TTORIAL #5 - LATHE TURNING

PREFACE

This tutorial provides a general overview of the Turn Module. Specific definitions for the Turn Menu were given in Section 13 and not repeated. It is assumed that the reader understands basic concepts previously presented such as layers, chains, entities, and how to navigate around and change the display. Material here is lathe specific. It is highly recommended that the user read thru the LAZYTURN manual and it's appendixes.

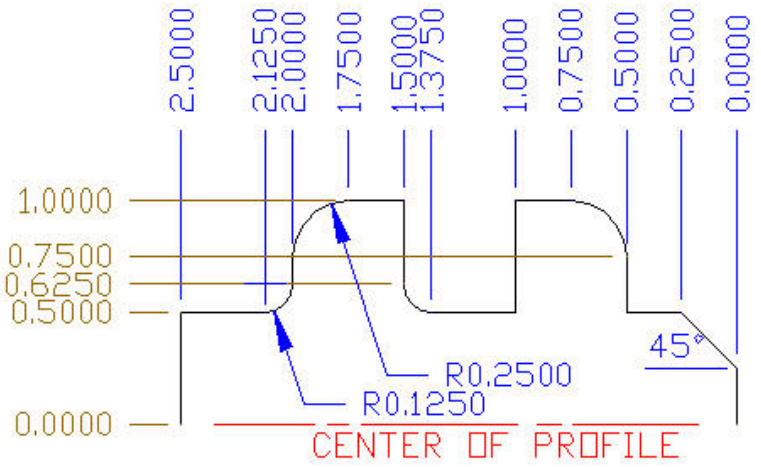
LazyTurn will replace the turn module in LazyCam in the future. There are a number machining operations that LC currently provides for that LT does not. LT does not provide for a finish pass as of this writing. LC can do internally boring, and, in some ways is not as restrictive as LT. This tutorial will show how to get a finish pass from LC quickly and covers basic differences of the Module.

Many of the tools or commands used for the Mill are available, but frankly, don't apply to the lathe. The lathe is rather simple to use.
Advanced techniques are not presented, but, concepts previously given can be appied.

This tutorial uses the same drawing used in the LazyTurn Manual , in particular Section 5.

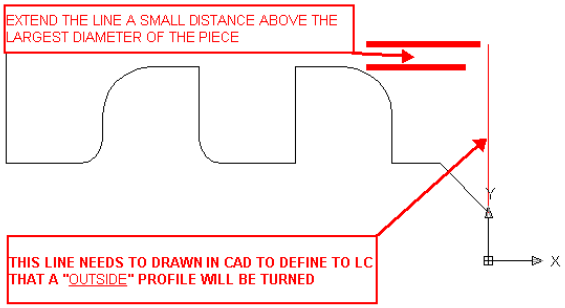
PROFILE DRAWING

No drawing is required for this tutorial although you would need to create one if you wish to generate code for a different profile.different. It is highly suggested that you review Appendix "B" and "C" of LayTurns manual since the general drafting principles as applicable. The figure below shows the drawing fully dimensioned for reference use in this section. The Z dimensions are all positive just for easier reading in the figure.
The profile is drawn accurately. Poorly prepared drawings with, improper origin, lines not connected, lines to arc's not connected, and "paint program" generated DXF's top the list of generating problems in LC Turn.



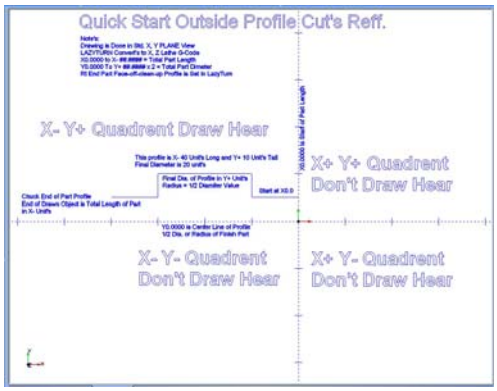
T5- FIGURE 1

There are a few rules in CAD which are very important since on import they define to LC Turn whether the profile portrays an inside or an outside profile. The figure below is for outside machining of a profile. Note you also extend the line above the largest diameter of the profile to define the billet size that will be used.

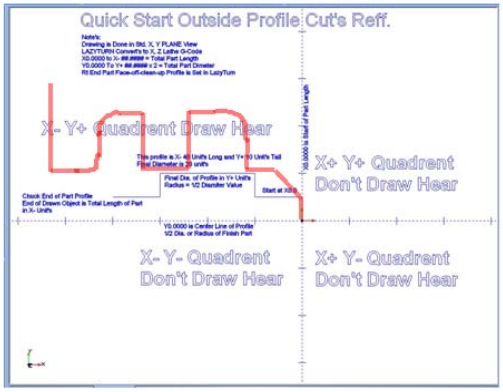


T5- FIGURE 2

Draw your profile in the correct quadrant (information from LT Manual) as shown below. Draw wherever you want, but move the profile when done. I have sketched the profile used in this tutorial in the other figure.

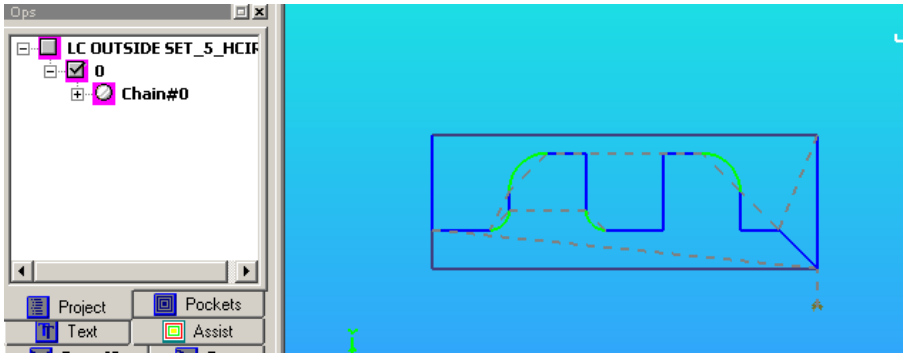


T5- FIGURE 3



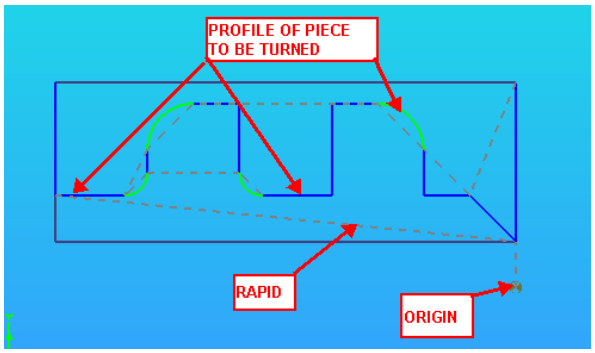
T5- FIGURE 4

The CAD file was saved as a DXF and will look as shown below post importing into LC.
Notice that a chain was created.



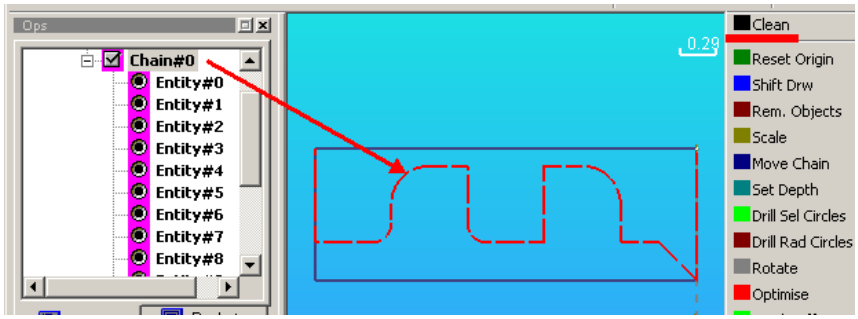
T5- FIGURE 5

The following figure shows what you are looking at in the screen display.



T5- FIGURE 6

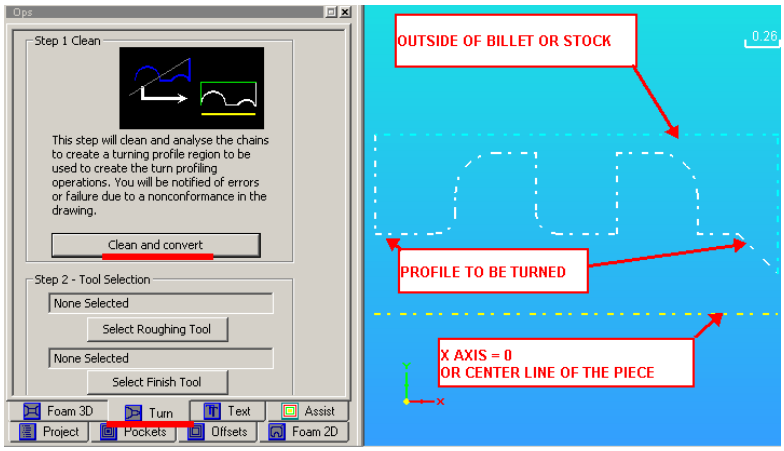
Chain#0 contains all the entities of the profile. Post cleaning and the chain selected is shown below.



T5- FIGURE 7

OUTSIDE PROFILE – GENERATING THE CODE

The next step is to click the Turn Tab which will invoke the Turn menu.
Click the Clean and convert button and the profile is now ready for selecting the tool you wish to use in the machining operation. Should there be a problem with the provided drawing you will get a message. Note how the screen display changed, and what it portrays as shown in the figure below.

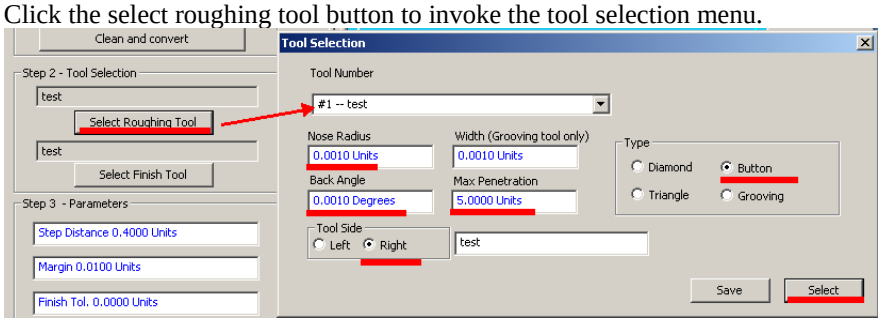


T5- FIGURE 8

You need to define to LC what kind of tool and some tool parameters so it can provide pathing. Most users will have problems with their files because of improper tool use. (ie; couldn't properly mill a ½" wide x 1" deep slot with a 1" wide end mill only and ¼" long cutting edge). You need to apply the proper tool to the different profile sections to be able to do actual machining operations.

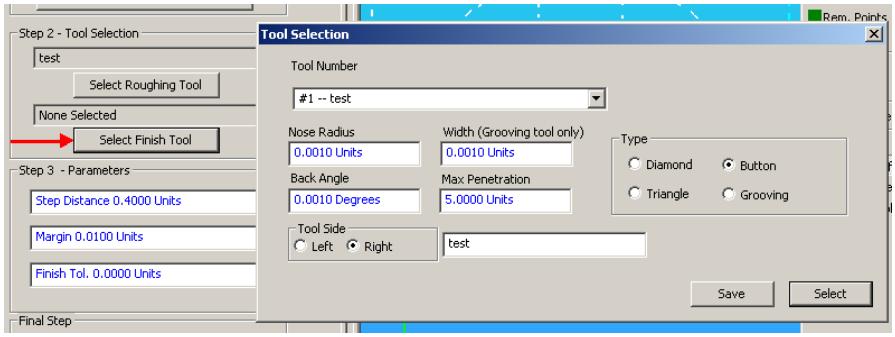
The only concern for this tutorial section is that LC generate the proper Gcode for a finish/ final continuous cut of profile. To do that we will use and create a **make believe lathe tool**. It's a button tool that is only 0.002" wide (nose radius is only 0.001"), is 5 inches long (so it can penetrate / cut any depth on the profile). It is like a long pointed pin! The tool will be used for both roughing and finishing the profile to dimension.

Click the select roughing tool button to invoke the tool selection menu.



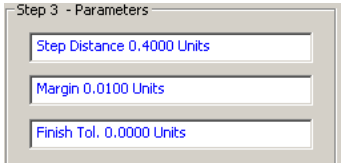
T5- FIGURE 9

A roughing and a finish pass is required to generate a path. The same tool is selected to keep it simple.



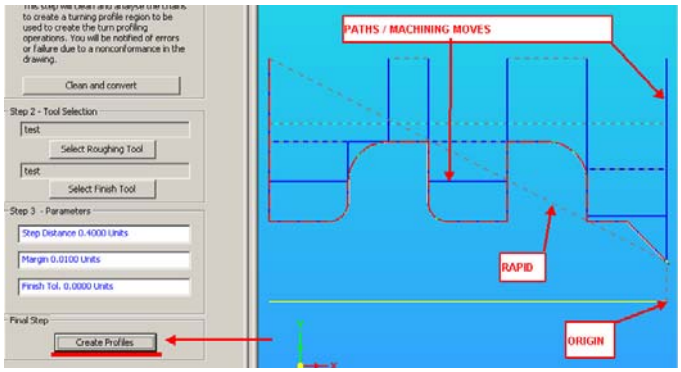
T5- FIGURE 10

LC knows what kind of tool we are going to use, but it also needs to know how deep the tool will cut for each pass it makes (Step Distance), how far to pull out and away from the billet (margin), and also how much finish we want left after the rough or finish cuts are done. The parameters shown below will minimize the amount of gcode generated. (ie; if the step distance was only .001" and we have to remove .6" that would amount to a lot of passes)



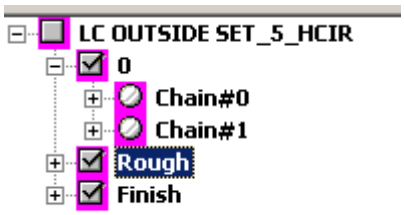
T5- FIGURE 11

Click the Create Profiles button and LC generated the pathing. Note in the figure below that the paths and rapids are now shown.



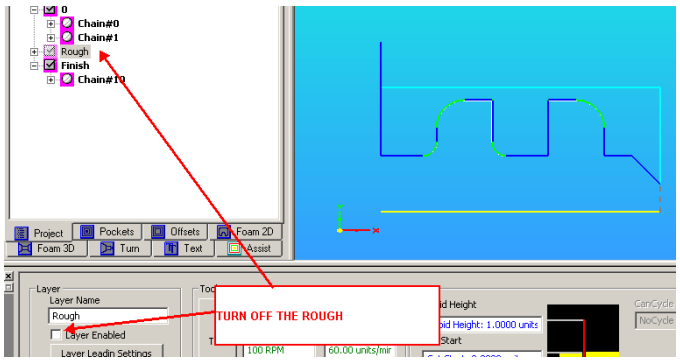
T5- FIGURE 12

Click the project tab and note that a rough and finish path was created based on our defined tool parameters.



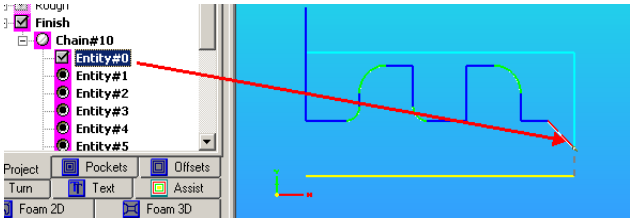
T5- FIGURE 13

The rough passes will not be required when posting the code and turning them off simplifies the display. Once turned off you only see the single continuous finish pass as shown below.

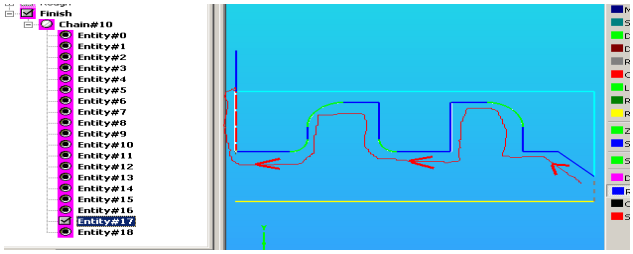


T5- FIGURE 14

Check the entities for proper order of machining. This is done by selecting each entity in the listing, one by one, and making sure the order is correct. If not you can always change the order. In this case we just want one continuous cut from right to left. As shown in figure 15 and 16 below.

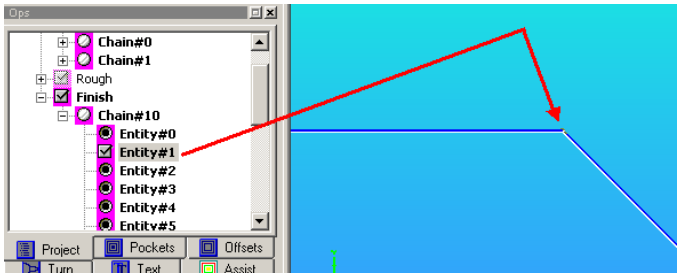


T5- FIGURE 15



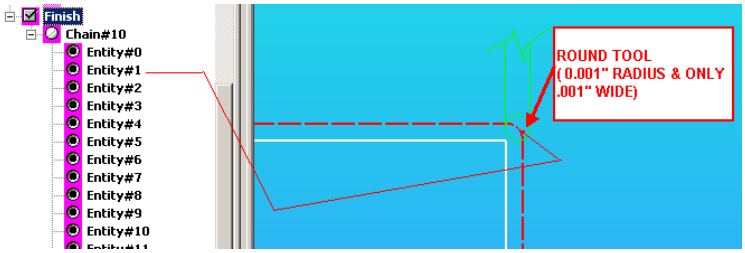
T5- FIGURE 16

Note that in the chain you may notice that nothing highlights as shown below.



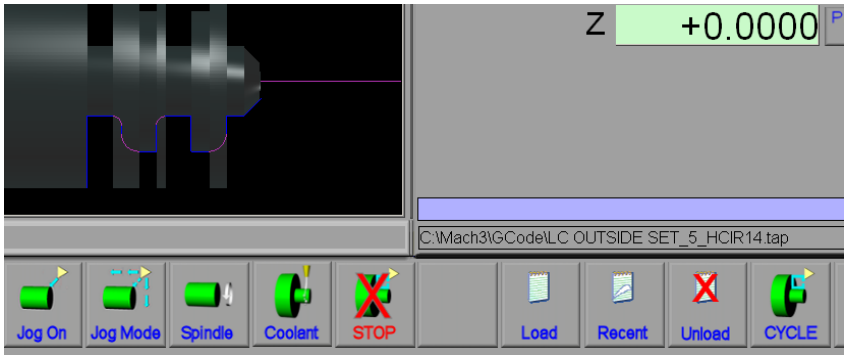
T5- FIGURE 17

LC compensates for how the tool is shaped. The tool has a radius and to cut a sharp corner the tool path must take into account how the tool was defined as portrayed below.



T5- FIGURE 17

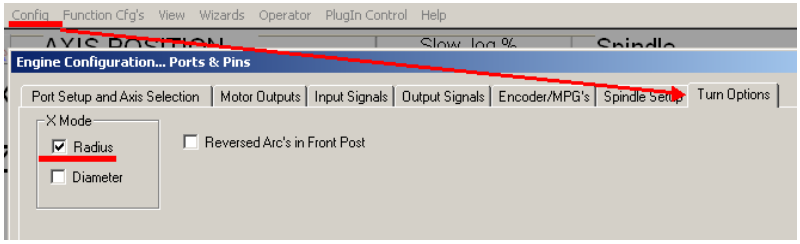
Below is a screen shot of the finish posted gcode in MACH3 Turn.



T5- FIGURE 18

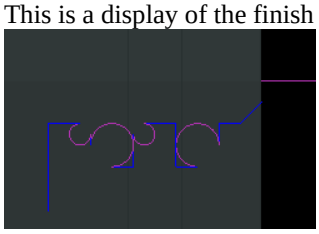
MACH3 TURN CONFIGURATION

The profile was drawn in terms of the profiles radius. Additionally Reversed arc's in front tool post is not checked as shown below. (you may want to take another look at the configuration chart that was shown in Section ???). The Arc Motion Tutorial located in the Members Docs provides detailed information about arc gcode, the affects of different configurations, and even code. It is highly suggested you read it since is lathe specific.



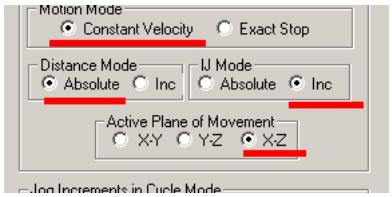
T5- FIGURE 19

This is a display of the finish code if reversed arc's was checked. It will cut as shown!



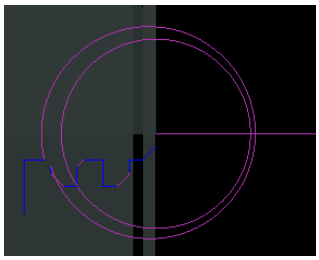
T5- FIGURE 20

Here is additional configuration which was used for this tutorial.



T5- FIGURE 21

This is a display of the in MACH3 TURN if absolute IJ Mode was checked in the configuration. It will cut as shown!



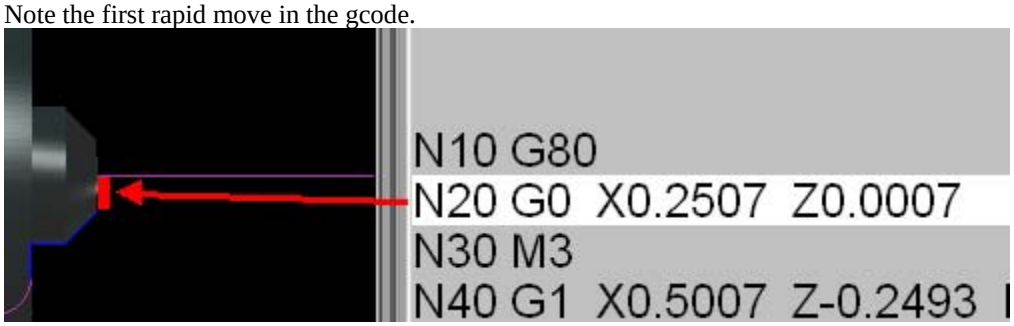
T5- FIGURE 22

His is the actual gcode for the finish pass.

```
N10 G80
N20 G0  X0.2507  Z0.0007
N30 M3
N40 G1  X0.5007  Z-0.2493  F60.00
N50 G3  X0.5010  Z-0.2500  I-0.0007  k-0.0007
N60 G1  Z-0.4990
N70  X0.7500
N80 G3  X1.0010  Z-0.7500  I0.0000  k-0.2510
N90 G1  Z-1.0000
N100 G3  X1.0000  Z-1.0010  I-0.0010  k-0.0000
N110 G1  X0.5010
N120  Z-1.3749
N130 G2  X0.6250  Z-1.4989  I0.1240  k0.0000
N140 G1  X1.0000  Z-1.4990
N150 G3  X1.0010  Z-1.5000  I-0.0000  k-0.0010
N160 G1  Z-1.7500
N170 G3  X0.7500  Z-2.0010  I-0.2510  k-0.0000
N180 G1  X0.6250
N190 G2  X0.5010  Z-2.1250  I0.0000  k-0.1240
N200 G1  Z-2.4990
N210  X1.1137
N220  X1.5237
N230 M30
```

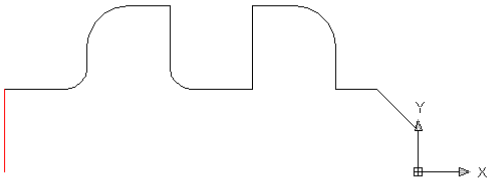
T5- FIGURE 23

How you actually set your tool tip (the controlled point of the cutter) to the stock is important.
Note the first rapid move in the gcode.

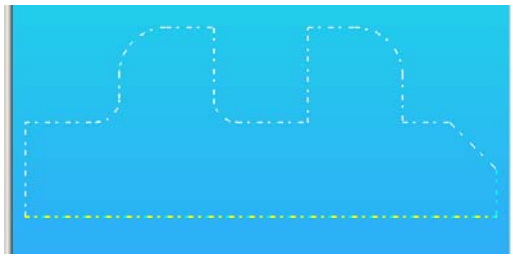


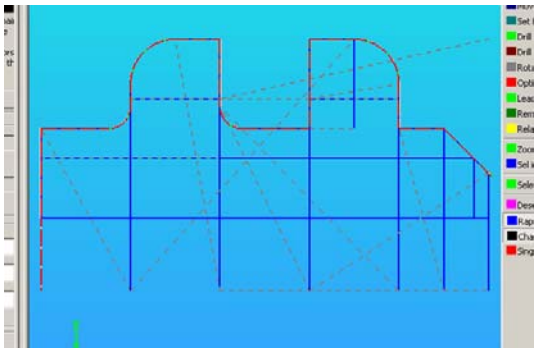
INSIDE PROFILE – GENERATING THE CODE

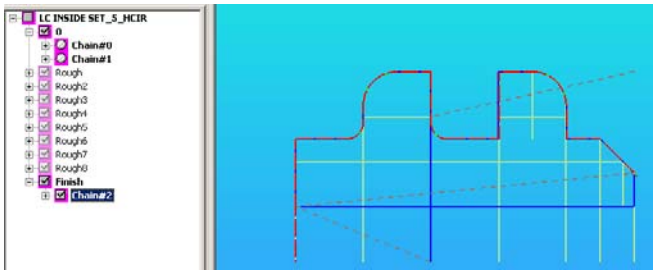
The figure below is for inside machining of a profile. Note you also extend the line above the largest diameter of the profile to define the billet size that will be used.



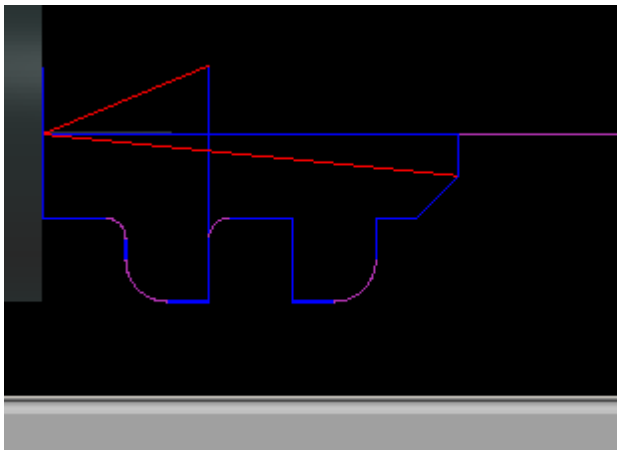
T5- FIGURE 24

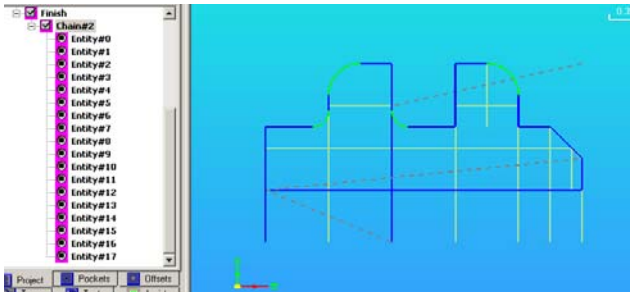


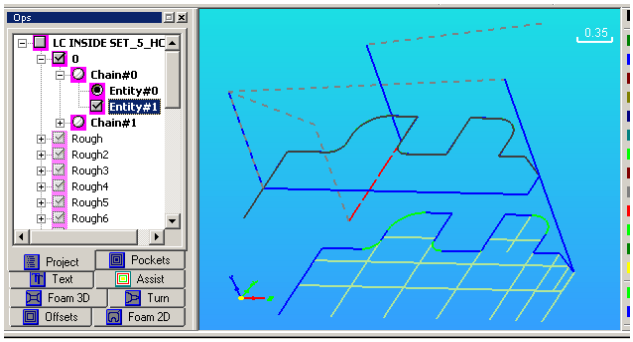


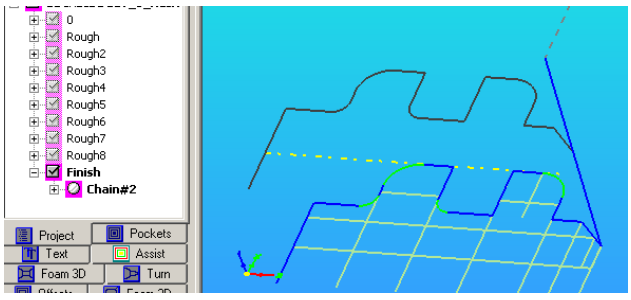


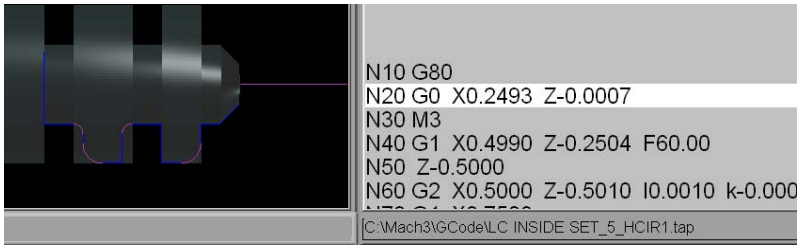
NOT CHECKING THE CUTTINGORDER











LC INSIDE SET_5_HCIR1 - Notepad

File Edit Format View Help

```
N10 G80
N20 G0 X0.2493 Z-0.0007
N30 M3
N40 G1 X0.4990 Z-0.2504 F60.00
N50 Z-0.5000
N60 G2 X0.5000 Z-0.5010 I0.0010 k-0.0000
N70 G1 X0.7500
N80 G3 X0.9990 Z-0.7500 I0.0000 k-0.2490
N90 G1 Z-0.9990
N100 X0.5000
N110 G2 X0.4990 Z-1.0000 I0.0000 k-0.0010
N120 G1 Z-1.3749
N130 G2 X0.6250 Z-1.5009 I0.1260 k-0.0000
N140 G1 X0.9990 Z-1.5010
N150 Z-1.7500
N160 G3 X0.7500 Z-1.9990 I-0.2490 k0.0000
N170 G1 X0.6250
N180 G2 X0.4990 Z-2.1250 I0.0000 k-0.1260
N190 G1 Z-2.4990
N200 X0.0000
N210 X-0.4100
N220 M30
```