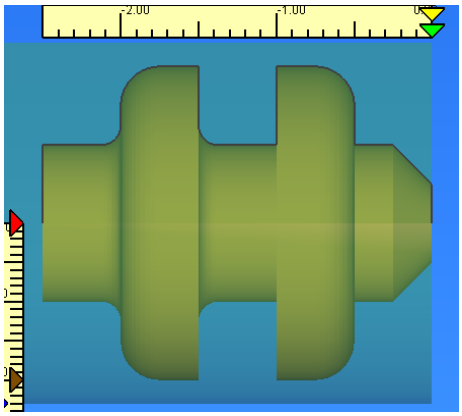
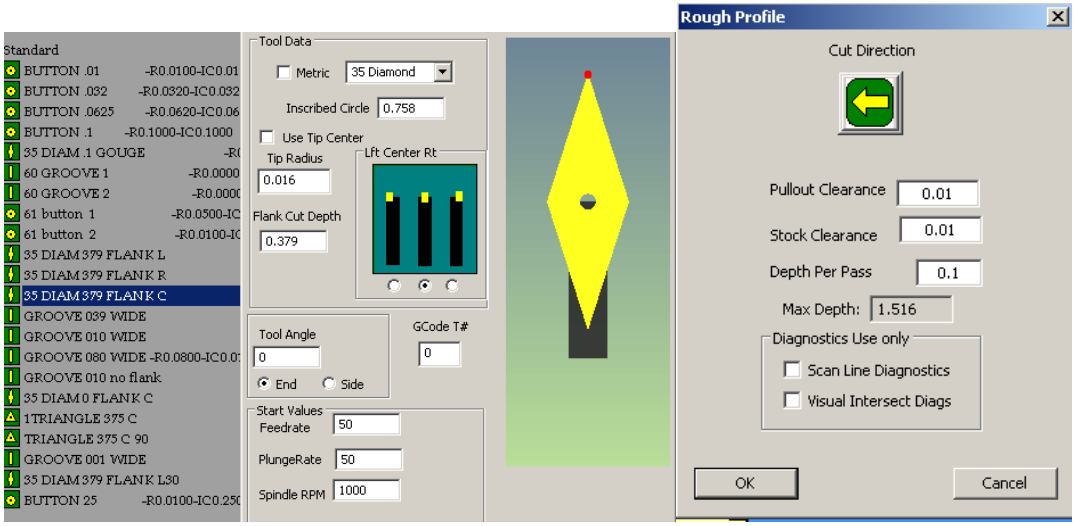
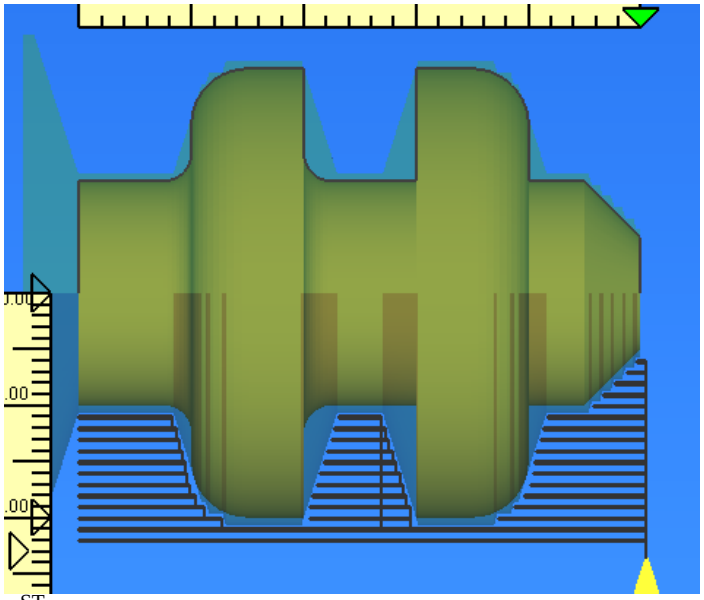


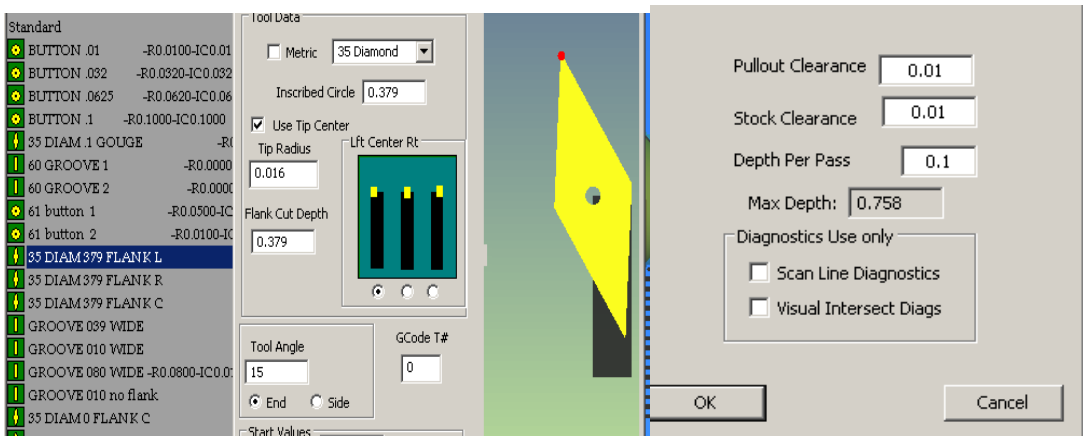
FILE: SECTION 5 HCIR



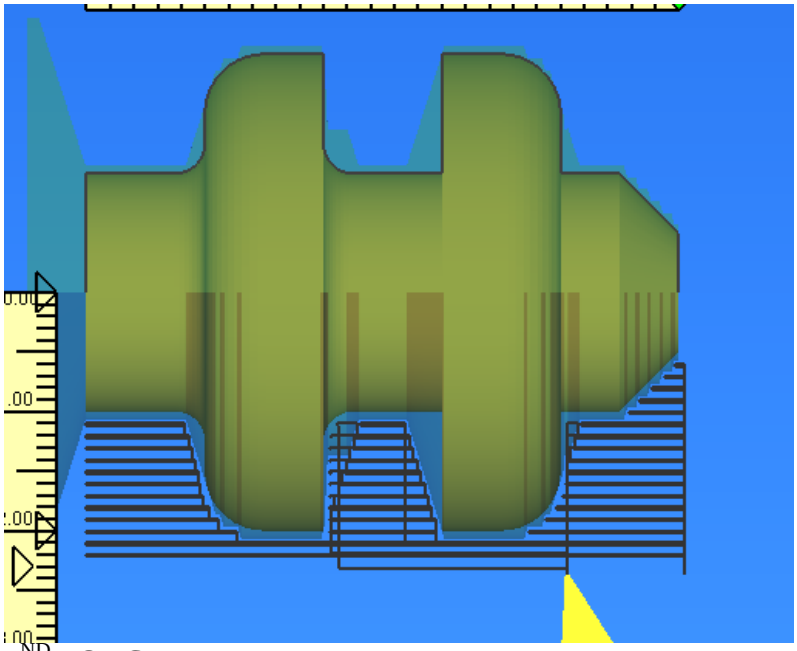
ROUGH TOOL



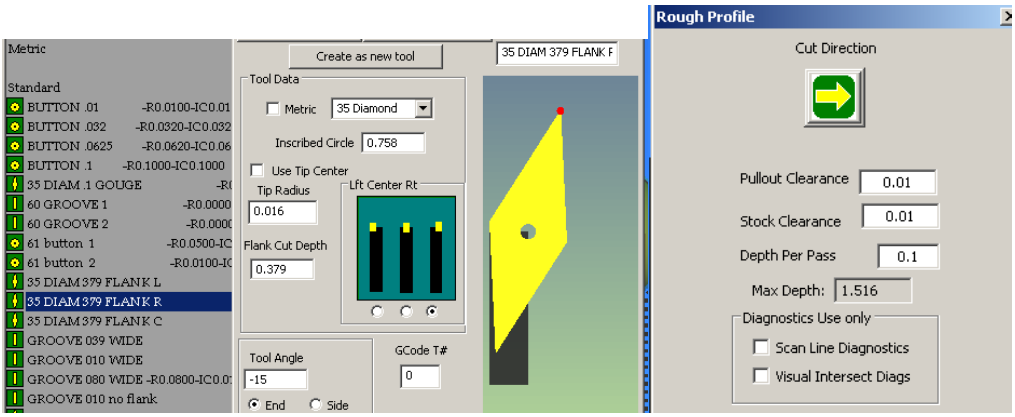
1ST ROUGH



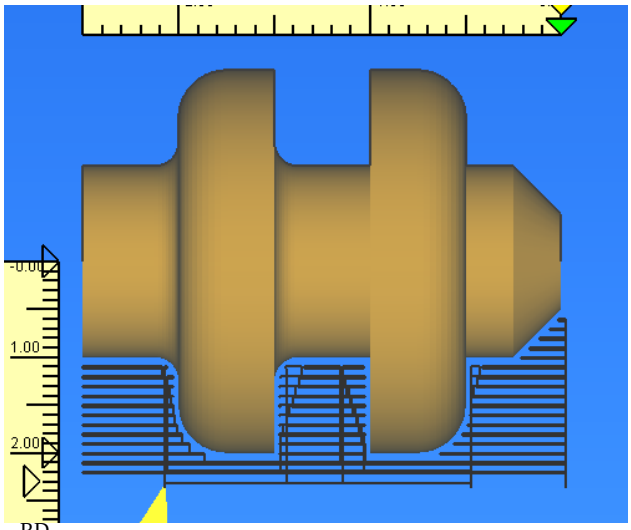
ROUGH TOOL



2ND ROUGH



ROUGH TOOL

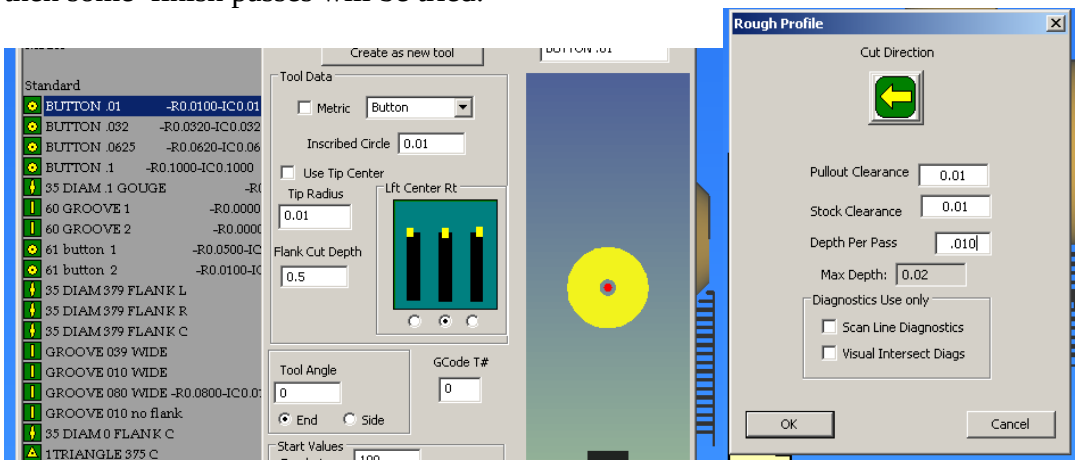


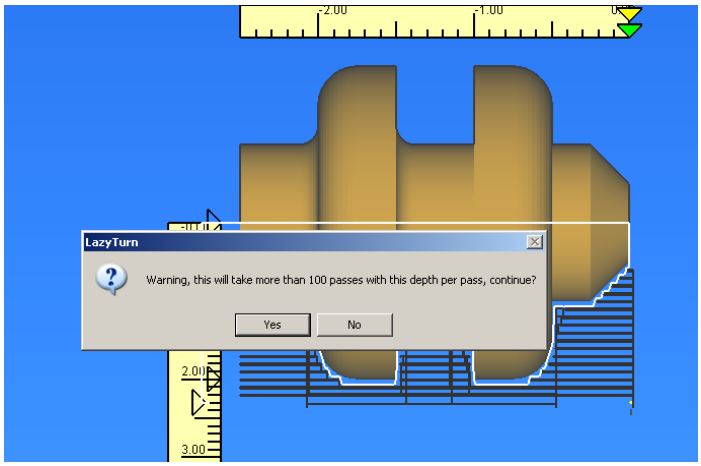
3RD ROUGH

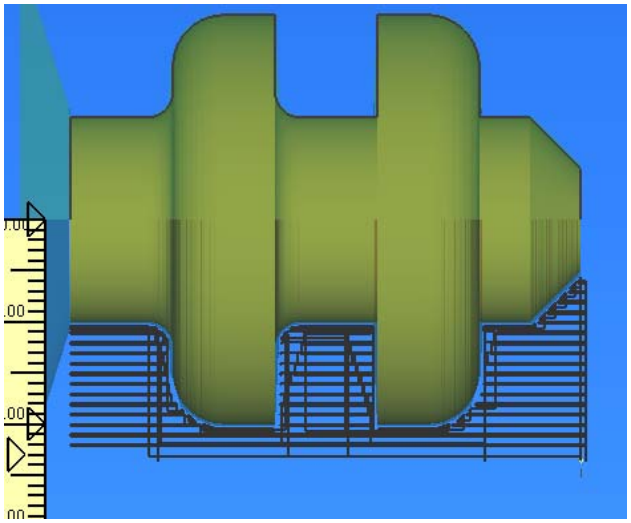
The 3 rough passes have been used to machine down the billet to within 0.010” of the profile. Now it’s time to do some finish passes.

As a test of the finish passes, miscellaneous tools will be tried to machine down to the profile. The only single tool which could be used to do this would be a grooving or point tool (small button or a very narrow grooving tool). Of course you could also use the same rough tools but because of the restriction on tip radius it would never truly cut down to the profile in a sharp corner. Practicaly speaking the design should be such that it includes a suitable corner radius. Should grooving be implemented in the future then you could do some plunge cuts. An option would be to accept the code at this point and split the profile up for individual machining operations.

This is just another rough pass made with a button tool for kicks . It will be deleted and then some finish passes will be tried.



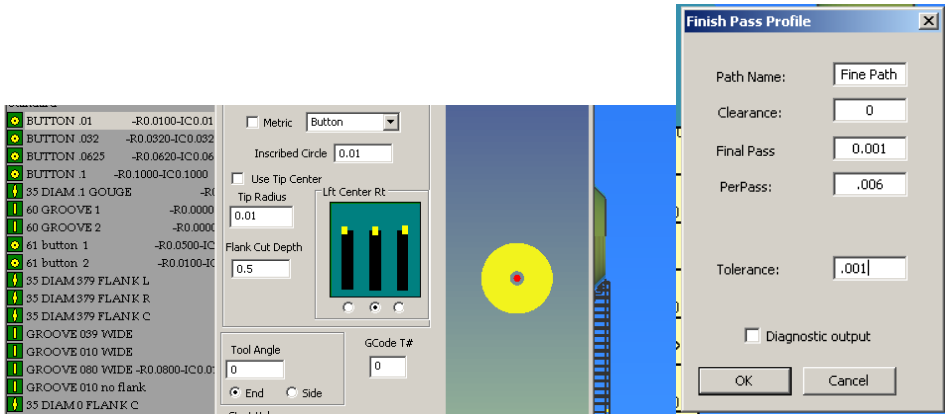


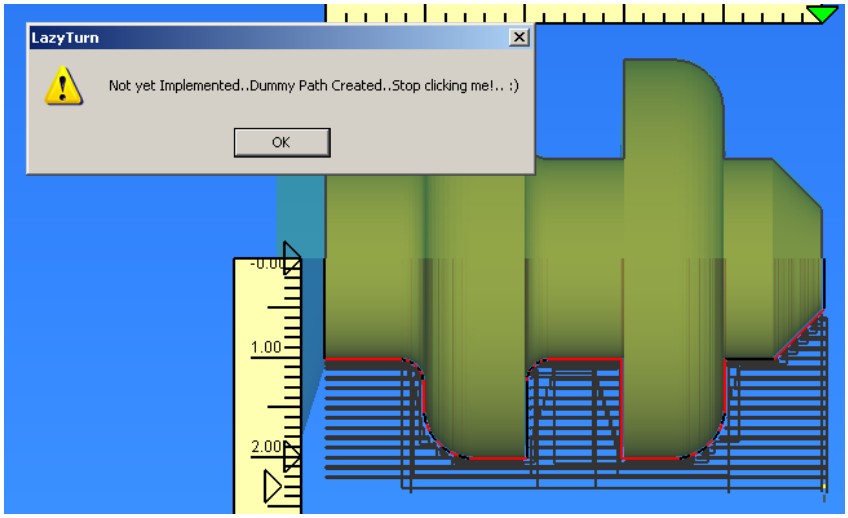


DIFFERENT FINISH PASSES

Make note of the different tools used and the passes. All of these finish passes are being done to show the various results of the finish pass as indicated by red / black line .The tool info is given and also the displayed finish indicator for each tried cut.

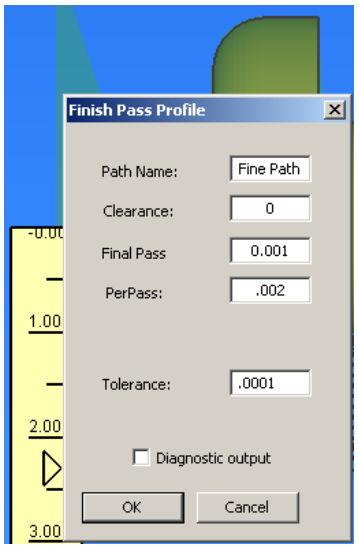
FINISH#1

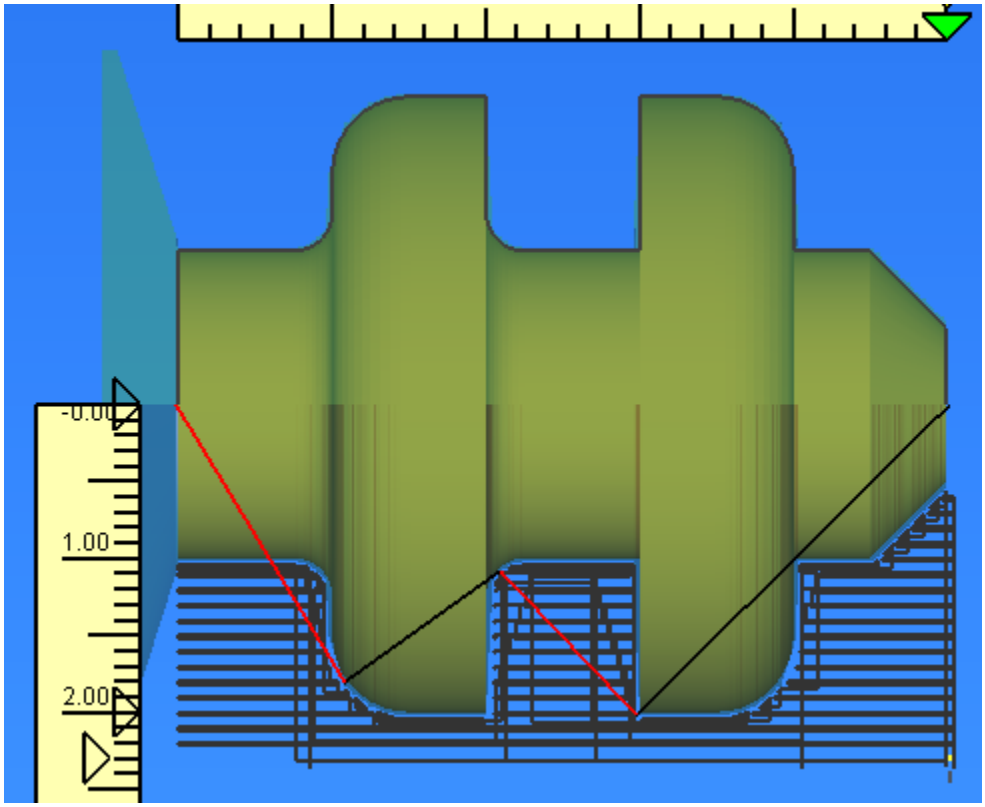




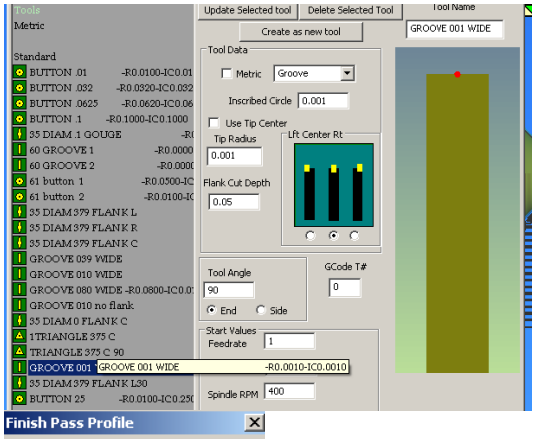
FINISH#2

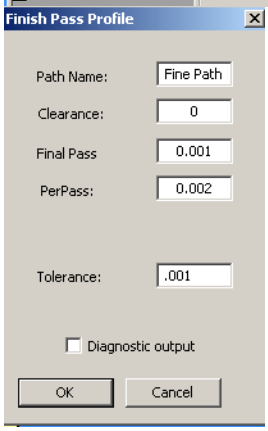
Same tool as #1 but different pass. Obviously there is a problem with this one.

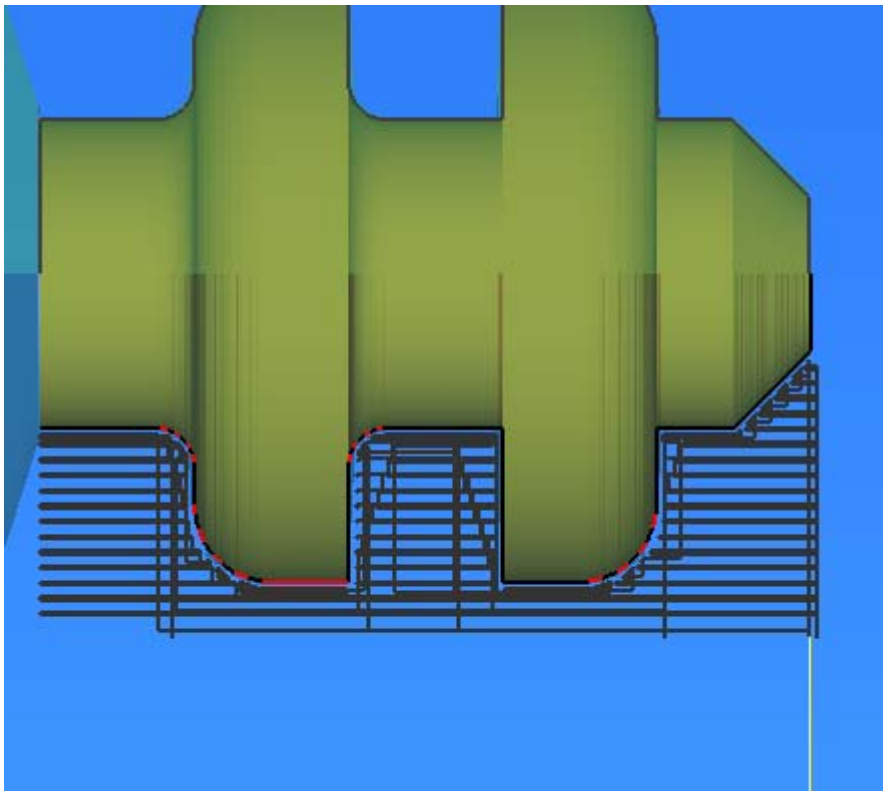




FINISH#3

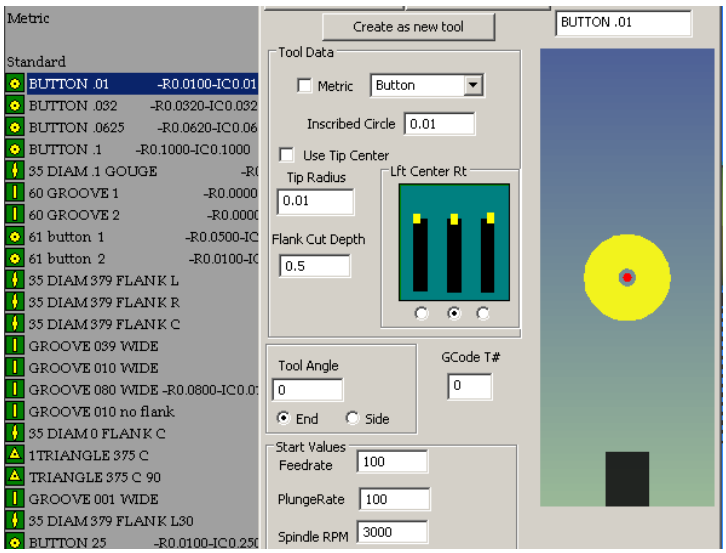

Tools
Metric
Standard
BUTTON .01
BUTTON .032
BUTTON .0625
BUTTON .1
35 DIAM .1 GOUGE
60 GROOVE 1
60 GROOVE 2
61 button 1
61 button 2
35 DIAM 379 FLANK L
35 DIAM 379 FLANK R
35 DIAM 379 FLANK C
GROOVE 039 WIDE
GROOVE 010 WIDE
GROOVE 080 WIDE
GROOVE 010 no flank
35 DIAM 0 FLANK C
1TRIANGLE 375 C
TRIANGLE 375 C 90
GROOVE 001
GROOVE 001 WIDE
35 DIAM 379 FLANK L30
BUTTON 25
Update Selected tool
Delete Selected Tool
Create as new tool
Tool Name
GROOVE 001 WIDE
Tool Data
Metric
Groove
Inscribed Circle
0.001
Use Tip Center
Tip Radius
0.001
Flank Cut Depth
0.05
Lft Center Rt
Tool Angle
90
End
Side
GCode T#
0
Start Values
Feedrate
1
Spindle RPM
400

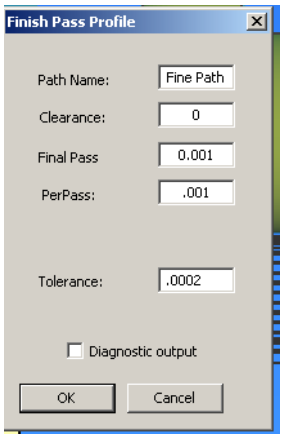

Finish Pass Profile
Path Name:
Fine Path
Clearance:
0
Final Pass
0.001
PerPass:
0.002
Tolerance:
.001
Diagnostic output
OK
Cancel

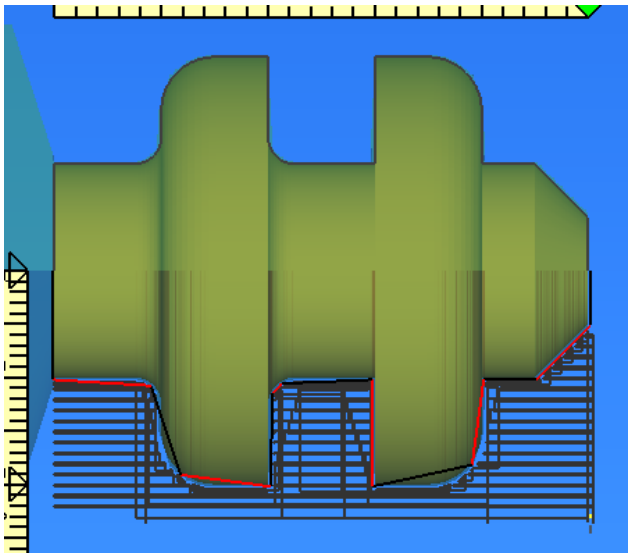


FINISH#4

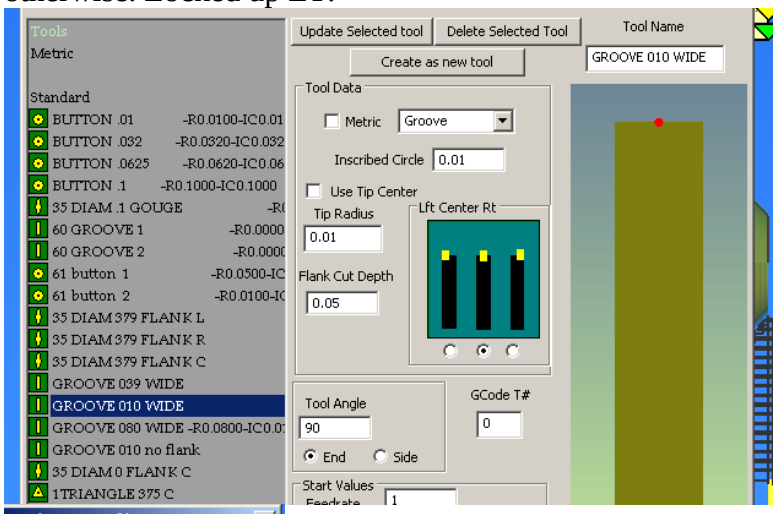
Problem with this one.

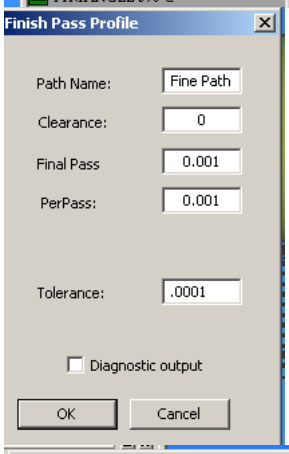


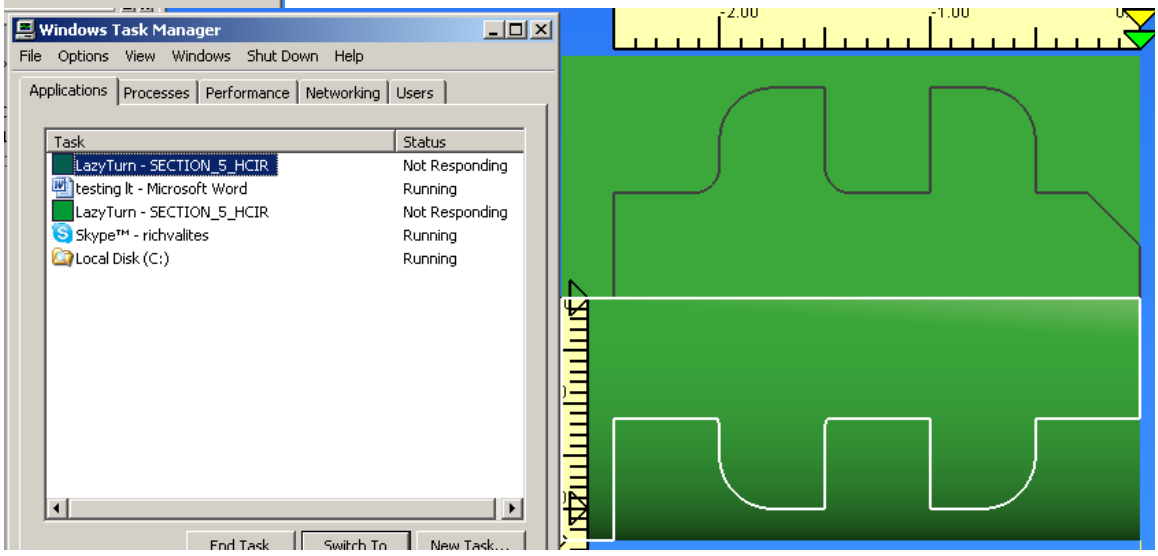




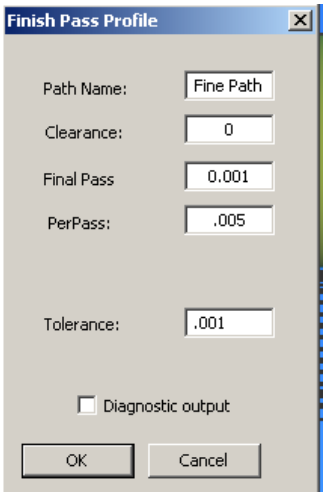
FINISH#5
Just let LT try and do a finish without roughs and it crapped out…almost made it though! Normaly you would of course use the diagnostics as the computing time is very long otherwise. Locked up LT.

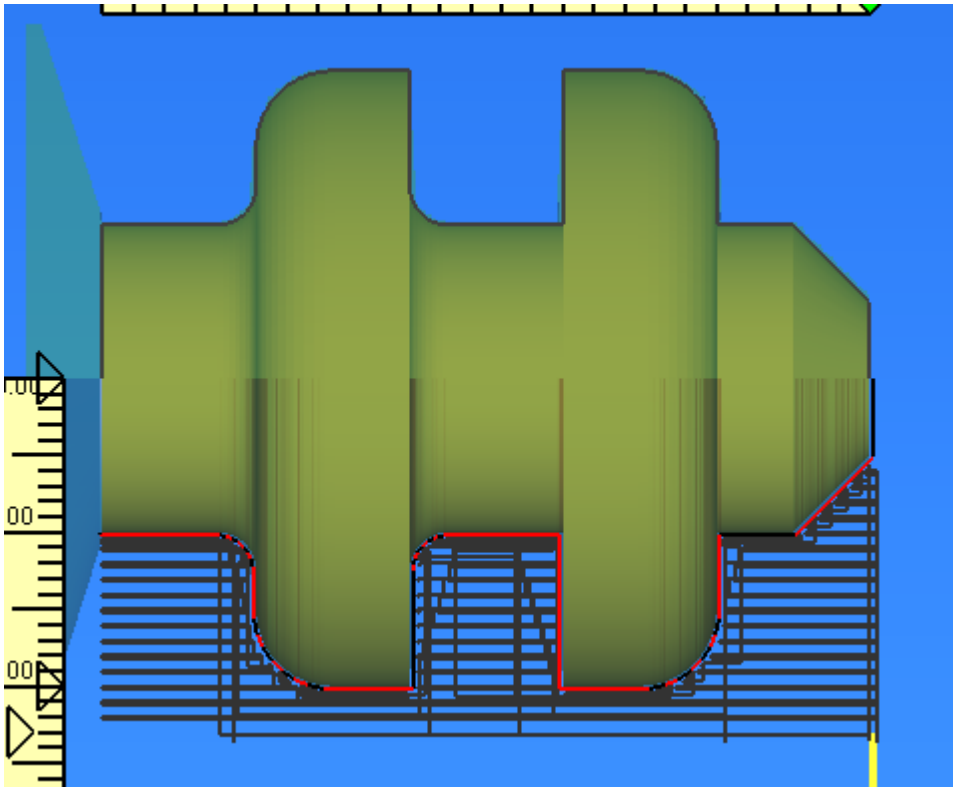




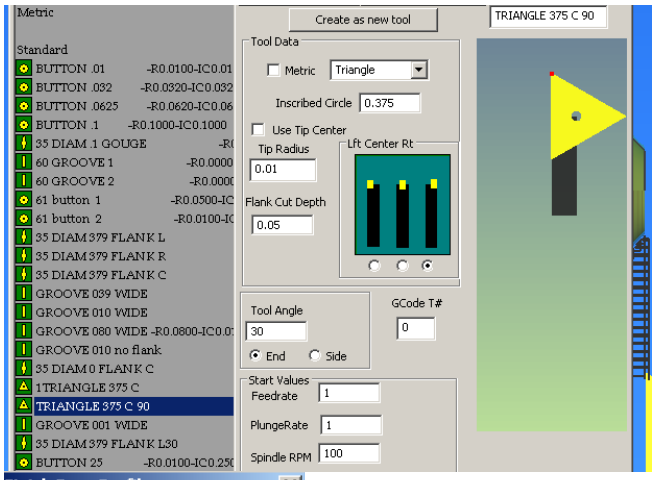


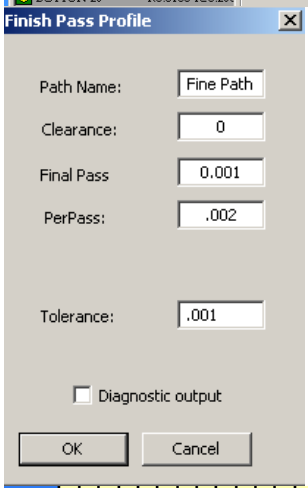
FINISH#6

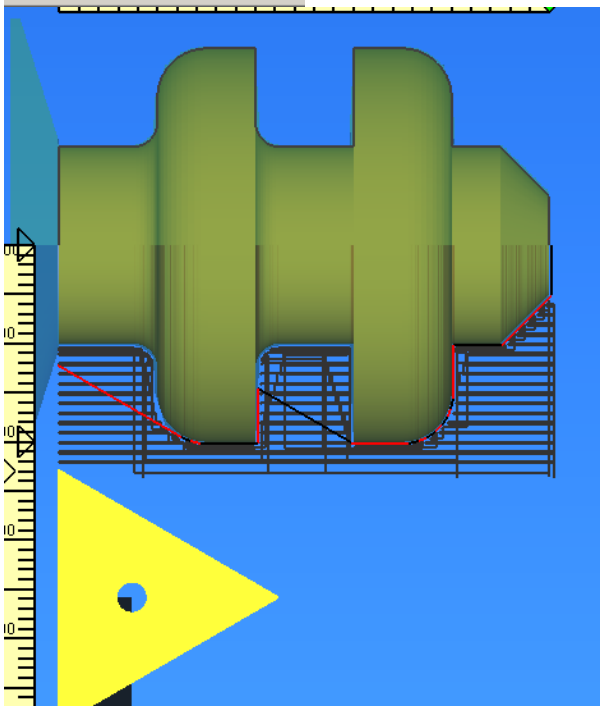

Finish Pass Profile
Path Name:
Fine Path
Clearance:
0
Final Pass
0.001
PerPass:
.005
Tolerance:
.001
Diagnostic output
OK
Cancel



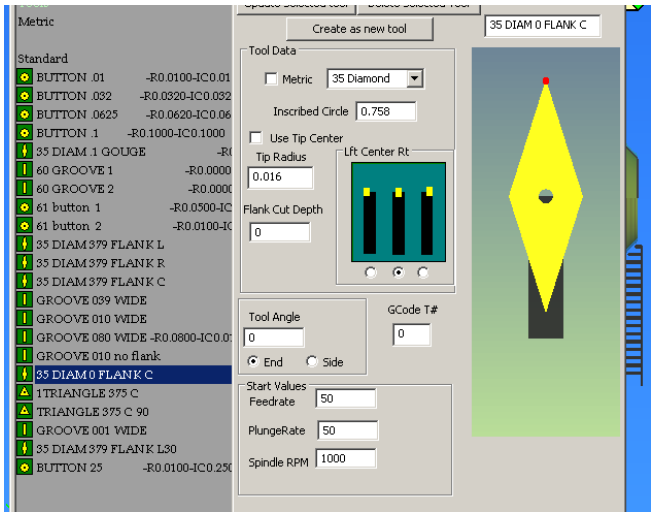
FINISH#7

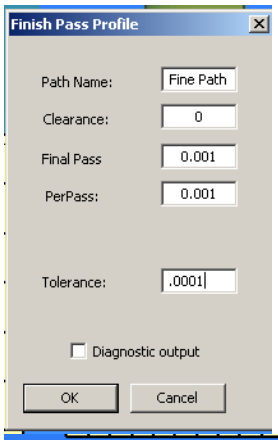

Metric
Standard
BUTTON .01 -R0.0100-IC0.01
BUTTON .032 -R0.0320-IC0.032
BUTTON .0625 -R0.0620-IC0.06
BUTTON .1 -R0.1000-IC0.1000
35 DIAM .1 GOUGE
60 GROOVE 1
60 GROOVE 2
61 button 1
61 button 2
35 DIAM 379 FLANK L
35 DIAM 379 FLANK R
35 DIAM 379 FLANK C
GROOVE 039 WIDE
GROOVE 010 WIDE
GROOVE 080 WIDE
GROOVE 010 no flank
35 DIAM 0 FLANK C
1TRIANGLE 375 C
TRIANGLE 375 C 90
GROOVE 001 WIDE
35 DIAM 379 FLANK L30
BUTTON 25
Create as new tool
TRIANGLE 375 C 90
Tool Data
Metric
Triangle
Inscribed Circle
0.375
Use Tip Center
Tip Radius
0.01
Flank Cut Depth
0.05
Lft Center Rt
Tool Angle
30
End
Side
GCode T#
0
Start Values
Feedrate
1
PlungeRate
1
Spindle RPM
100

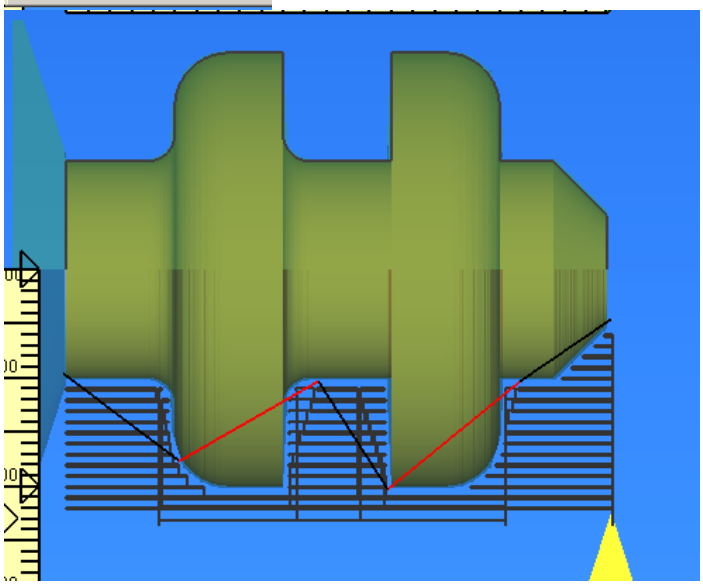

Finish Pass Profile
Path Name:
Fine Path
Clearance:
0
Final Pass
0.001
PerPass:
.002
Tolerance:
.001
Diagnostic output
OK
Cancel



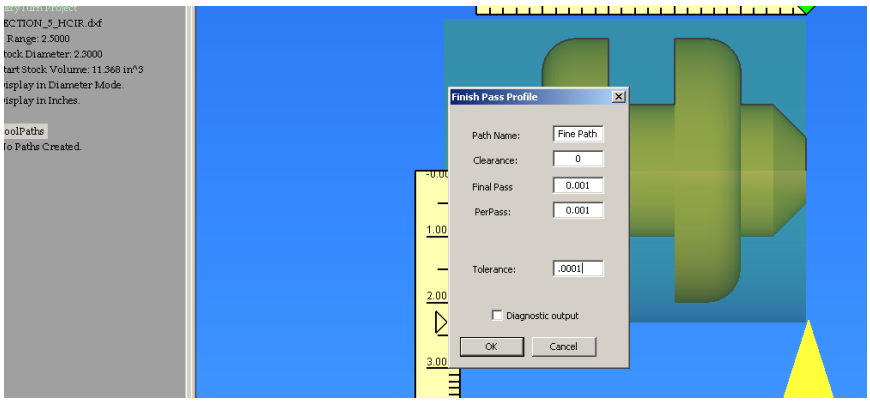
FINISH#7

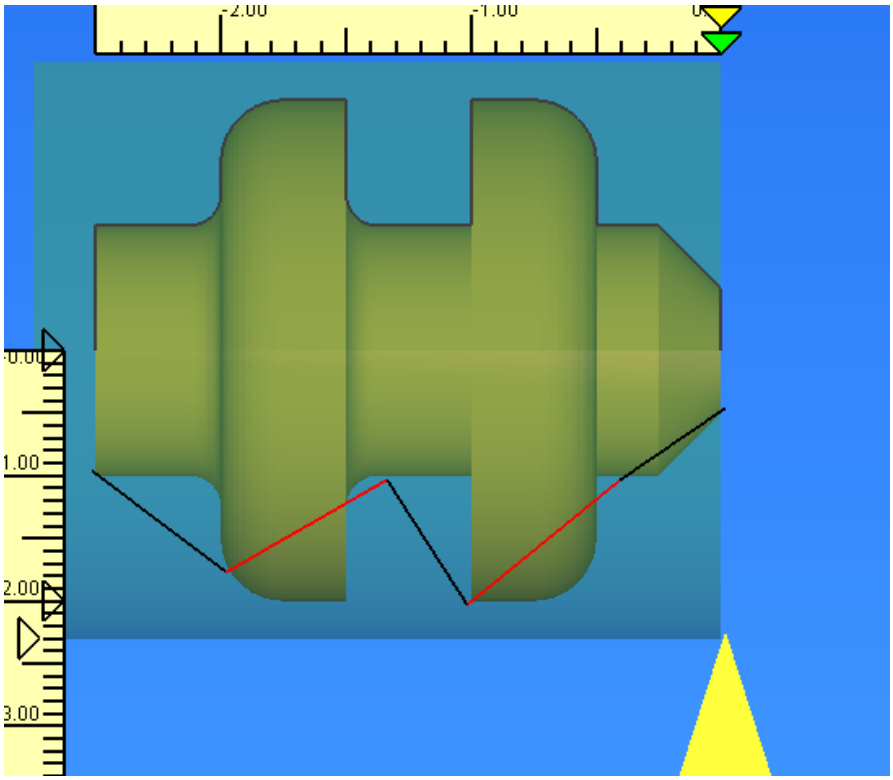

Metric
Standard
BUTTON .01 -R0.0100-IC0.01
BUTTON .032 -R0.0320-IC0.032
BUTTON .0625 -R0.0620-IC0.06
BUTTON .1 -R0.1000-IC0.1000
35 DIAM 1 GOUGE
60 GROOVE 1
60 GROOVE 2
61 button 1
61 button 2
35 DIAM 379 FLANK L
35 DIAM 379 FLANK R
35 DIAM 379 FLANK C
GROOVE 039 WIDE
GROOVE 010 WIDE
GROOVE 080 WIDE -R0.0800-IC0.0
GROOVE 010 no flank
35 DIAM 0 FLANK C
1TRIANGLE 375 C
TRIANGLE 375 C 90
GROOVE 001 WIDE
35 DIAM 379 FLANK L30
BUTTON 25 -R0.0100-IC0.250
Create as new tool
35 DIAM 0 FLANK C
Tool Data
Metric
35 Diamond
Inscribed Circle
0.758
Use Tip Center
Tip Radius
0.016
Lft Center Rt
Flank Cut Depth
0
Tool Angle
0
End
Side
GCode T#
0
Start Values
Feedrate
50
PlungeRate
50
Spindle RPM
1000


Finish Pass Profile
Path Name:
Fine Path
Clearance:
0
Final Pass
0.001
PerPass:
0.001
Tolerance:
.0001
Diagnostic output
OK
Cancel



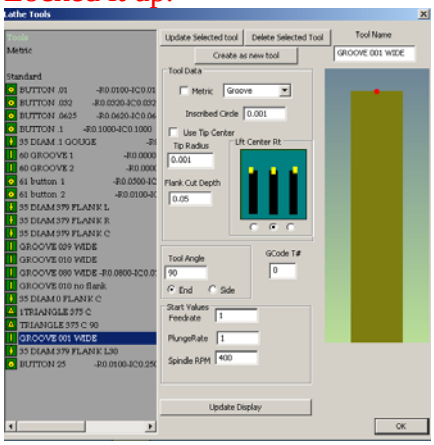
FINISH#8

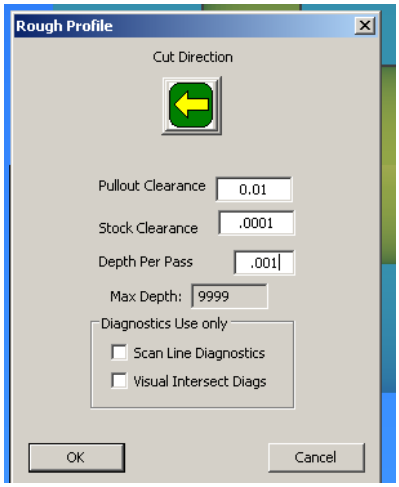

SECTION_5_HCIR.dxf
Range: 2.5000
Stock Diameter: 2.3000
Start Stock Volume: 11.368 in^3
Display in Diameter Mode.
Display in Inches.
ToolPaths
No Paths Created.
Finish Pass Profile
Path Name:
Fine Path
Clearance:
0
Final Pass
0.001
PerPass:
0.001
Tolerance:
.0001
Diagnostic output
OK
Cancel
1.00
2.00
3.00

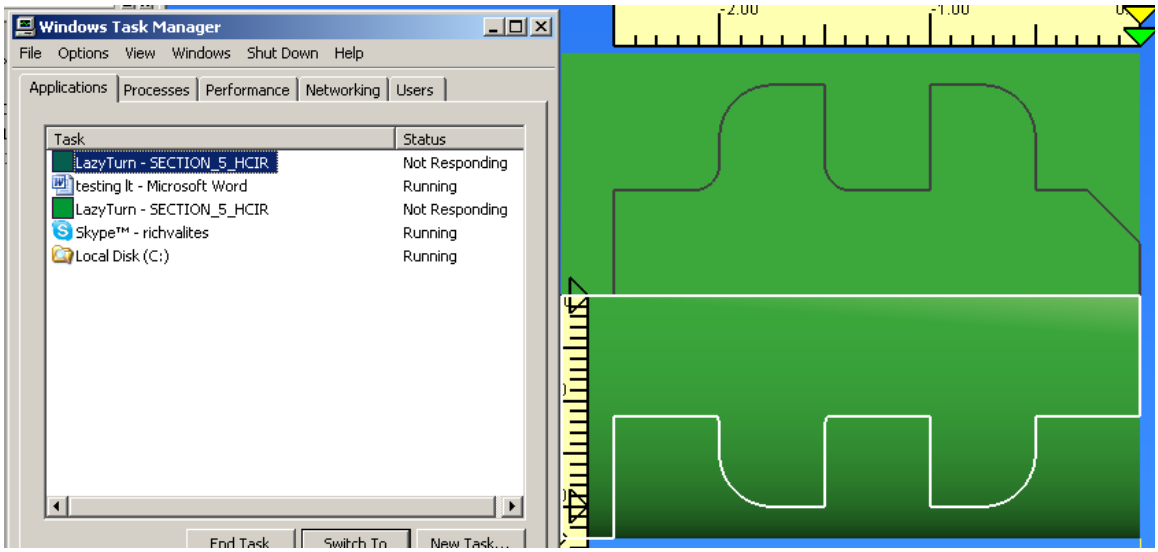

-2.00
-1.00
1.00
2.00
3.00

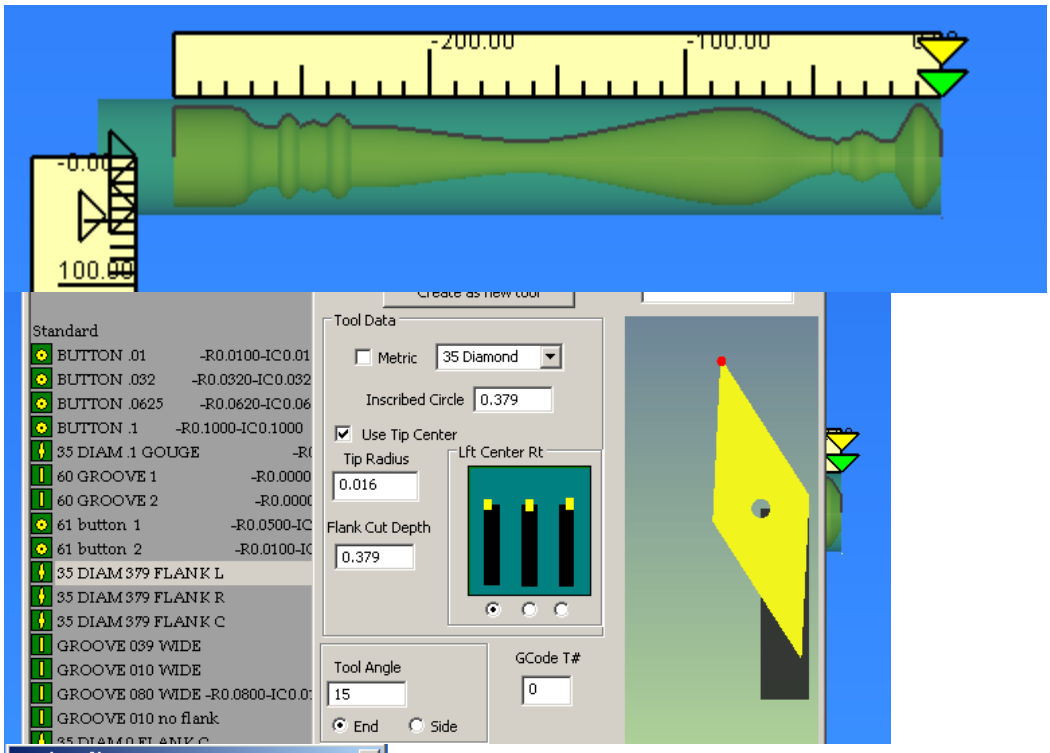
FINISH#9

Locked it up.

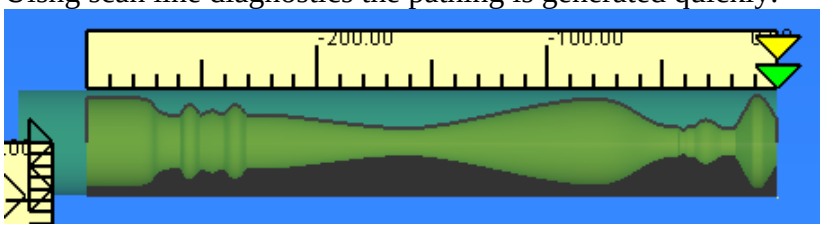




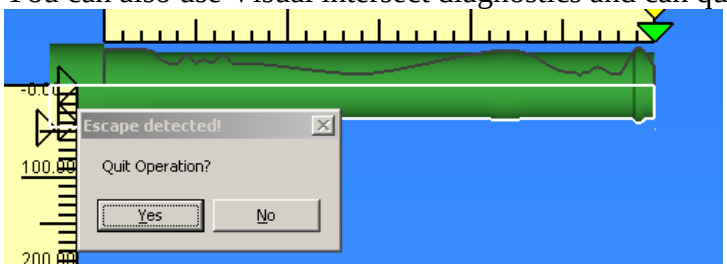




Uisng scan line diagnostics the pathing is generated quickly.

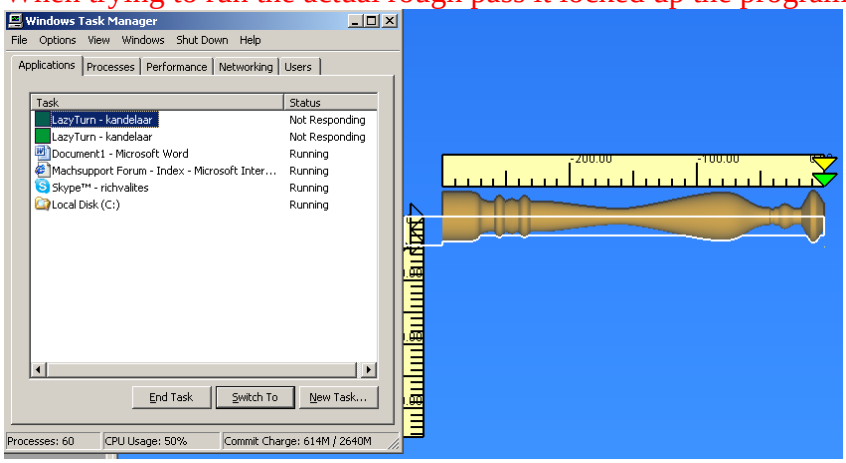


You can also use Visual intersect diagnostics and can quit at any time using the Esc key.

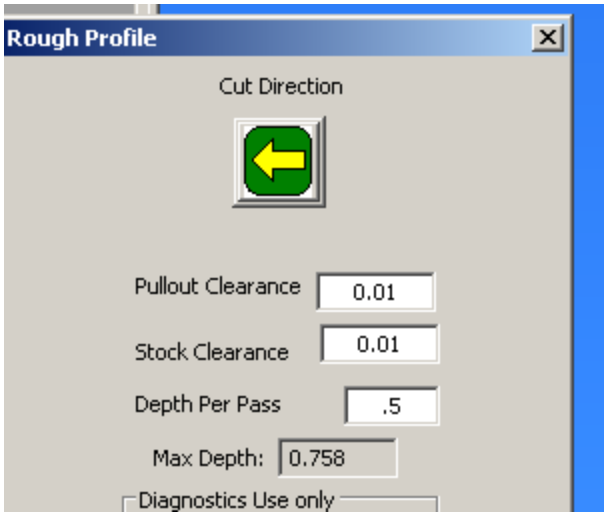


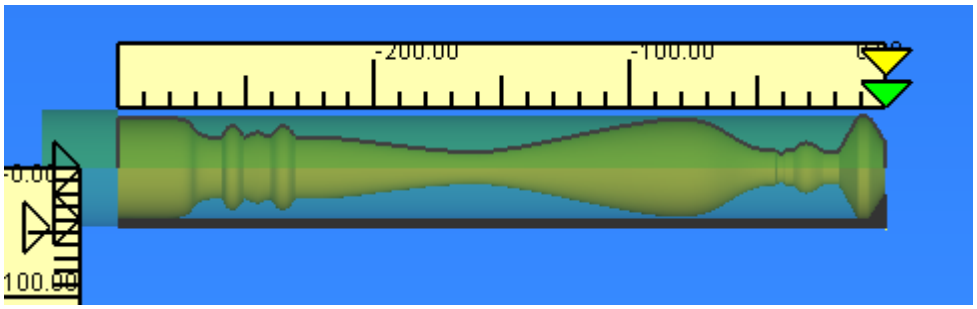
Note that you can still post code but it is not a complete code of all required moves.

When trying to run the actual rough pass it locked up the program.

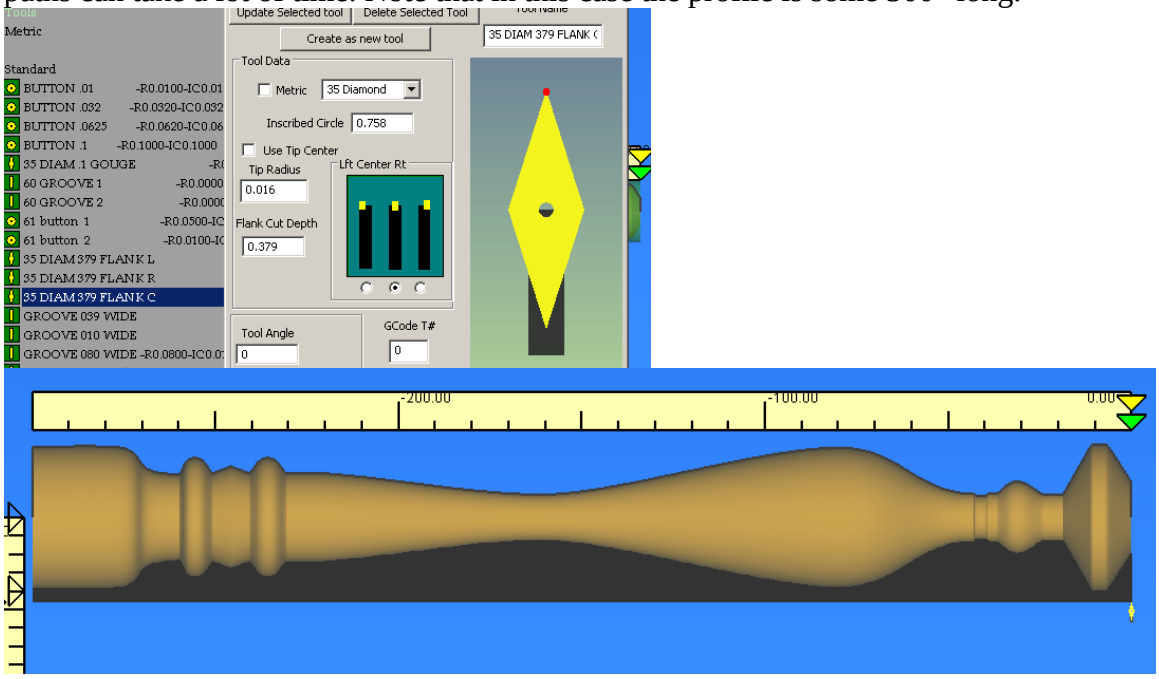


Using a deeper depth per pass. Remember that the piece is 4 foot in diameter!

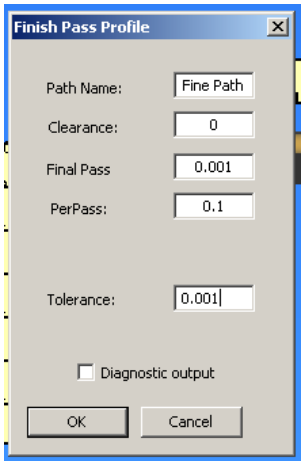




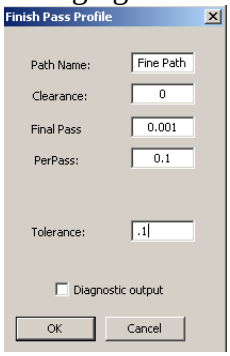
Using a rotated tool requires a whole lot more “computing” thus the time to analyze the paths can take a lot of time. Note that in this case the profile is some 300” long.



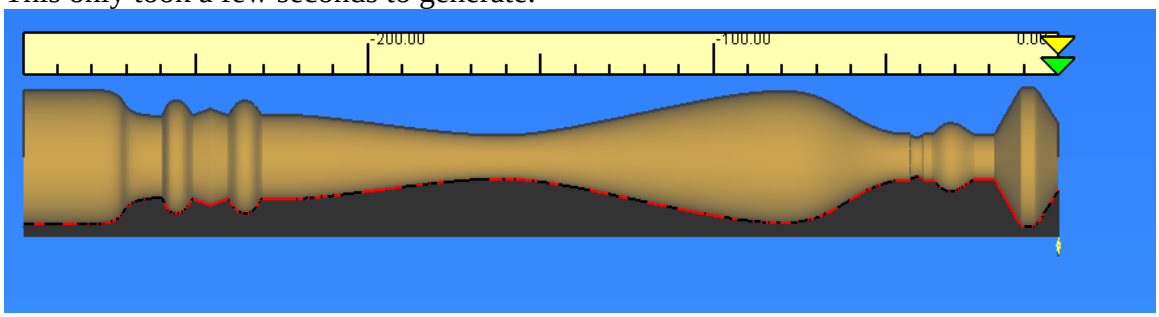
A finish pass using the following locked the program up. Read on!

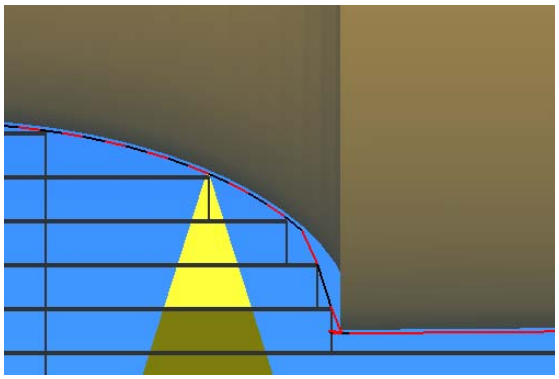


Changing the tolerance as follows:



This only took a few seconds to generate.





This piece is some 300” long (12 feet!). So if you have a tolerance set at say .001”, LT inspects the profile 300 / .001= 300000 as compared to say 300 / .1= 3000.
Use judgment on tolerance. You can look at what it means verse the profile by doing both and measuring / looking in the graphics.