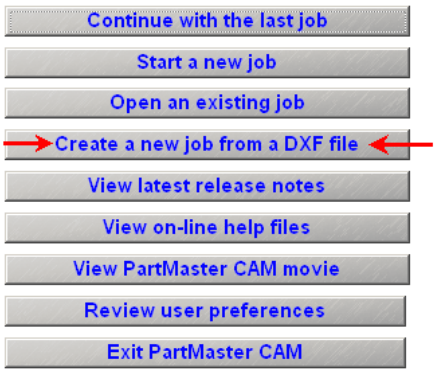
The basic steps for using Dolphin Lathe are as follows:

1. Run Partmaster Cam
2. create a job from the dxf
3. select a dxf file
4. define the importing parameters
5. define what you want shown in the graphics display
6. SETUP the machine
7. Machining the profile:
   - create a tool
   - select the tool
   - select the type of operation to be done
     Some options need to be defined so that the tool starts at the right place, does the cutting, the tool exits the operation to some point
   - run the operation to see the pathing that will be created and correct as needed
8. Repeat # 7 for the next operation
9. Post the code to a file

There is a quick tutorial at the end for multiple cutting along a profile.

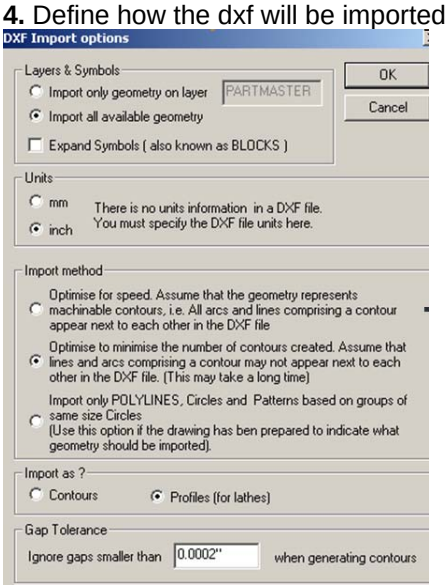
---

1. & 2. Open PartMaster Cam and from the menu select

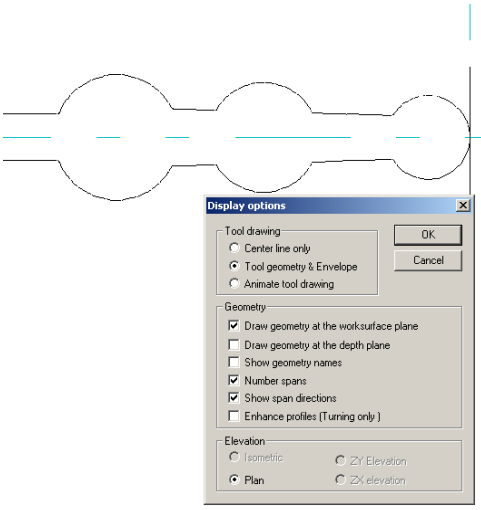


3. When the window opens just select the dxf file.

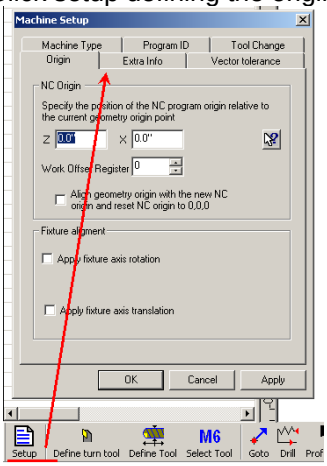
4. Define how the dxf will be imported into Dolphin.



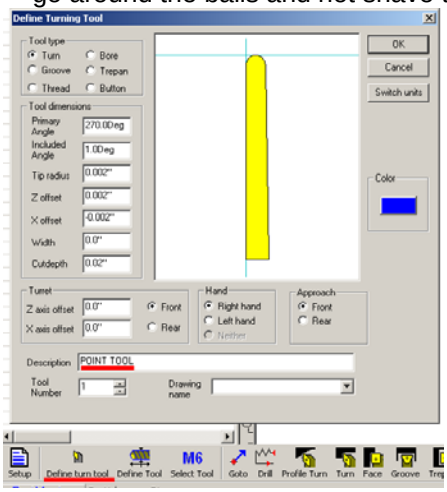
5. Right click in the graphics screen and define the display.



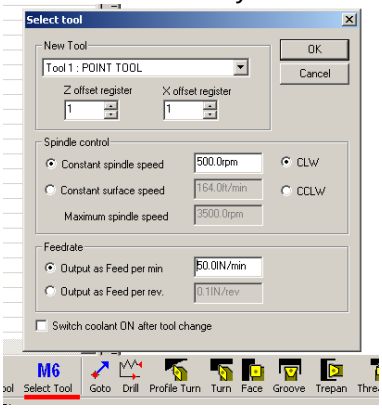
6. Click setup defining the origin location.



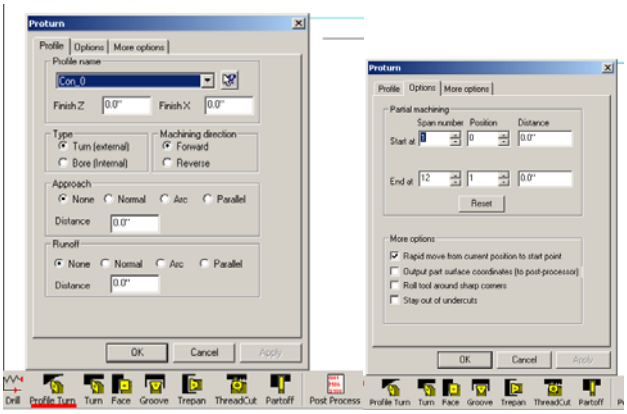
7. Now define a tool. In this case I created a “point” tool which is very sharp and narrow since it needs to go around the balls and not shave the ball on the back side of the tool when cutting.

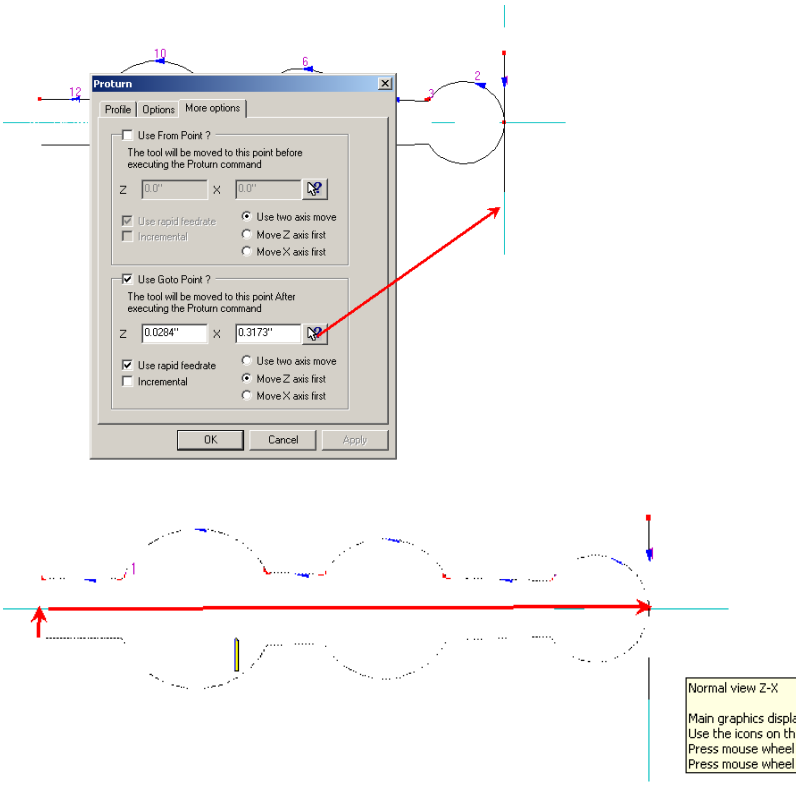


Now select the tool you will use for the machining task.

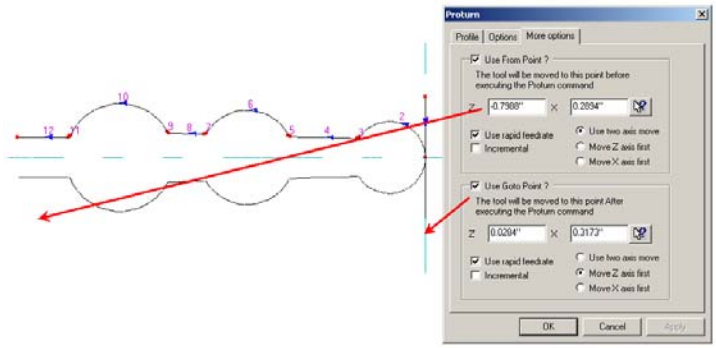


Now select what the machining task is and in this case a profile cut is wanted.
You need to make selections in options and more options as shown below.
Here are some examples based on the different selections and the resulting pathing.

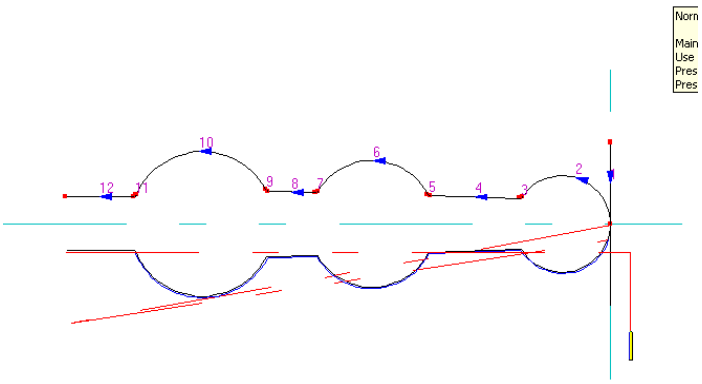




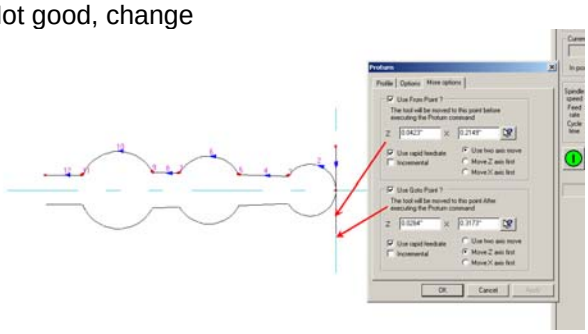
The above did the profile, but, it also returned to 0,0 and cut thru the piece
So change the settings for that machining task.



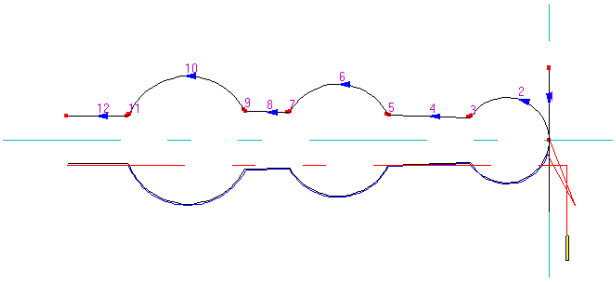
Pathing result below



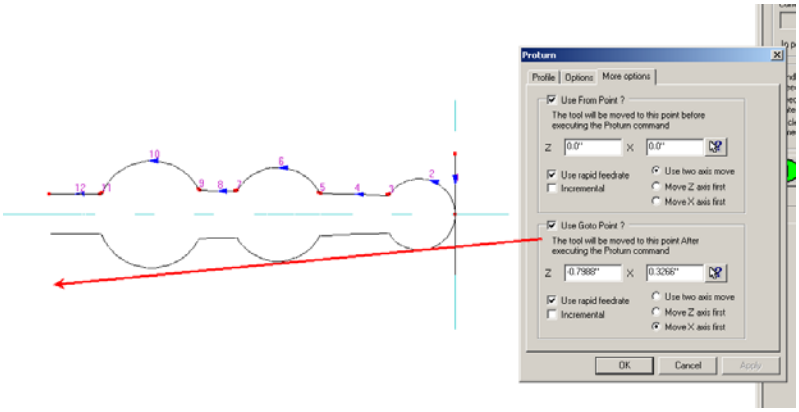
Not good, change



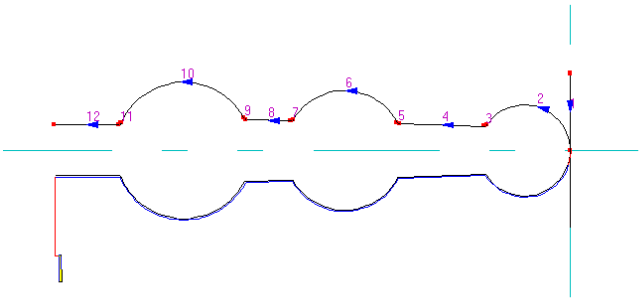
Pathing result



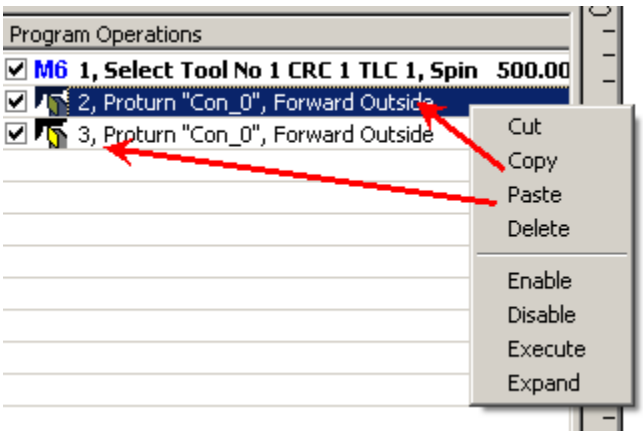
Change option again



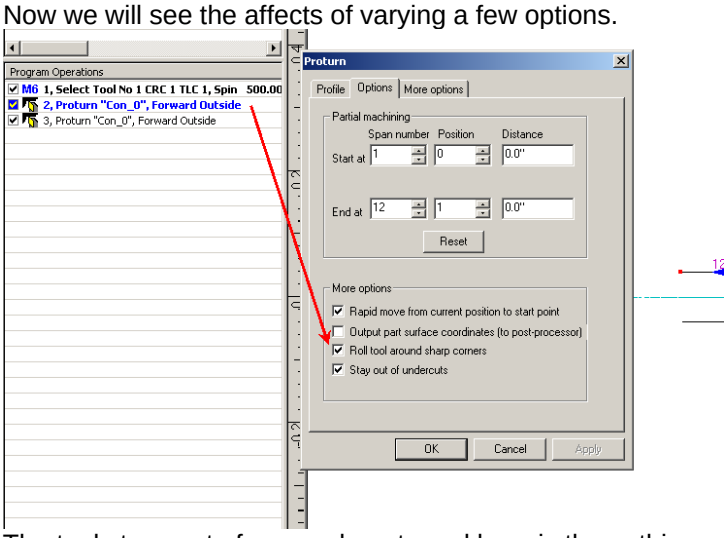
Pathing for the profile and the tool is pulled out from the stock based on options chosen.



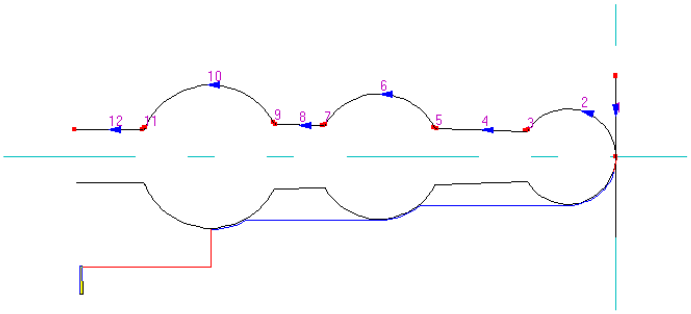
The above were about getting a proper pathing of only the profile.
You can copy any machining task.

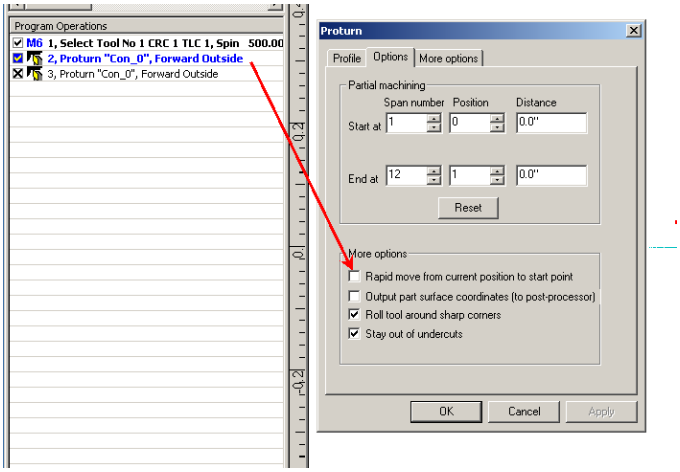


Now we will see the affects of varying a few options.

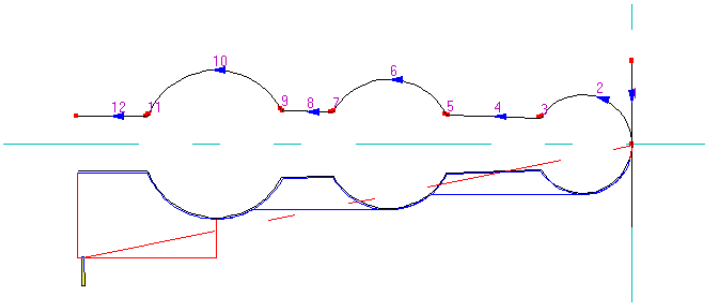


The tool stays out of any undercuts and here is the pathing.

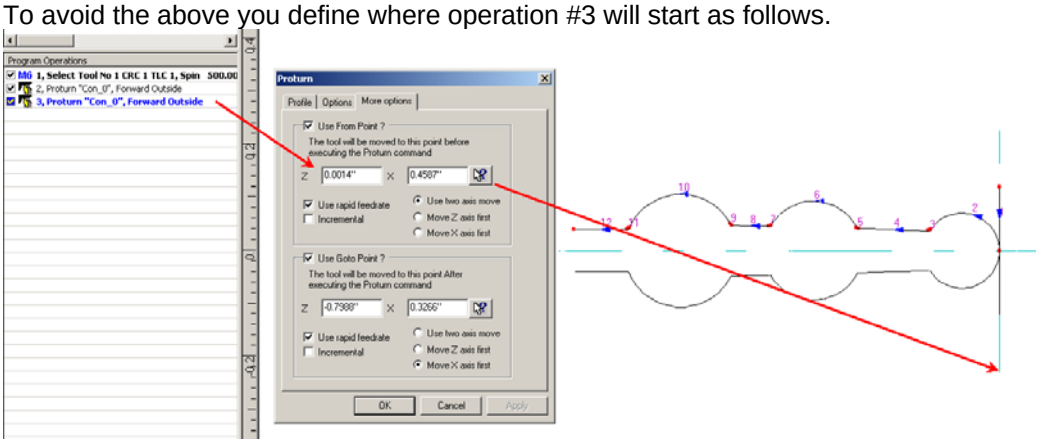




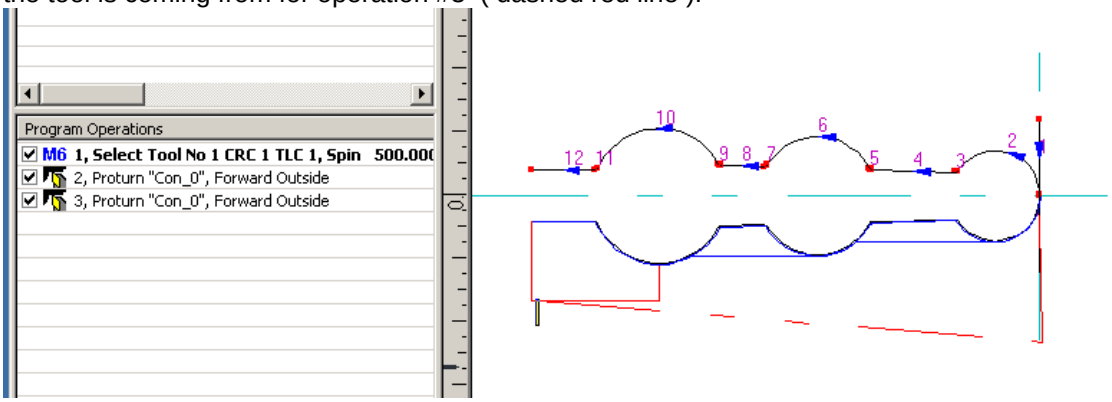
The tool rapids back to 0,0 for operation #3 and cuts thru the profile.



To avoid the above you define where operation #3 will start as follows.



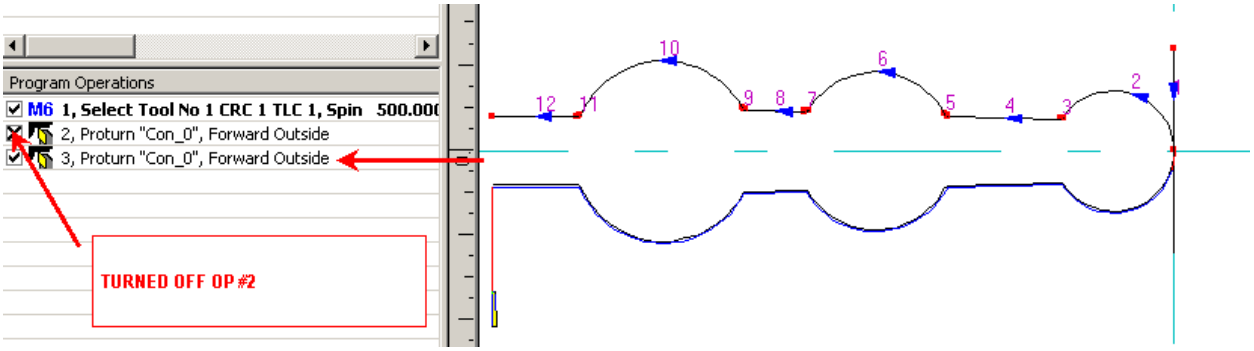
Now operation #2 finishes at where that tool is and then we defined a start point of where the tool is coming from for operation #3 (dashed red line).



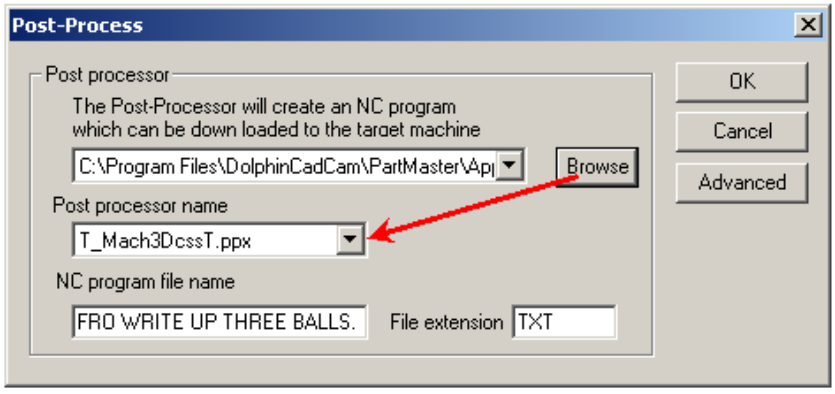
You need to define the start and end points in each machining operation.

9. POSTING & CREATING THE GCODE

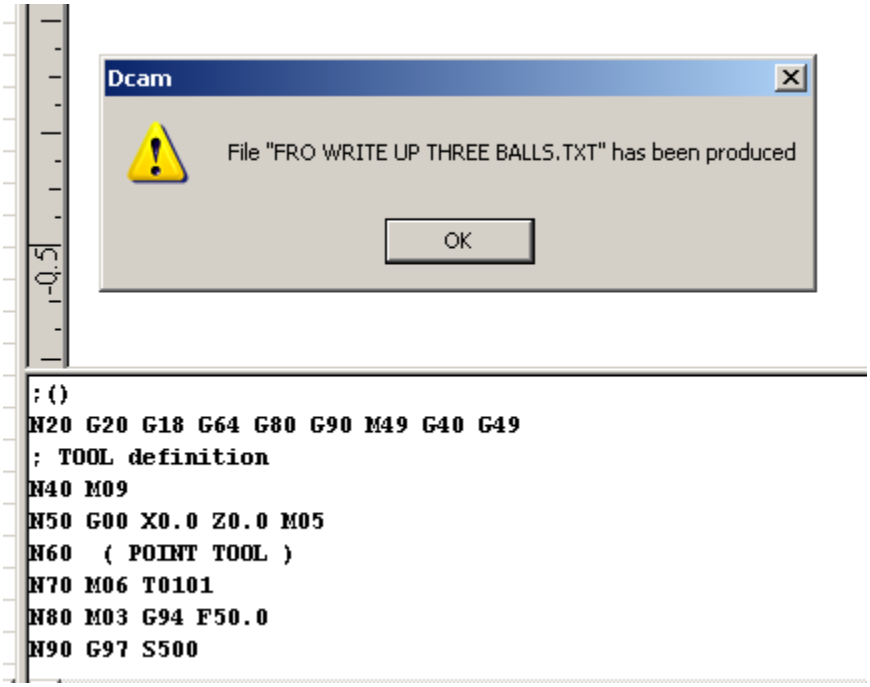
Run the operation for the profile cut. Turn off operation #2 which avoided undercutting.



Click the POST button, the operation will be run, and a window will open so you can select the post processor. Note: draw as a diameter, post as diameter, and have Mach configured for diameter!



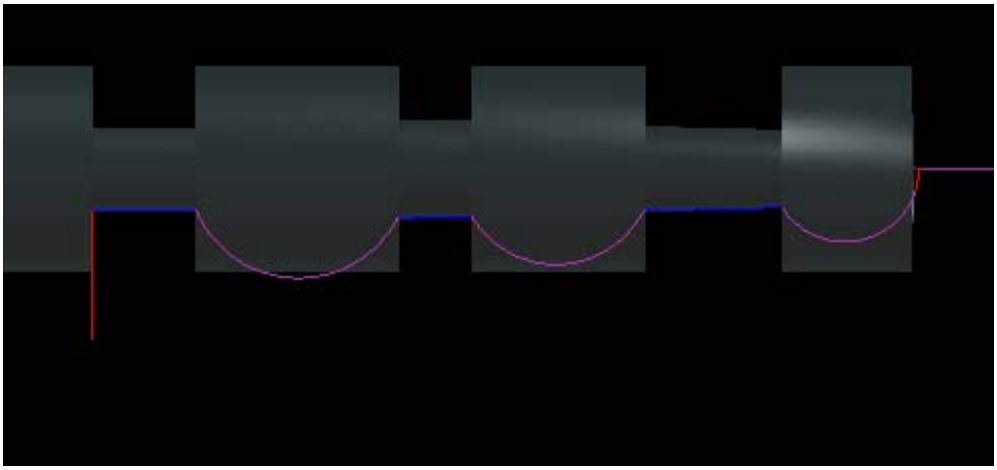
The file is saved or you can also highlight all the code and copy paste into Notepad, etc.



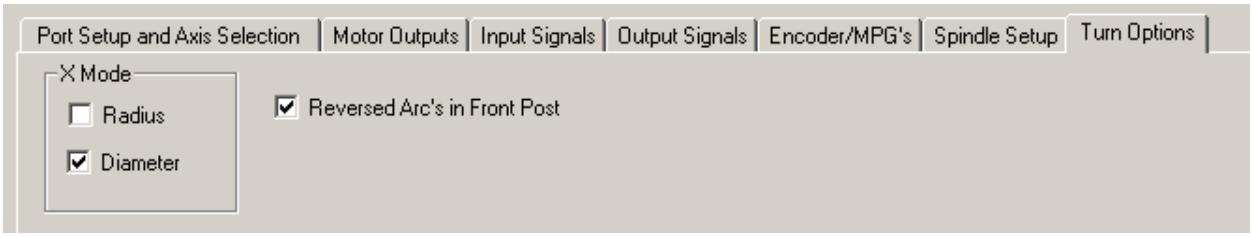
Posted code below:

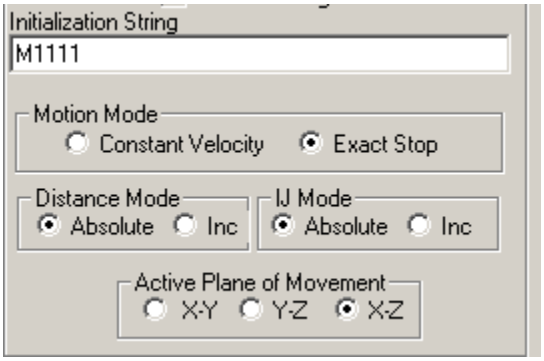
```
;()
N20 G20 G18 G64 G80 G90 M49 G40 G49
; TOOL definition
N40 M09
N50 G00 X0.0 Z0.0 M05
N60  ( POINT TOOL )
N70 M06 T0101
N80 M03 G94 F50.0
N90 G97 S500
N100 G00 X0.0437 Z-0.004 G94
N110 G02 X0.0769 Z-0.1338 I-0.002 K-0.073
N120 G03 X0.0751 Z-0.1355 I0.0396 K-0.1354
N130 G01 X0.0827 Z-0.2688
N140 G03 X0.0849 Z-0.2705 I0.0434 K-0.2687
N150 G02 X0.0943 Z-0.4361 I-0.002 K-0.3546
N160 G03 X0.0924 Z-0.4378 I0.0482 K-0.4378
N170 G01 X0.0964 Z-0.5082
N180 G03 X0.0985 Z-0.5099 I0.0502 K-0.5081
N190 G02 X0.0824 Z-0.705 I-0.002 K-0.6055
N200 G03 X0.08 Z-0.7069 I0.042 K-0.7069
N210 G01 X0.08 Z-0.8069
N220 G00 X0.3301 Z-0.8069
N230 X0.3301 Z-0.8059
N240 M05 M30
```

Open the file in Mach3 LATHE



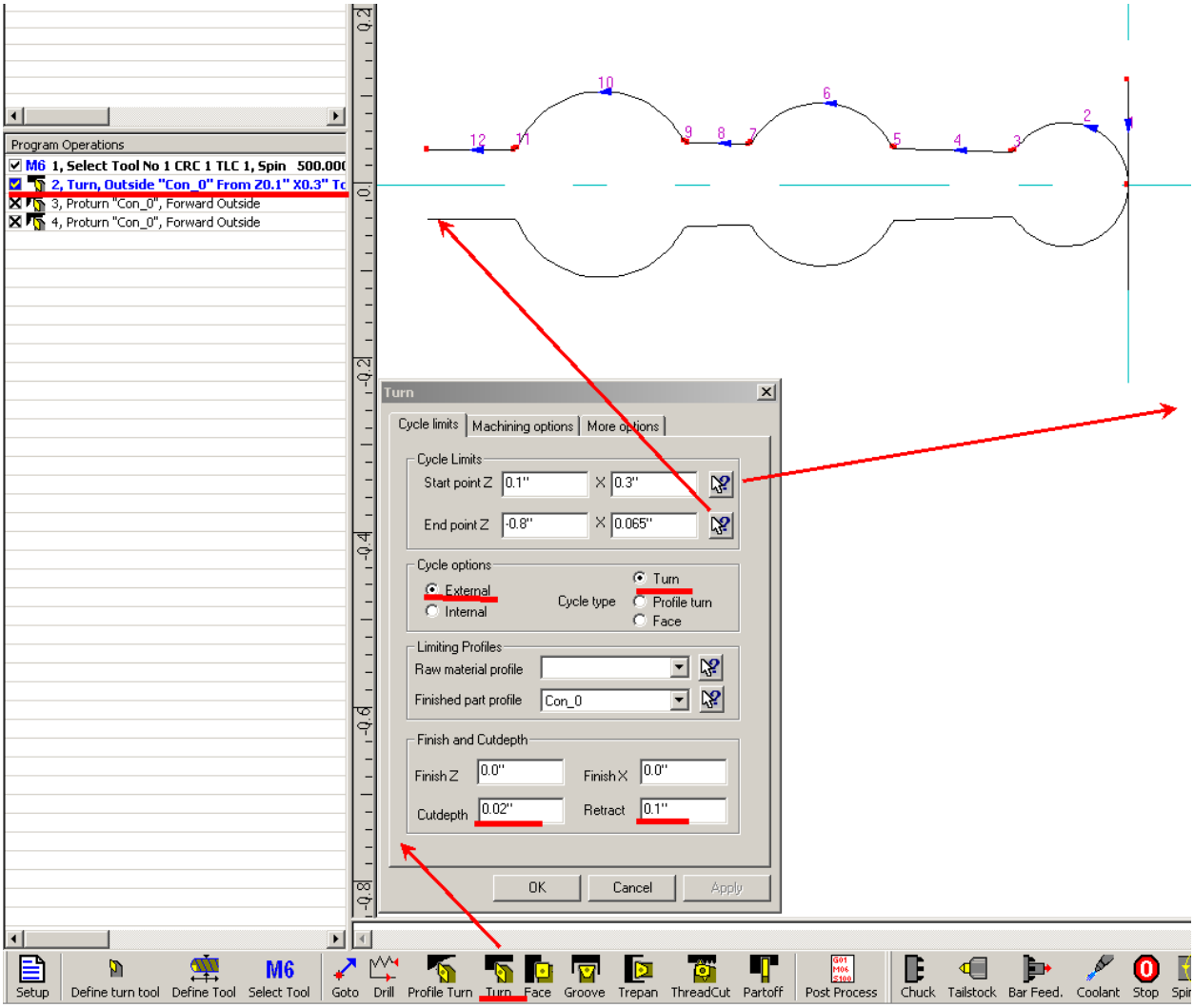
Mach configuration required for code as post. Had to check reversed arcs to get rid of the crop circles.



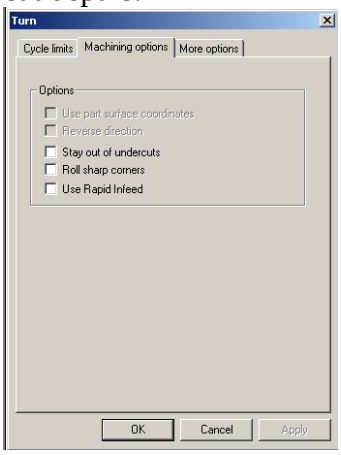


MULTIPLE PATHING / CUTTING

Select Turn operation, and in the fly out define the point you want the pathing to start from and where you want it to end as shown below. Just an un-fancy way you can do this, as the tool can be touched off / set at 0,0 and then move to a point to start the pathing, and the tool will return to that point when done.



No machining options where selected because I wanted to widdle down the profile also include a profile cut. Remember that I am using a “point” tool so all can be done and with very small cut depths.



The generated pathing for progressive cuts down to a zero clearance on the profile.