


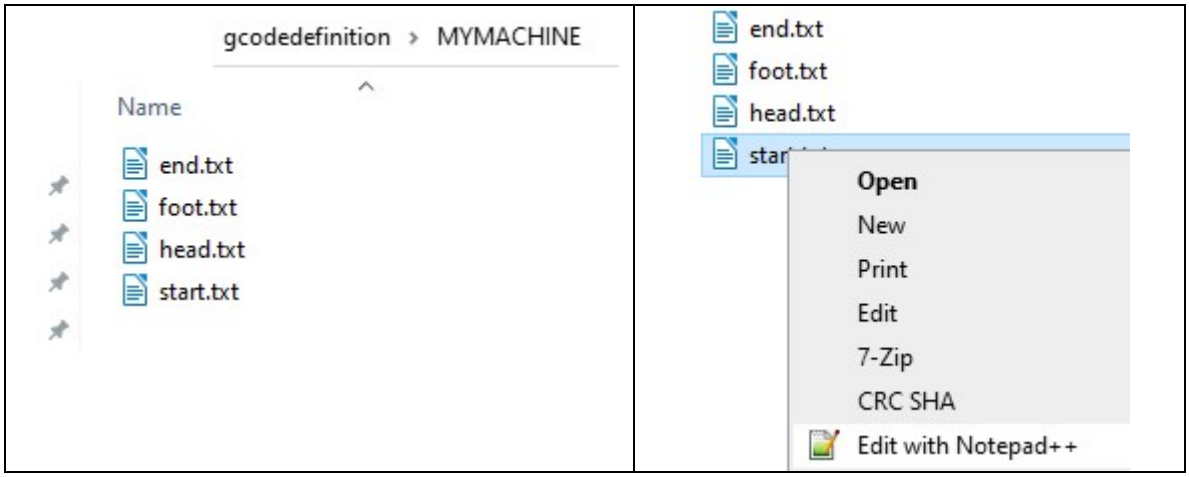
File head.txt will be inserted at the beginning of the file, and foot.txt at the end.

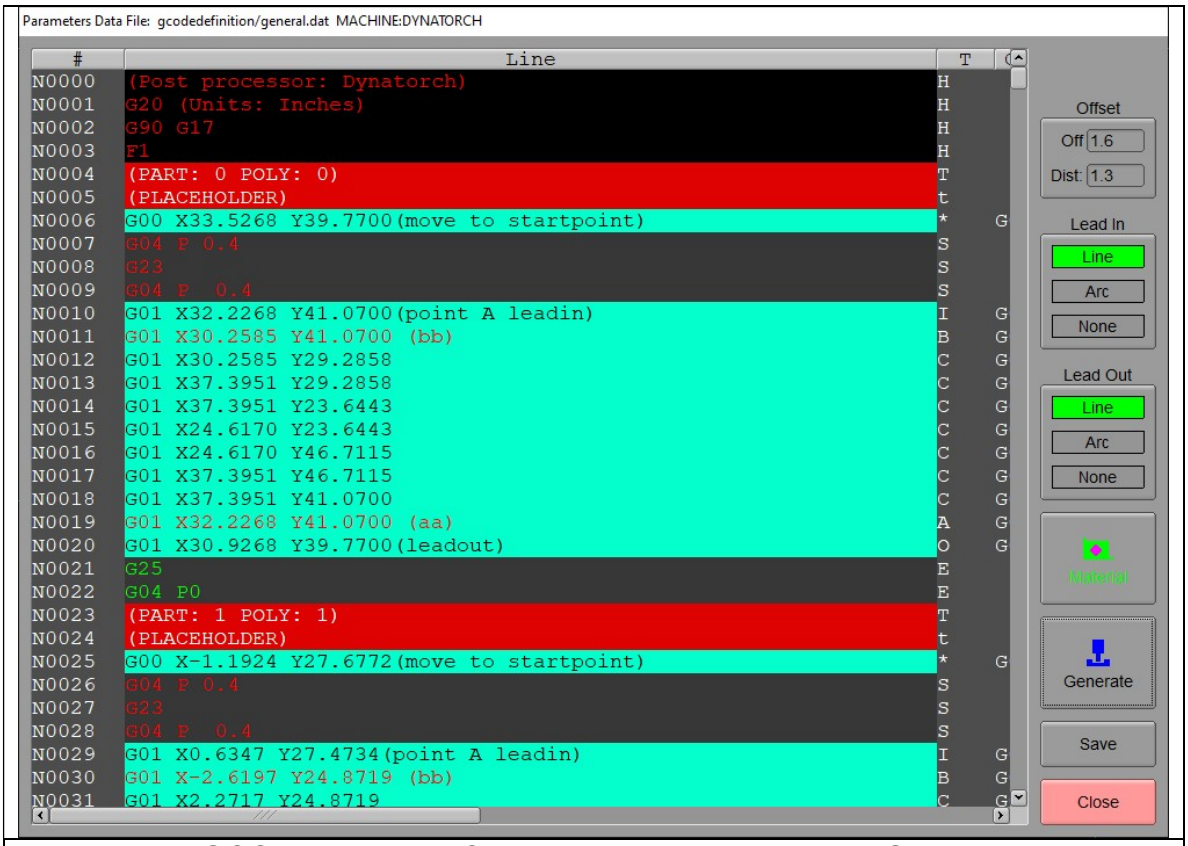
All variables from material file will be replaced.



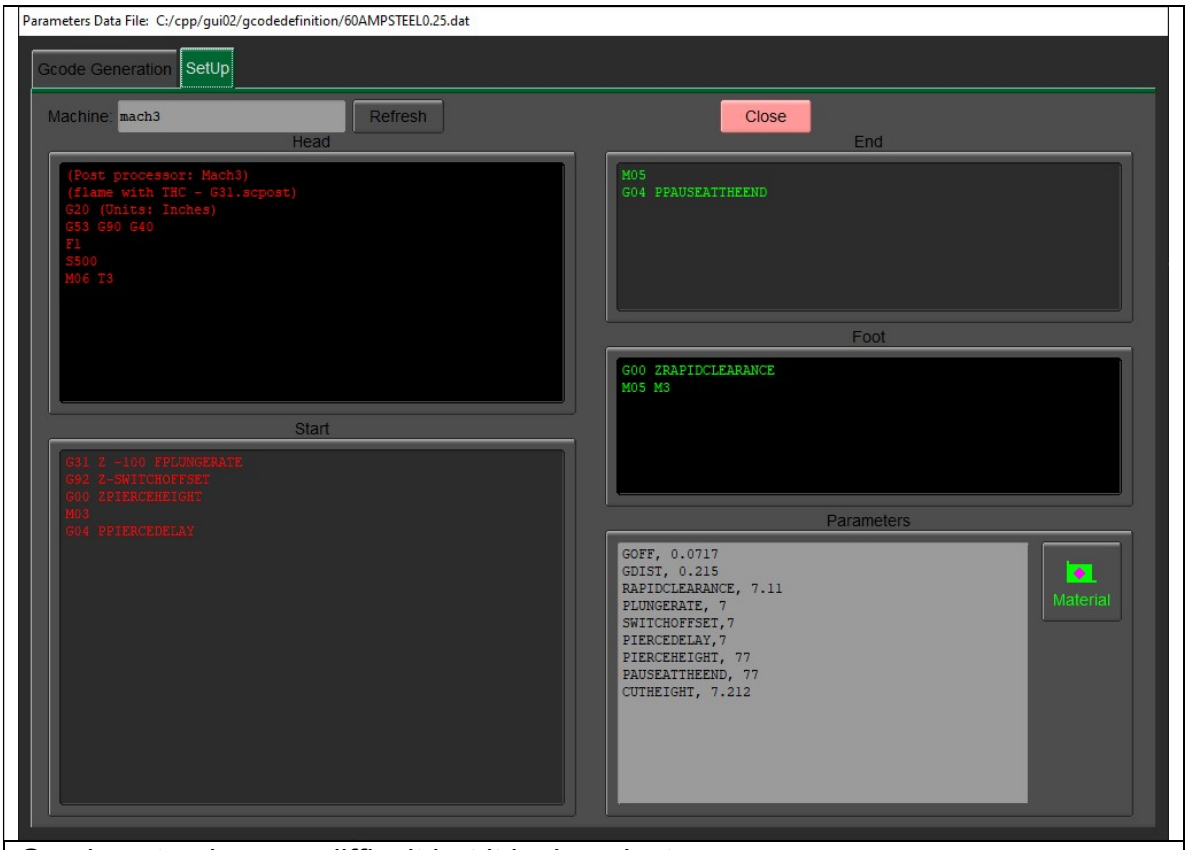
How to create files for new machine


gcodedefinition
Name
MACH3
MACH3 - Copy
simple
60AMPSTEEL0.25.dat
general.dat
machine.txt
cpp
gui02
gcodedefinition
Name
MACH3
MYMACHINE
simple
60AMPSTEEL0.25.dat
general.dat
machine.txt


gcodedefinition
MYMACHINE
Name
end.txt
foot.txt
head.txt
start.txt
end.txt
foot.txt
head.txt
Open
New
Print
Edit
7-Zip
CRC SHA
Edit with Notepad++



To generate GCODE is simple Choose all options and press Generate

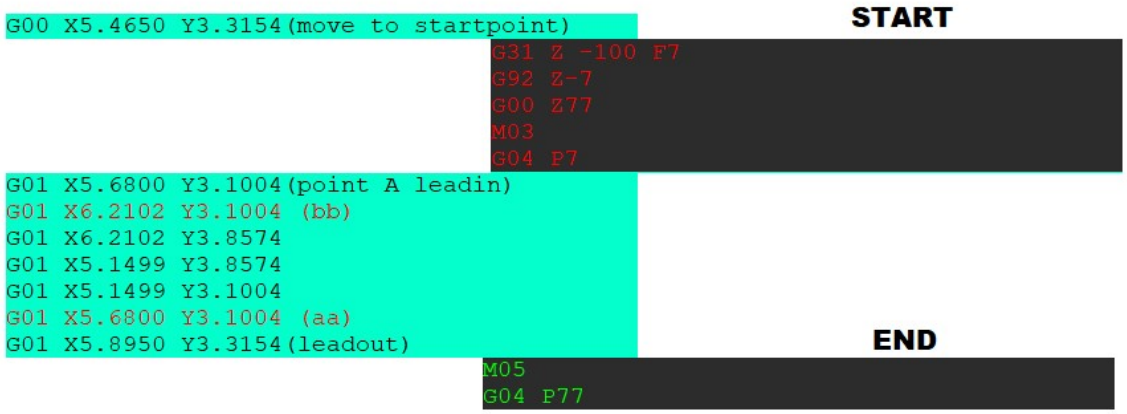


Gcode setup is more difficult but it is done just once

There is 4 files: head.txt, foot.txt, start.txt and end.txt

head.txt is a header foot.txt is a footer
and it is inserted at the beginning of new created nc file

For each polyline Gcode is inserted this way:



START.TXT

```
G31 Z -100 FPLUNGERATE
G92 Z-SWITCHOFFSET
G00 ZPIERCEHEIGHT
M03
G04 PPIERCEDELAY
```

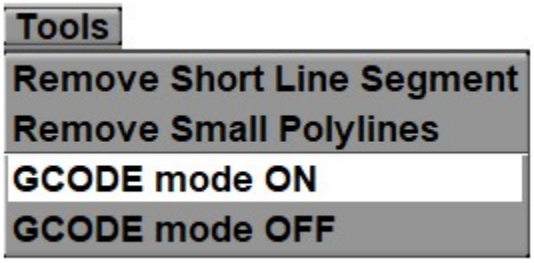
60AMPSTEEL0.25.dat

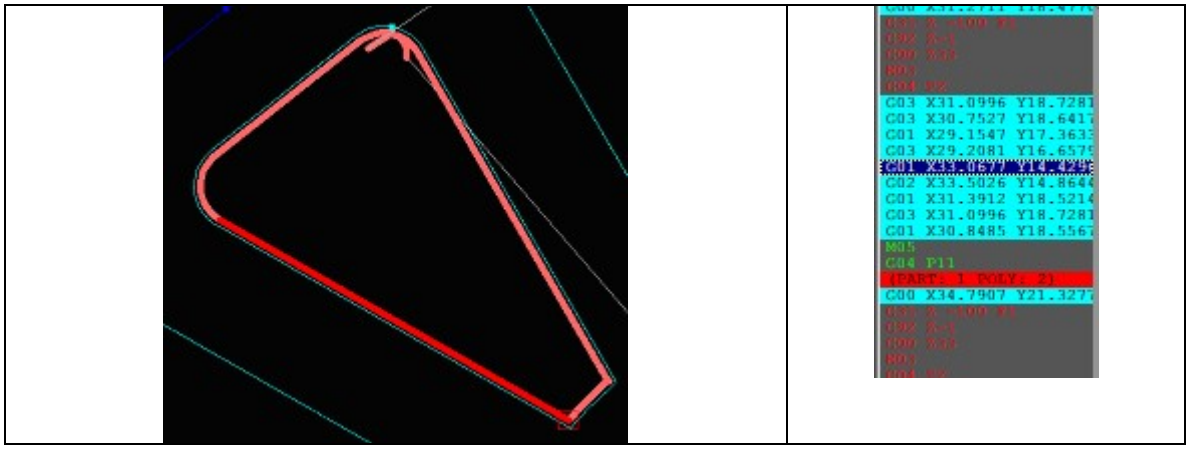
```
GOFF, 0.0717
GDIST, 0.215
RAPIDCLEARANCE, 4
PLUNGERATE, 8
SWITCHOFFSET, 2
PIERCEDELAY,7
PIERCEHEIGHT, 0.1
PAUSEATTHEEND, 1
CUTHEIGHT, 0.1
```

GENERATEDGCODE.NC

```
N0009 G31 Z -100 F8
N0010 G92 Z-2
N0011 G00 Z0.1
N0012 M03
N0013 G04 P7
```

To preview GCODE choose GCODE mode ON on Tools menu





Use arrow keys on right side list box to see commands one by one