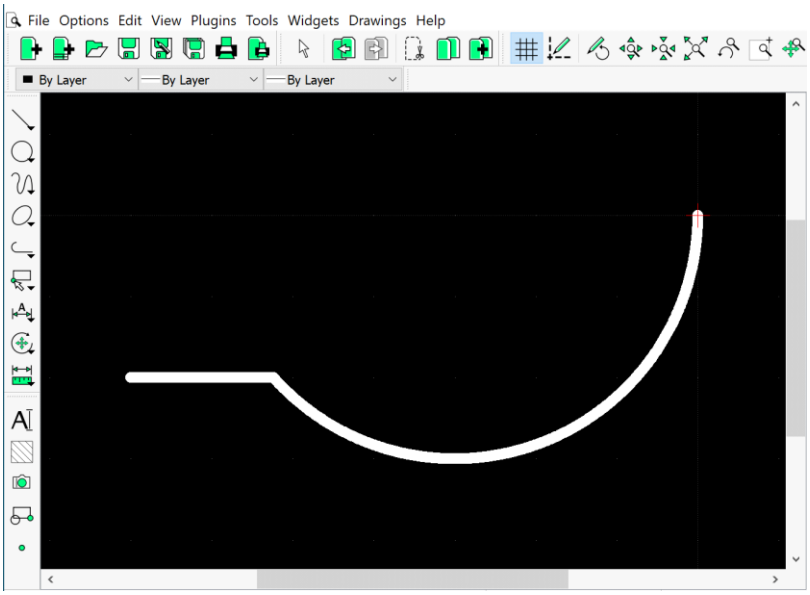
Generate Lathe Tool Path for ezNC2

Using GrblGru V5.1

Generate DXF file

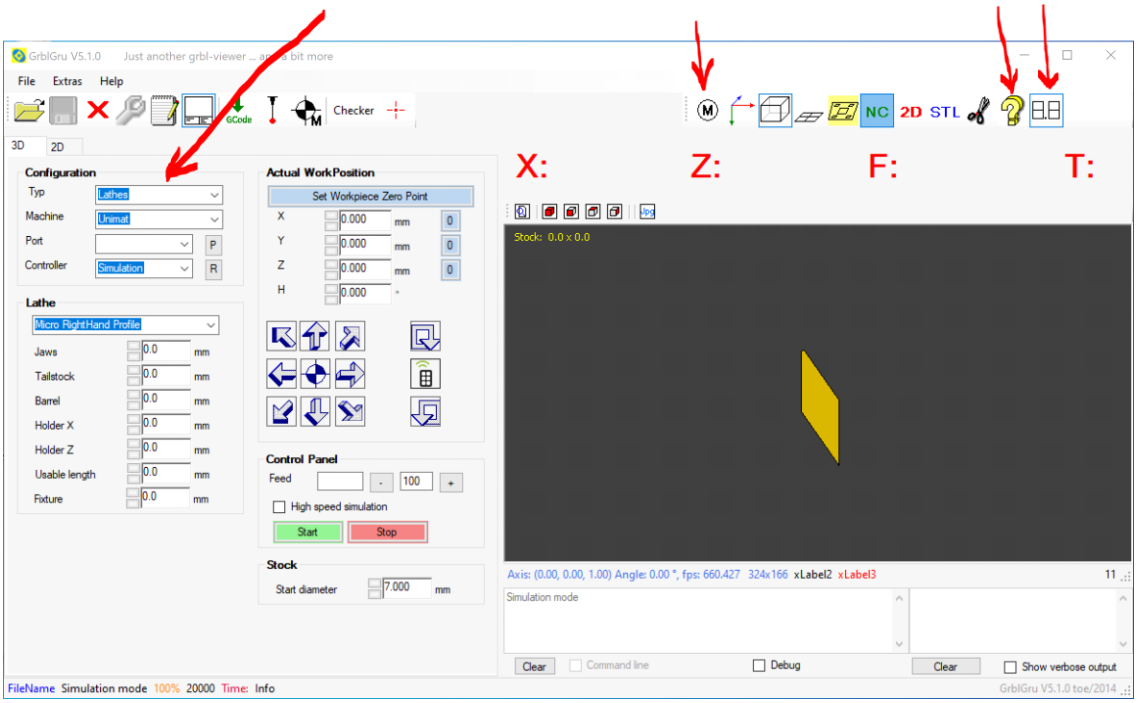
Use your favorite 2D CAD tool to draw the lower half of the desired object and export as DXF file. If you don’t have one in mind, LibreCAD is free, open sourced, and fairly easy to use. The figure below shows an example of a ball end with 6mm diameter. The center tip of the object should be at (0,0), as show in the figure. It's recommended to draw in metric unit (mm), 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 righ:

1. Select "Lathe" and "Unimat" as it's size is similar 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.

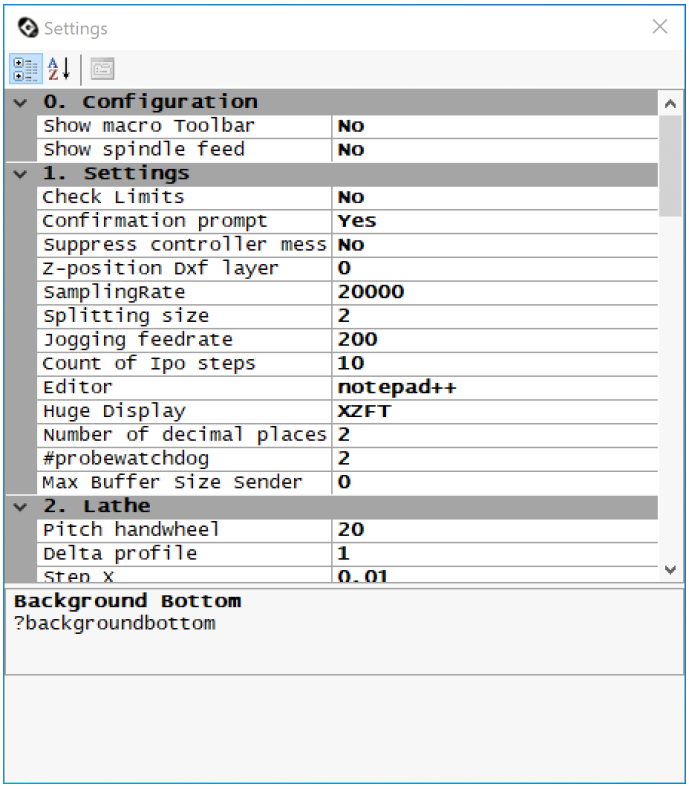


[Optional] Install Notepad++ for Windows

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 syntex 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 GrblGru, click on **Extra->Settings**, set **Editor** to "notepad++". Lower case works fine.

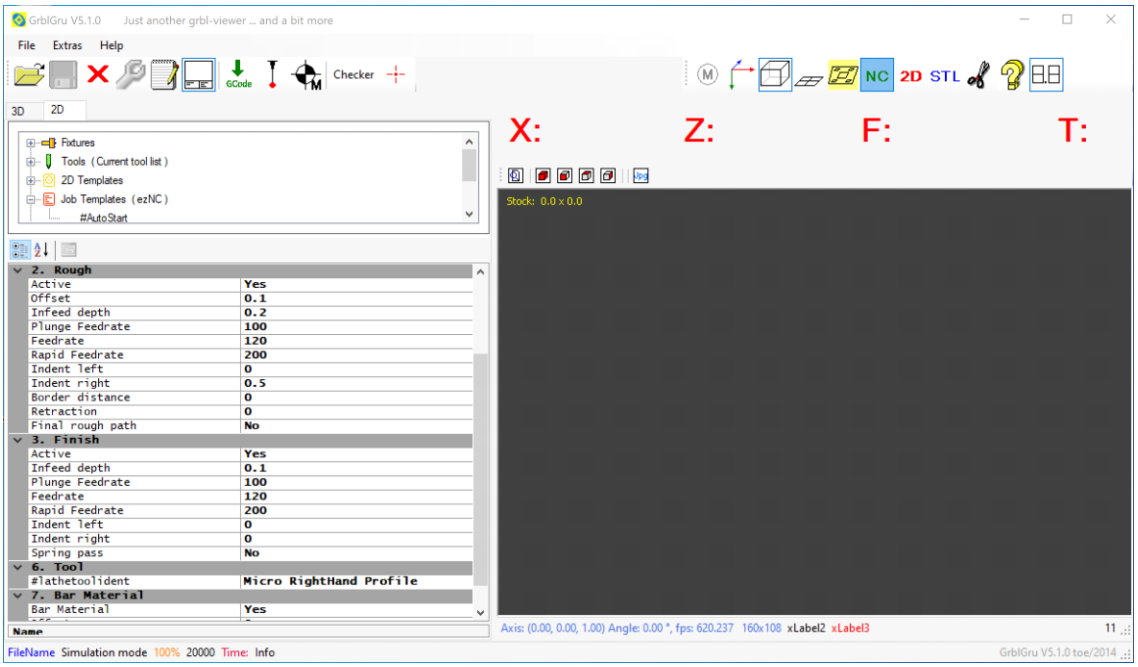


Configure GrblGru with User Defined Tools and Job Parameters

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 cannot locate GrblGru's DataBase directory, save it in your home directory and instruct GrblGru to use the one in your home directory.

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 Tech.

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



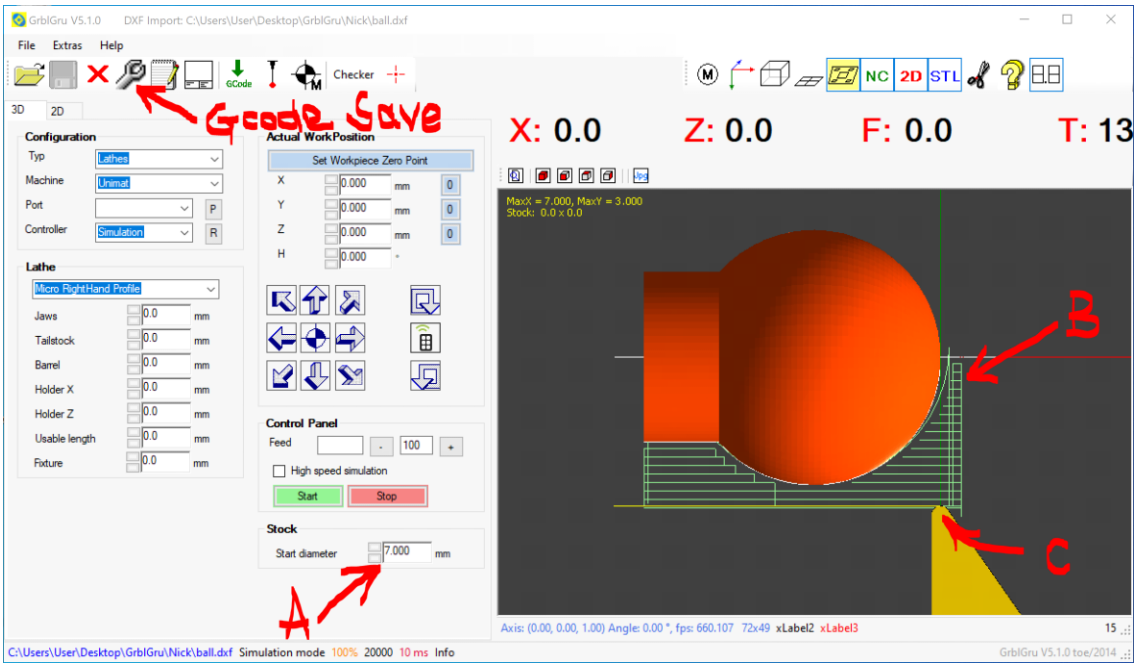
You will most likely change the Job parameters later depends on the type of work, stock material, and your available tools. It is recommended to save your commonly used job definitions under different file names for future use.

Import DXF File

Switch to **3D** tab (left hand side, below the file folder icon). Some may prefer other view angles, 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 the figure below) 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.



Next step is 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 may need one more step for older Candle-Lathe to recognize the tool pathes. The following post-processing is only needed for original Candle or older version of Candle-Lathe. You can load the file into Candle-Lathe, if there is no (green tool path) after loading the g-code file, then you need to either update Candle-Lathe or perform the following simple post-processing.

Post-process g-code [only needed for older Candle-Lathe]

The raw g-code from GrblGru (in lathe mode) has no information on Y position, which is not used in lathe mode anyway. Althyough the g-code runs fine with ezNC2, Candle or older Candle-Lathe are confused by it and will not display the tool path correctly.

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 this line with Notepad++ due to the colorful syntax highlights.

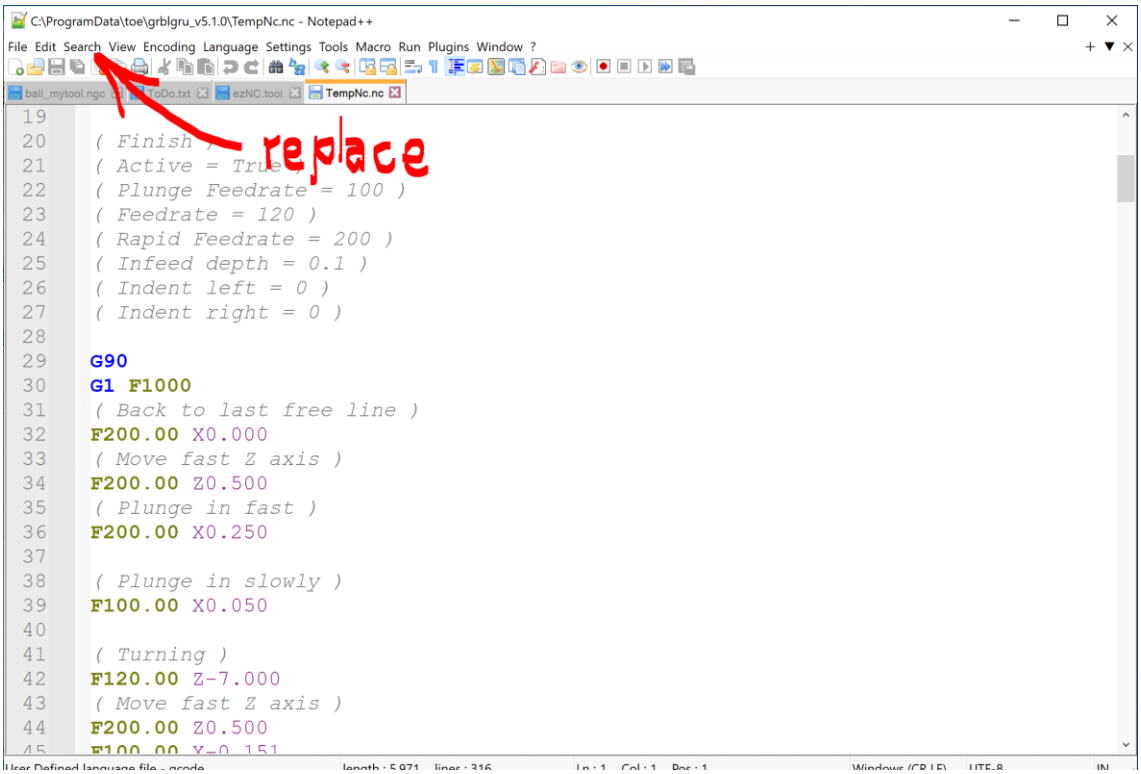
Append "Y0" to the line (#32) with "F200.0 X0.000". The new line should be "F200.0 X0.000 Y0".

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*** on how to upload a g-code file.

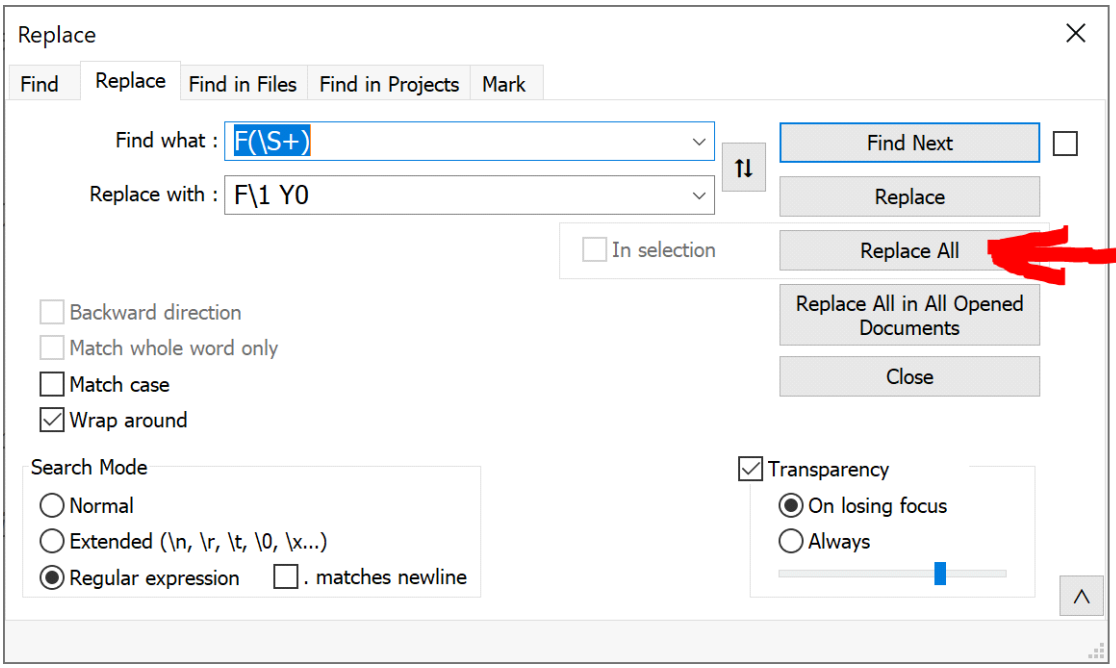
[For Reference Only] Post-process g-code Using Notepad++

The following steps also fix the problem, i.e. Candle not displaying tool path from GrblGru. However, it is unnecessarily and a little bit more 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).



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'. Read on or search Notepad++ or regex for more details.



Finally, save the processed file as .gc file. Then copy/transfer it to Raspberry or any computer running Candle-Lathe. Turn on spindle motor, touch off the tool tip at lower right hand corner of the stock. Set Z=0 and X=stock radius on ezNC. Then start the cut.

Quirks of GrblGru

One may notice that there is no G0 statement from GrblGru --- similar to the free version of Fusion360. This will slow down the machine throughput a little bit.

Tool definition with radius < 0.2mm seems to cause GrblGru V5.1 to produce too much x-offset in finish path. Don't do that.

After setting a parameter of **Finish** cut, the entries on 2D tab is refreshed and displayed from the beginning. If you have a small screen, you will need to scroll down to set other parameters for **Finish**. Without doing so, one may be editing the parameters for **Rough** cut instead of **Finish** cut.

The following item is not really a "quirk" of GrblGru. It should be called "ill-defined" boundary condition. If you have a work piece supported by a center at the tail end, double or even triple check the generated g-code. You may have to add more details (i.e. constrains) at the tail end to avoid cuts that are way too aggressive.