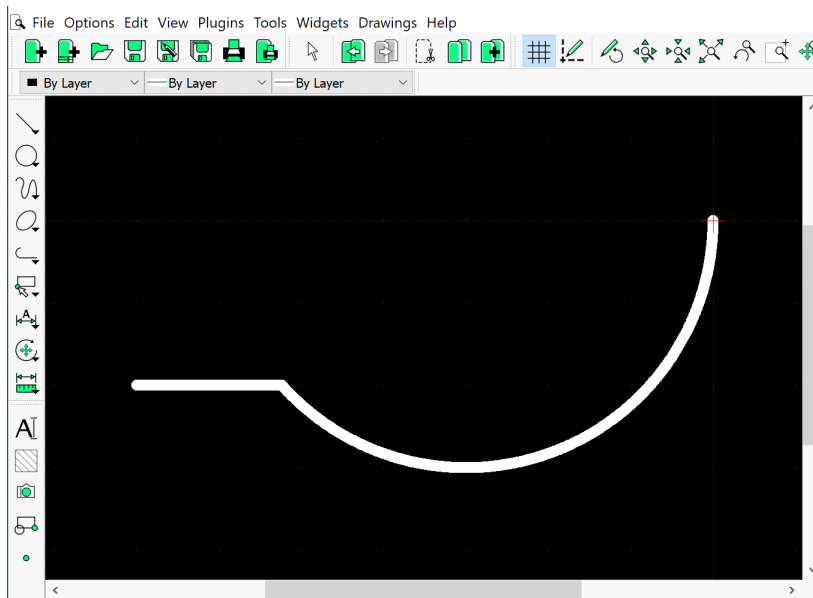


Generate Lathe Tool Path for ezNC

Using GrblGru V5.1

Generate DXF file

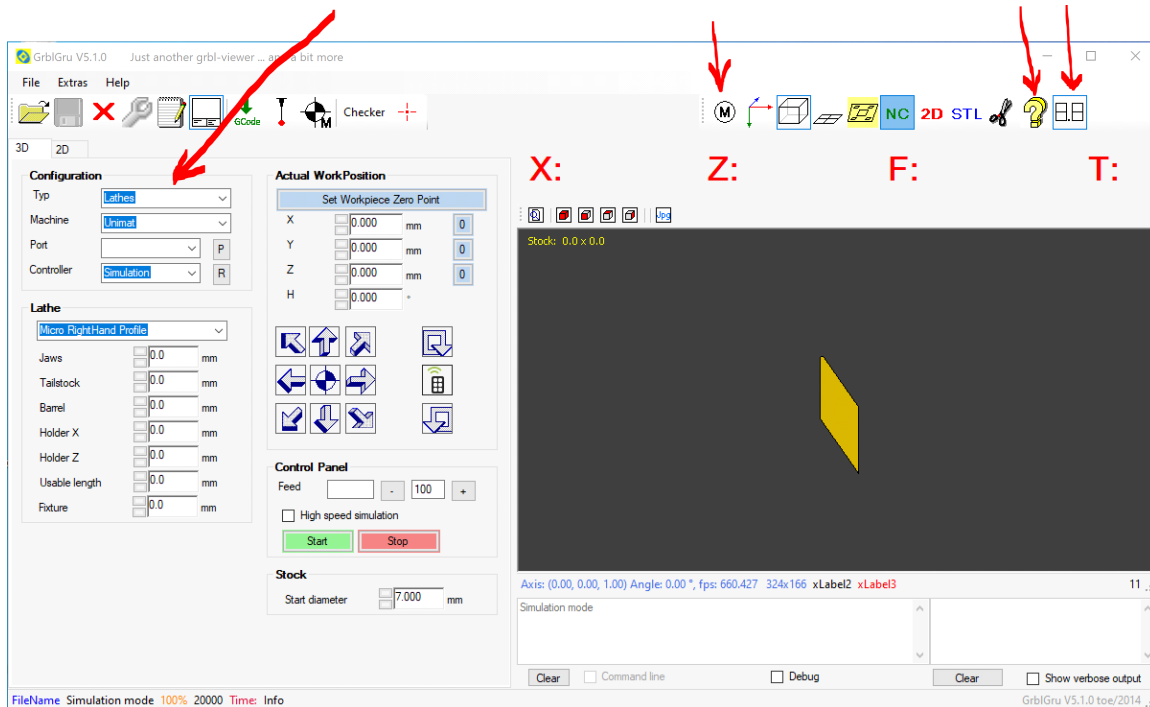
Use your favorite 2D CAD tool to draw and export as DXF file. The object center tip should be at (0,0), as show below which is a 6mm diameter ball tip. It's recommended to draw in metric unit, as GrblGru and some other software may not work well in imperial mode.



GrblGru V5.1 Lathe Configuration

Refer to the arrows in the figure below, left to right.

1. Select "Lathe" and "Unimat" as it's size is similiar to Sherline and Taig lathes.
2. Use icon with letter M to toggle machine display.
3. Use icon with question mark to toggle display of help message.
4. Use icon with square boxes (next to question mark) to toggle large text DRO.

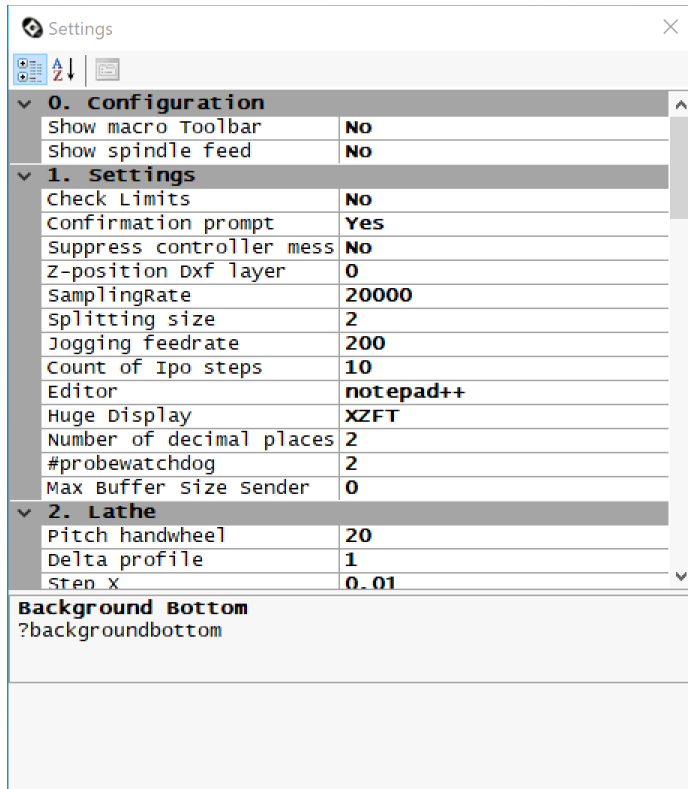


[Optional] Install Notepad++ for Windows

Sometime, one may need to post-process g-code. You can use any tool that can do advanced search and replace, or write a script to convert the file. It is recommended to install Notepad++ as well as g-code mode extension, which has nice syntax high-light.

Download Notepad++ at <https://notepad-plus-plus.org/downloads/>

- Click on "view->zoom in/out" to change font size of the text file.
- Download the "gcode.xml" file from <https://github.com/robEllenberg/gcode-syntax>
- In Notepad++, use **Language->User Defined Language** to load the downloaded gcode.xml. You may want to edit "gcode.xml" file to include .ngc as g-code file extension as well. Restart Notepad++.
- Refer to the figure below. In GribGru, click on **Extra->Settings**, set **Editor** to "notepad++". Lower case works fine.

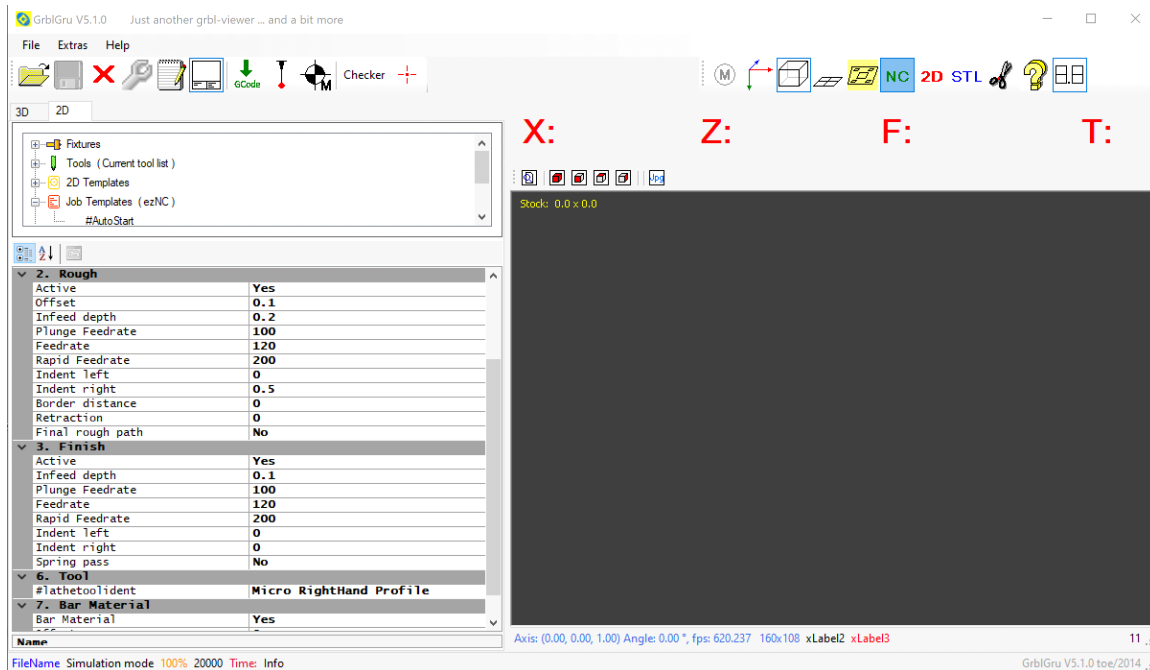


Configure GrblGru with User Defined Tools and Job Parameters

Copy files ezNC.tool and ezNC.job to your GrblGru **DataBase** directory. Mine is at **C:\ProgramData\toe\grblgru_v5.1.0\DataBase**, which is accessible from QuickStart or Search, but not directly from File Explorer starting at C:\. If you can not locate GrblGru's DataBase directory, save it in your home directory and instruct GrblGru to locate them accordingly.

In GrblGru, click on **2D** tab (left hand side, below the floppy disk icon), then right-click on the **Tools** field, then **Load...**, select "ezNC.tool" which defines both right-hand and left-hand micro-tools like those available from Tryally.

Similiarly, right click on **Job Template**, load "ezNC.job" which desines a #AutoStart job that will be invoked whenever a DXF file is imported.

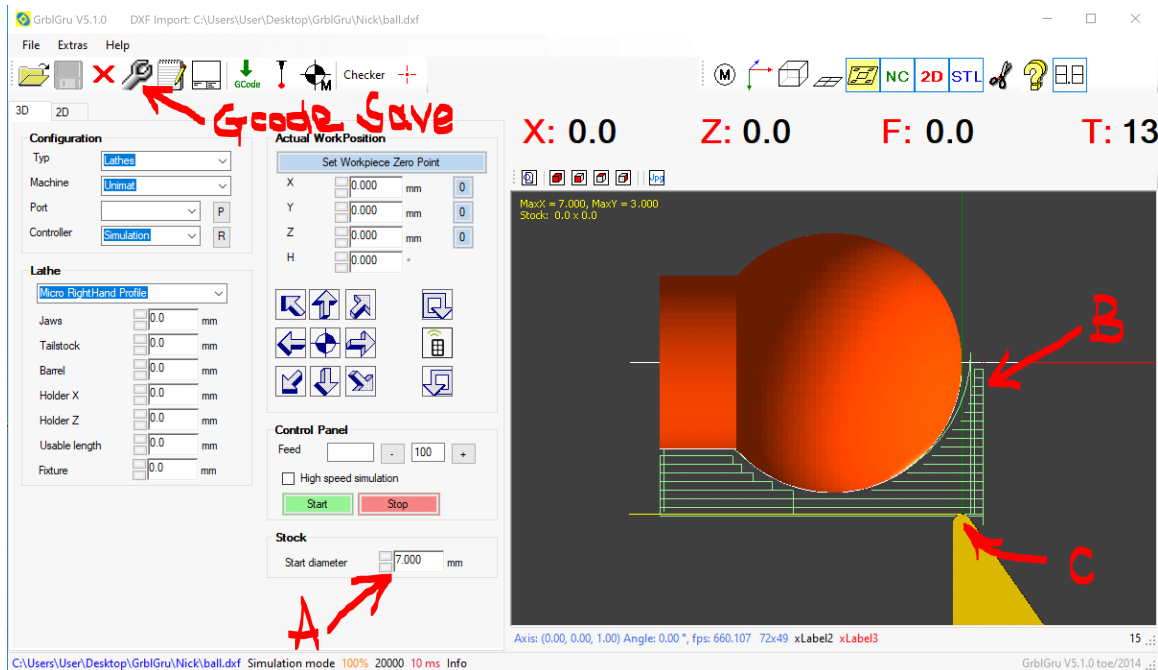


You will most likely change the Job parameters later depends on the type of work, stock material, and your available tools. It is recommended to save your commonly used job definitions under different file names for future use.

Import DXF File

Switch to **3D** tab. Some may prefer the 3D view, but the author finds that top view in 3D gives the best overall picture of the job at hand.

Set the **Stock diameter** (marking A in below figure) first. The example below uses a stock diameter of 7mm. Then click on **File->Import DXF** to import your design. With the tools and job template loaded as earlier steps, the screen should look like the following after the import. Note that GrblGru automatically set X/Z origin at the lower right hand corner of the stock (marking C). Since there is a non-zero **Right Indent** entered in **Roughing** (in file ezNC.job), the roughing cuts (green straight lines) start a little bit to the right (Z>0, marking B), while the finish cut (green curve) starts at Z=0 as the **Right Indent** filed in **Finish** is zero.



To inspect the generated g-codes, click on the wrench icon near top left (marking "Gcode Save" in the above figure). This raw g-code file will run on ezNC/Grbl, but we need one more step for Candle-Lathe to recognize the tool paths. The best solution is to update Candle code, but post processing is much easier at the stage.

Post-process g-code

The raw g-code from GrblGru (in lathe mode) has no information on Y position, which is not used in lathe mode anyway. But it confuses Candle enough to cause it to hide the tool path.

To fix that, click on the **Edit/Save g-code icon** (icon with a wrench) in GRBLGru. Scroll down to the line after first G1 command, which is at line #32 in the figure below. Note that it's very easy to spot with Notepad++ with syntax highlight.

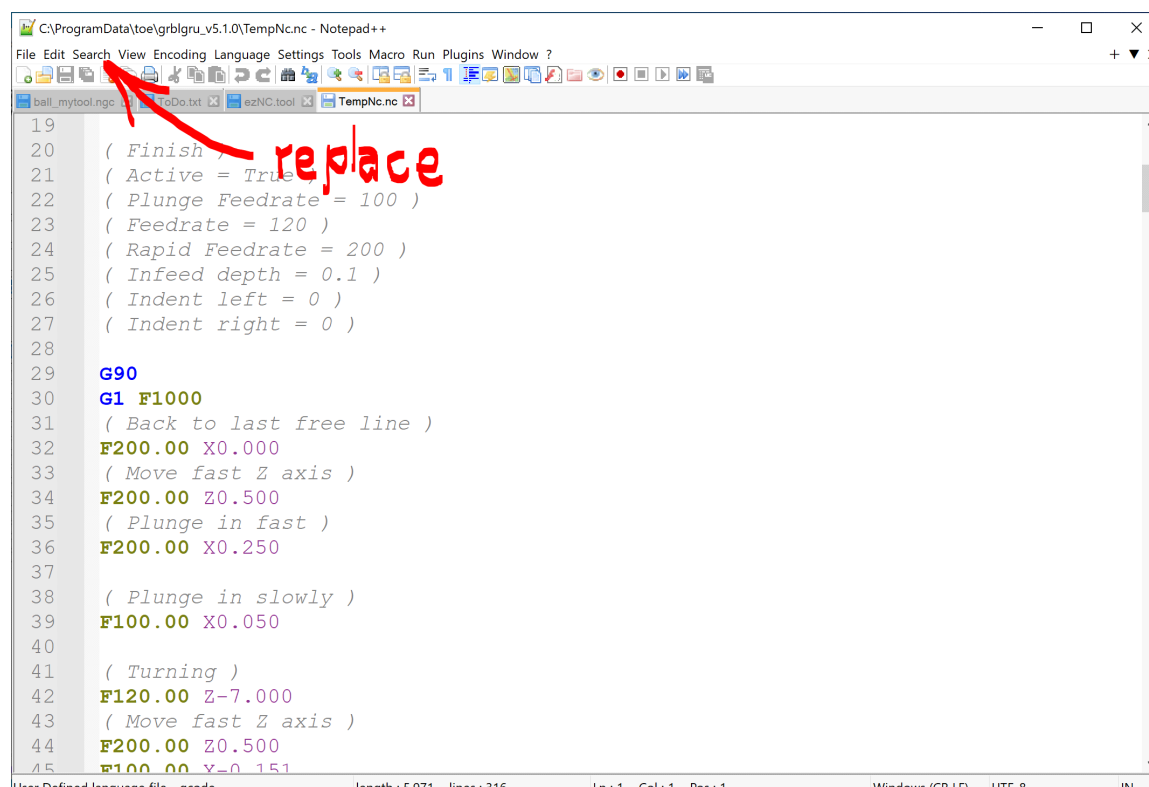
Append "Y0" to the line (#32) with "F200.0 X0.000". The new line should be "F200.0 X0.000 Y0".

Save the modified file, and copy it to Raspberry Pi or any computer running Candle-Lathe. You can also run the g-code directly from ezNC-2. Please consult the **ezNC-2 User Manual** for how to upload g-code.

[For Reference Only] Post-process g-code Using Notepad++

The following steps also fix the Candle not displaying tool path from GrblGru problem. But it is unnecessarily complicated. It's kept here to illustrate the power of Notepad++.

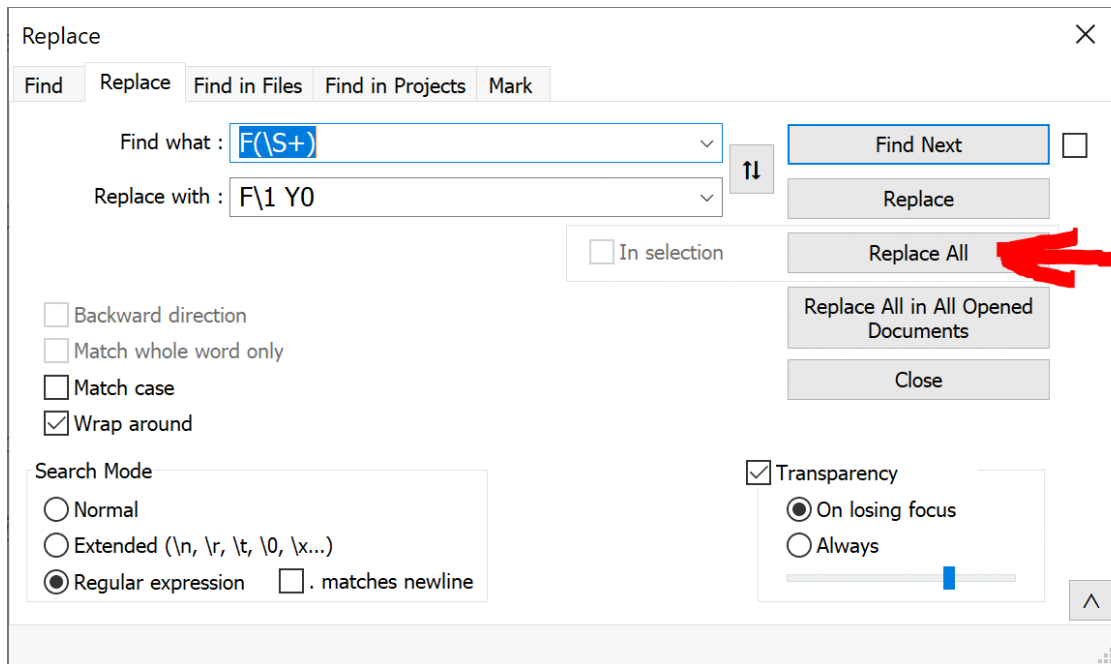
Click on the **Edit/Save g-code icon** (icon with a wrench). Then click on **Serach->Replace** (or simply ctrl-H).



The screenshot shows the Notepad++ interface with a file named 'TempNc.nc' open. The menu bar includes 'File', 'Edit', 'Search', 'View', 'Encoding', 'Language', 'Settings', 'Tools', 'Macro', 'Run', 'Plugins', and 'Window'. A red arrow points from the 'replace' button in the 'Search' menu to the 'replace' button in the 'Edit' menu. The main text area contains g-code with various feed rates and moves. The status bar at the bottom shows 'Line: 1 Col: 1 Doc: 1'.

```
19
20 ( Finish )
21 ( Active = True )
22 ( Plunge Feedrate = 100 )
23 ( Feedrate = 120 )
24 ( Rapid Feedrate = 200 )
25 ( Infeed depth = 0.1 )
26 ( Indent left = 0 )
27 ( Indent right = 0 )
28
29 G90
30 G1 F1000
31 ( Back to last free line )
32 F200.00 X0.000
33 ( Move fast Z axis )
34 F200.00 Z0.500
35 ( Plunge in fast )
36 F200.00 X0.250
37
38 ( Plunge in slowly )
39 F100.00 X0.050
40
41 ( Turning )
42 F120.00 Z-7.000
43 ( Move fast Z axis )
44 F200.00 Z0.500
45 F100.00 X-0.151
```

Enter the replacement rule exactly as shown below then click on **Replace All**. This will append string " Y0" to any non-white space text starting with upper case letter 'F'. Reads on Notepad++ or regex for more details.



Finally, save the processed file as .gc file. Copy it to Raspberry or any computer running Candle-Lathe. Turn on spindle motor, touch off the tool tip at corner of the stock. If not at X=Z=0, set as zero position on ezNC. Then start the cut, happy ezNC-ing.

Quirks of GrblGru

One may notice that there is no G0 statement from GrblGru. This will limit the rapid travel speed of ezNC, and will be addressed in the future -- with post-processing.

Tool definition with radius < 0.2mm seems to cause GrblGru V5.1 to produce too much x-offset in finish path.

After setting a parameter of **Finish** cut, the entries on 2D tab is refreshed and displayed from the beginning. One needs to scroll down to set other parameters for **Finish**. Without scrolling down, one may be editing the parameters for **Rough** cut instead.