

## TABLE OF CONTENTS

<b>1.0 PREFACE</b>	<b>3</b>
<b>1.1 GENERAL MANUAL COMMENTS</b>	<b>3</b>
<b>2.0 HOW TO USE THIS MANUAL</b>	<b>4</b>
<b>3.0 LAZYTURM PROGRAM DEVELOPMENT OVERVIEW</b>	<b>5</b>
<b>4.0 INSTALLING LAZYTURM</b>	<b>6</b>
<b>5.0 SCREENS – MENUS - TOOLBAR BUTTONS – STATUS BAR</b>	<b>8</b>
<b>5.1 MAIN SCREEN</b>	<b>8</b>
<b>5.2 FILE MENU</b>	<b>9</b>
<b>5.3 OPTIONS MENU</b>	<b>9</b>
<b>5.4 EDIT MENU</b>	<b>10</b>
<b>5.5 VIEW MENU</b>	<b>10</b>
<b>5.6 DEBUG MENU</b>	<b>12</b>
<b>5.7 HELP MENU</b>	<b>12</b>
<b>5.8 TOOLPATH MENU</b>	<b>12</b>
<b>5.9 FILE FOLDER BUTTON</b>	<b>13</b>
<b>5.10 ZOOM &amp; FIT BUTTONS</b>	<b>13</b>
<b>5.11 SELECT TOOL BUTTON</b>	<b>13</b>
<b>5.12 LATHE TOOLS DIALOG</b>	<b>14</b>
<b>5.13 DESCRIPTION OF INPUT BOXES FOR LATHE TOOLS</b>	<b>15 to 19</b>
<b>5.14 CUT BUTTONS</b>	<b>20</b>
<b>5.14.1 ROUGH CUT BUTTON</b>	<b>20</b>
<b>5.14.2 FINISH CUT BUTTON</b>	<b>22 to 23</b>
<b>5.14.3 DIAGNOSTICS</b>	<b>24</b>
<b>5.14.4 SCAN LINE DIAGNOSTICS</b>	<b>24</b>
<b>5.14.5 VISUAL INTERSECT DIAGNOSTICS</b>	<b>25</b>
<b>5.15 ERASE TOOLPATH BUTTON</b>	<b>26</b>
<b>5.16 POST GCODECODE BUTTON</b>	<b>27 to 28</b>
<b>6.0 GRAPHICS AREA</b>	<b>29</b>
<b>6.1 PROFILE and RULERS</b>	<b>30</b>
<b>6.2 RULERS</b>	<b>30</b>
<b>6.2.1 RULER PENTAGONS</b>	<b>30</b>
<b>6.3 PENTAGON VALUE INPUT</b>	<b>31</b>
<b>6.4 CROSS HAIR</b>	<b>32</b>
<b>6.5 MANIPULATION OF THE GRAHIC DISPLAY</b>	<b>32</b>
<b>6.6 BUTTONS</b>	<b>33</b>
<b>6.7 PATH PORTRAYAL</b>	<b>34 to 35</b>
<b>7.0 TOOLPATH GENERATION</b>	<b>36</b>
<b>7.1 TOOLPATHS GENERATION OVERVIEW</b>	<b>36</b>
<b>7.2 DXF DRAWING</b>	<b>37</b>
<b>7.3 TOOL CREATION</b>	<b>38 to 40</b>
<b>7.4 MULTIPLE PATHS AND TOOLS</b>	<b>41 to 47</b>
<b>7.5 DEBUG TOOL</b>	<b>48</b>
<b>7.6 SIMULATION OF THE TOOLPATHS</b>	<b>49</b>

**APPENDIX LISTING**

<b>LTR</b>	<b>APPENDIX TITLE</b>	<b>REV</b>	<b>REMARKS</b>
A	LAZYTURNTURN QUICK AND DIRTY	5	
B	CAD DRAWINGS AND COMMON MISTAKES	4	
C	DXF FILE EXPORTS	4	
D	POSTED DXF FILES	2	
E	PRACTICAL MACHINING CONSIDERATIONS	5	
F	MACH 3 TURN - QUICK REFERENCE	0	
G	PROFILE CODE FROM LAZYCAM	0	

## 1.0 PREFACE

### 1.1 GENERAL MANUAL COMMENTS

**The information contained in this manual is based on the following information:**

**PROGRAM FILE: LazyTurnTest1**

**SITE TOPIC: LAZYTURMAN – posted replies as of this date : March 14, 2010**

- 1.2 This manual pertains only to the LazyTurn lathe program. Other manuals are specifically written for Mach and LazyCam and the user is advised to consult those manuals if required. Only very specific information relating to those programs are included in this manual as necessary for LazyTurn.
- 1.3 The manual is arranged differently such that it follows how you would actually use the program and is supplemented by specific appendixes.
- 1.3 LazyTurn development is dynamic and what is currently written can change quickly. Thus you are encouraged to follow the LazyTurn topic.
- 1.5 Only the latest manual along with a designated REV # is posted in Members DOCS. The manual will be updated without notice and without revision history. Here is the link:  
<http://www.machsupport.com/forum/index.php/topic,12078.0.html>
- 1.6 The verbage contained has not been checked and verified and by all means feel free to post any errors or comments you may have in the LazyTurn Manual site thread. Here is the link:  
<http://www.machsupport.com/forum/index.php/topic,10061.0.html>
- 1.7 Thanks to all that have contributed in any way. Without replies and testing this manual would not be available today. Only one name is noted in the manual, ART, and to him the community of turners should be thankful.
- 1.8 Verbage in [blue](#) is not currently implemented in LazyTurn.

## 2.0 HOW TO USE THIS MANUAL

The manual has a natural progression of presenting material as if you never knew what LazyTurn was. The blue text highlights what is not implemented in the program, Italic text is commentary on a particular subject, numerous figures are used along with red text / arrows / underlines to assist in relating the figure to what is written.

The manual is progressive and there very little duplication. Here is the progression:

- It provides information on the programs concept
- Tells you how to install the software
- Talks about each and every screen, menu, pull down so you know what your see.  
The information presented stays focused on the material being presented and will refer to other sections or appendixes to maintain that focus.
- The manual then covers the graphics part of the program and how to move about it.
- Tool path creation and numerous figures show how to use program menus, tool use at a high level, and creation of the cutting paths
- Posting of the Gcode is then presented

You should “read” the complete manual to get a flavor of the software. Then open LazyTurn and try out the different operations paying attention to what the different screens are showing. Then go to section 6 and duplicate the paths using the supplied DXF drawing and tool inputs. Try different tool input values and combinations and pay attention to the results. Review the appendixes for more in depth information.

LazyTurn is a very intuitive program. Don’t hack your way through the program, have pre-conceived notions on how it should work, or relate it to other programs. Reading the manual will provide the information necessary to achieving results with the program.

Finally, be practical in the application of LazyTurn.

### 3.0 LAZYTURNO PROGRAM DEVELOPMENT OVERVIEW

LazyTurn was released to general users community as a free program and doesn't require a license to use. The program was released on 2 /24/2010 and provides for both roughing and finishing cuts of an outside profile.

LAZYCAM TURN will be removed from LazyCam and will be replaced by LAZYTURNO. This program is a DXF or other vector file loader, which then assists in creating a turning Gcode. It is planned to do [boring](#), [facing](#), roughing, [grooving](#) from an imported profile.

This program will, in the end, also allow for profiles to be drawn. At the moment, all you can do is load pre-existing dxf's, [hpgls](#), or [other vector format data](#).

The concept of LT is a quick load of a profile, a fast easy way to set general relationships between objects, and quick button presses to generate paths. Tools are a library of graphical tools so you can see the effects as a path is generated.

The toolpaths remove the stock, and leave a subprofile behind of uncut material which can then be cut with another tool if necessary.

[Drawing is planned and refinement of the finishing paths to add arc coding and few additional enhancements.](#)

There are a few CAM packages out there to draw profiles and create paths, their high cost is due to the complexity of doing all that, and the required support. If the program easily imports a DXF and generates toolpaths, It may decided that is enough. It was never intended to compete with true CAM contour packages.

Just a note on complexity, LT was never intended to replace the high end turning modelers out there, like Dolphin and OneCNC, they do a great job for those that use CAM programs. LazyTurn will continue to be dedicated more to the hobbyist who really just wants an easier way to generate paths from DXF's.

Keep that in mind, so expect simpler things at first, like simple [boring](#), profiling, roughing, finishing, [facing and grooving](#). Hi end analysis isn't likely to occur, I'll feel pretty good if you can draw a quick profile, and perform a multi-tool stock to product Gcode list.

[Inside cuts](#) won't get done until we have totally replaced the LazyCam functionality

In the end the program may allow for drawing of the tool bit, might make for interesting profiles, since the design is really an object to object collision detector, any shape would work in theory not just the current ones.

LazyTurn is a hobby project so there is no set timeline as of this manual update. The stages of development are as follows:

Stage#1 will be rough paths

Stage#2 will be finish-rough paths

Stage #3 is finish paths. ( [ARC CODING IS STILL IN DEVELOPEMENT](#) )

[Stage 4 is facing paths.](#)

[Stage 5, 6, 7.](#) inside, cutoff, various things..... all optional if it is decided to go that far. By end of stage #3 the program should be better than most can get for the price (free) and decisions will be made at that time. I really just want to get to the stage where I can remove turning from LazyCam altogether and replace it with LazyTurn. Seems having it dedicated to turning makes it easier to do than mixing it into LazyCam.

## 4.0 INSTALLING LAZYTURMAN

4.1 LazyTurn ( LT ) currently can be run independently via hard drive, USB stick etc as it's installation can stand alone.

4.2 When installing Mach3 you are asked if you also want to install LazyCam. The current lockdown version of Mach3 installs LazyCam 3.00.2. If you are using an older version of Mach3 you may need to install the LazyCam 3.00.2 Update. Both versions can be downloaded from the ArtSoft site. Here is the link:

<http://www.machsupport.com/downloads.php>

4.3 You can either update LazyCam 3.00.2 by installing the update or create another directory for a stand alone installation, but both versions must be installed for LazyTurn to work.

"LazyCam 3.00.2 Update" is a special build that includes additional required files.

4.4 Now you will need to update the LazyTurn REL 1.1 application file to the most recent version posted. As of 3/14/2010, The latest application file posted is: Re:LazyTurn thread, reply #1349, page 135

Download and unzip the above file and copy the LazyTurn application file into the directory you installed Lazycam. You will replace the existing file that is there.

Once installed, you can play with LazyTurn by running lazyturn.exe in the resultant folder.

4.5 LazyTurn can also be downloaded from the ArtSoft site. Click on downloads, Select Mach and LazyCam and then scroll down to LazyCam and click on the elect highlighted text which will allow you to download a self extracting zip file. Follow the on screen instructions for where you want to install LazyTurn. All required files are included with the download. An icon will be created on your desktop.



### LazyCam (BETA)

LazyCam is a beta-release free importer included with Mach3. Its purpose is to import standard dxf, cmx, and other file types to allow those that do not use CAM programs to more easily generate Gcode to be run under Mach3. The free edition of LazyCam that is included with Mach3 carries no cost. However, there is an optional PRO level upgrade that turns on advanced features such as object offsetting, pocketing, primary foam functions, importation of DXF profiles for conversion to Lathe turning code, etc. It is optional, but gives one the ability to do more advanced importation of objects. **Due to the availability of other low cost CAM packages, LazyCam is in semi-permanent Beta and no longer being actively developed. LazyCam is not intended for use in a production environment.**

(Note: Requires the ".net 2.0 update" from Microsoft to prevent an error at startup.)

*LazyCam 3.01 (including LazyTurn 1.0)*

**Note:** If... **only if** .... you have run LazyTurn before and installing to the same directory You may want to create a backup of the tools you have created. The dat file name is LTurnTools. The one shown below with \_MINE is a backup I created and after installation you will need to rename the backup to just LTurnTools.

 LTurnTools	1 KB	DAT File
 LTurnTools_MINE	8 KB	DAT File

As of this writing rev 5 of this manual is included in the download so always check for the latest revision of the manual in Members Docs.

## 5.0 SCREENS – MENUS - TOOLBAR BUTTONS – STATUS BAR

### 5.1 MAIN SCREEN

The “red” text was added to the screen shot. Take a moment and familiarize yourself with what is shown.



FIGURE 5.1

5.1.1. Figure 5.1 above is the main screen which you will see when LazyTurn is opened. The screen provides different pull down menus as shown in Figure 5.1 below and behaves just like any other screen in a Windows environment.



FIGURE 5.1.1

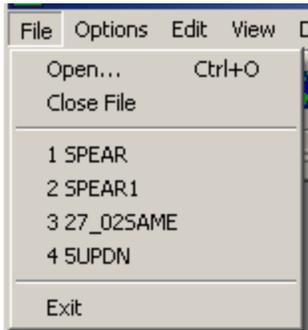
The button functions are included in the menus but differ by providing access to more frequently used selections in the program. Left mouse click to open a menu or a button.

Additionally you can use the keyboard to select a menu as follows:

**Alt** key with individual **letter** F,O,E,V,D,H,T will open associated menu.

Note the coordinates in the lower left of the status bar will change as you move about with the crosshair pointer.

## 5.2 FILE MENU



**FIGURE 5.2**

Open – Opens a windows dialogue box for loading of a file

### *NOTE ON PROGRAM READING OF A FILE:*

*The program "AutoTolerates" so that lines that don't join, automatically stitch together as long as it doesn't create discontinuities in the math functions. This means the program will try a tolerance of .1 for example, and if it finds that this creates a conflict, it will jump back and set the tolerance to .01 and try again, then .001 etc. until it either solves the problem, or rejects the file. ( SEE APPENDIX "B" )*

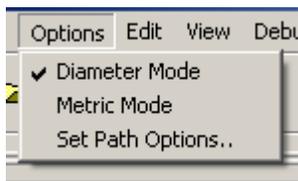
Close File – Closes the current loaded file

Exit – Use to exit LazyTurn

You can also use the keyboard as follows:

**Ctrl** with **O** - opens Window's directory dialogue for selection of a file

## 5.3 OPTIONS MENU



**FIGURE 5.3**

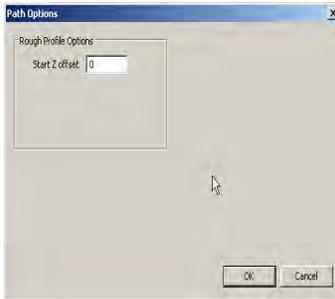
The options menu provides initial setup information about the file you will import and the settings are saved when you exit the program.

The default values are Radius Mode, Imperial Units

You can select diameter mode or metric units by clicking on the text and a tick mark will be left along side the text.

Clicking the Set Path Options provides additional initial setup to LazyTurn.

Currently this is limited to having a Z offset. The start Z offset is how far the tool starts from end of the stock.



**FIGURE 5.3.1**

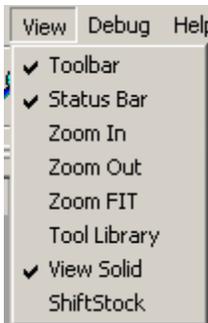
## 5.4 EDIT MENU

Currently not applicable.



**FIGURE 5.4**

## 5.5 VIEW MENU



**FIGURE 5.5**

Toolbar – Turns the top TOOLBAR on or off

Status Bar – Turns the lower STATUS BAR on or off. This status bar provides Z,X coordinates along with other prompts.

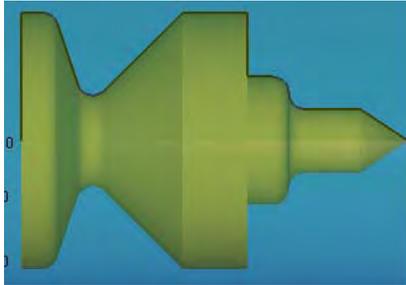
Zoom In – Increases viewed size in the graphics screen

Zoom out – Decreases viewed size in the graphics screen

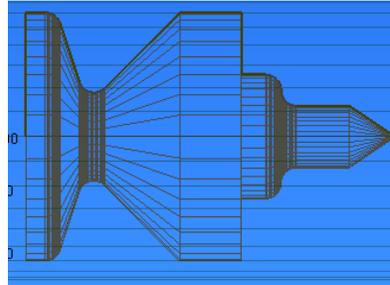
Zoom Fit – Restores the graphics back to the way they were when loaded

Tool Library – Invokes the tool library tab to open. ( see Section 5.12 Lathe Tools Dialogue for detailed information on use of the dialogue )

View Solid – toggles between solid frame or wire frame display of the profile as shown below. Re-click to return to previous frame display.

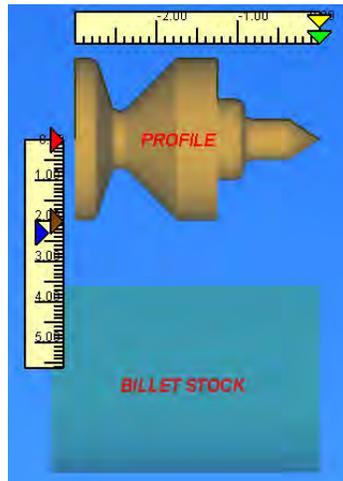


**SOLID**  
**FIGURE 5.5.1**



**WIRE FRAME**  
**FIGURE 5.5.2**

Shift Stock – moves overlay of the billet stock used for the turning to a position below the profile as shown below.

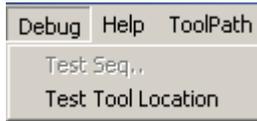


**FIGURE 5.5.3**

Note:

Selecting wire frame will cause the tools to be shown in wire frame also.

## 5.6 DEBUG Menu



Associated Test Tool Location Button

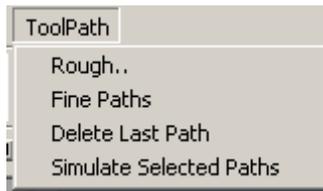
**FIGURE 5.6**

Test Tool Location – Provides for placing the current tool anywhere in the graphics area. To invoke, simply click the button or select from the pulldown.. The placement is based on the center of defined tool tip radius. Use of this tool is covered in Section 7.5.

## 5.7 HELP MENU

Provides preface about the program. For help with LazyTurn you need to refer to this manual.

## 5.8 TOOLPATH MENU



**FIGURE 5.8**

This tab provides choice of a rough or fine path generation and clicking on either will invoke specific menus for input. ( See Section 7 for a detailed explanation. )

Delete Last Path – will delete the last created path. The created passes are show in the PROJECT INFO box. (See Section 7 for additional information)

Simulate Selected Paths – Invokes a dialog for simulating the created paths. The tool will be shown moving along the paths in the Graphics Area.  
( See Section 7.5 for additional information )

The following parts of *SECTION 5* pertains to the **BUTTONS** available in the TOOLBAR. A prompt for any selected button appears in the status bar.

## 5.9 FILE FOLDER BUTTON

This file open button automatically opens a Windows screen for a file search on your computer.

**Ctrl** with **O** - opens Window's directory dialogue for selection of a file

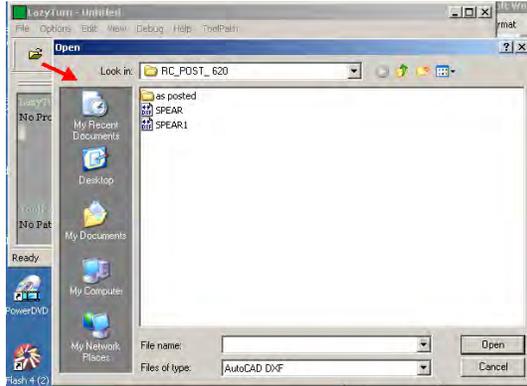


FIGURE 5.9

## 5.10 ZOOM & FIT BUTTONS

These buttons allow for increasing or decreasing the magnification of the displayed profile via left mouse clicking. Selecting the world globe button will fit the profile back to its originally imported size.

**Ctrl** with **R** - will fit the profile as it was originally imported also



FIGURE 5.10

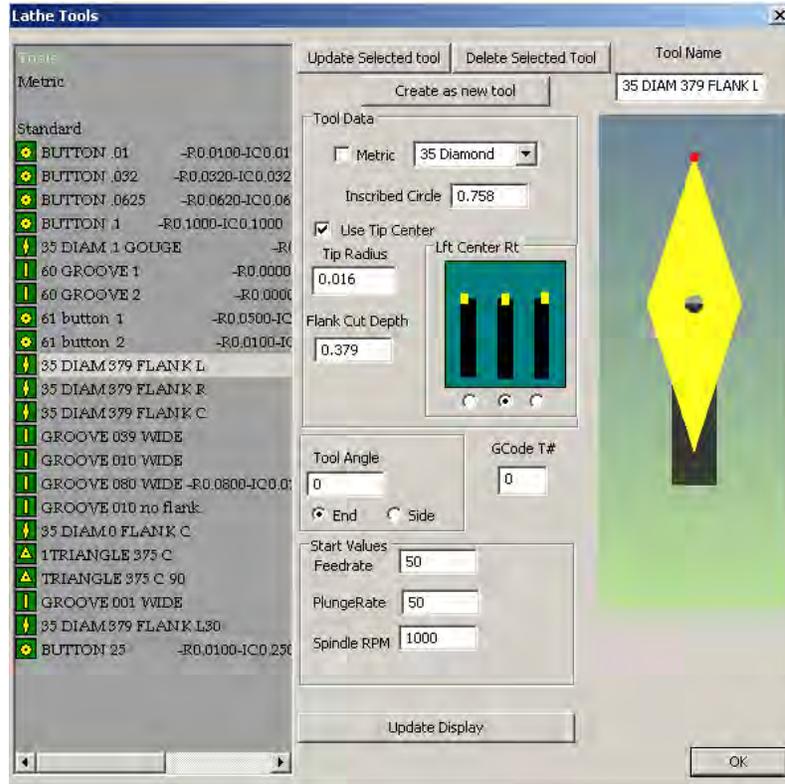
## 5.11 SELECT TOOL BUTTON

This button opens the lathe tool dialog screen.



FIGURE 5.11

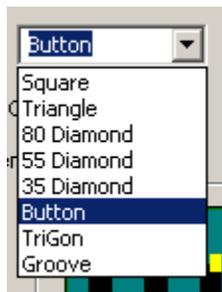
## 5.12 LATHE TOOLS DIALOG



**FIGURE 5.12**

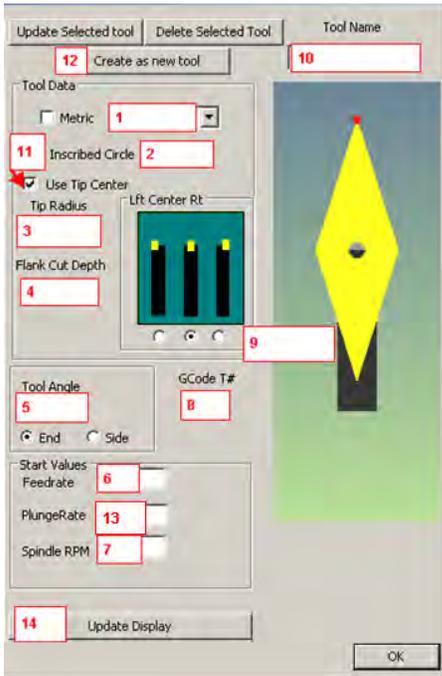
The Lathe Tools dialog provides for creating, defining, selecting, updating a lathe tool which will be used in the various tool path generations.

You will need to create a tool suitable for the desired machining operation. The screen will show the created tool on the left side along with description. The description will be posted along with the generated gcode when posted. A choice of common lathe tools can be selected by clicking on the box and then selecting the desired tool.



See Appendix “E” PRACTICAL MACHINING CONSIDERATIONS for information relative to tool use when creating a cutting path for a profile.

**5.13 DESCRIPTION OF INPUT BOXES FOR LATHE TOOLS**

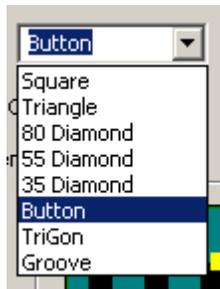


**FIGURE 5.13**

Note that multiple tools can be used for a project. All the tools can be used for the different paths that you will create. Common sense dictates you should use the correct tool for the intended path. The following explains each of the Red numbered boxes shown in figure

5.13 which require selection or input.

1. **Tool selection** - opens the flyout shown below for selection of a tool. Note that by clicking the box to the left ( show in the above figure ) all input values will default to metric.



**FIGURE 5.13.1**

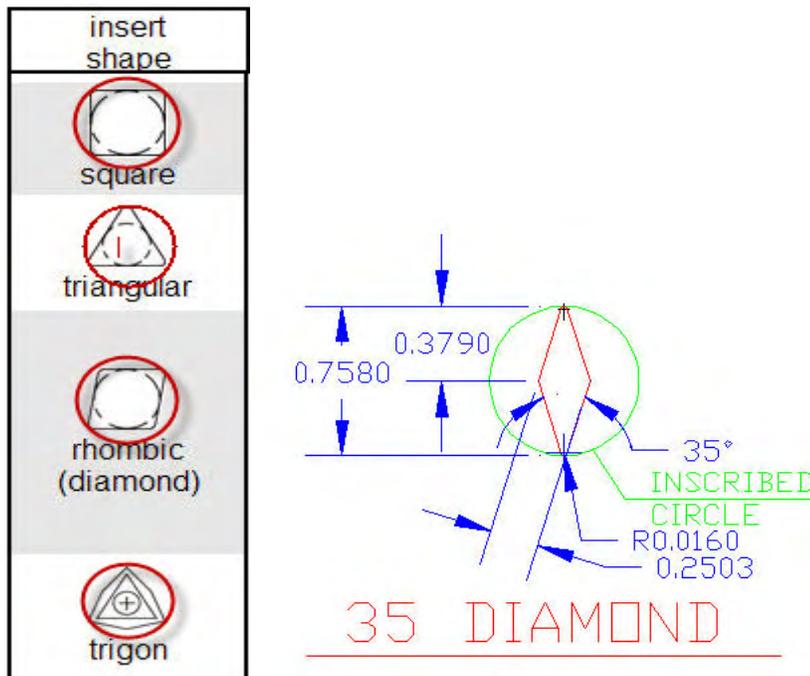
2. **Inscribed Circle** - Defines a circle, which when drawn about the center of the tool insert, will include the entire tool insert. For example in a triangle insert, the inscribed circle is the size of a circle placed in the center of the triangle with the diameter including the complete tip. The inscribed circle is set to a min value of 0.010” .

The program will not allow you to use a value which is mathematically less than the above definition and will provide a default value in the input box.

The figure below shows different inserts provided by a manufacturer.

Note that the manufacturer specifies the inscribed circle which is shown inside the insert shape by a dashed black line. Inscribed circle as defined in LazyTurn is represented by the RED circle. This was done to eliminate some manufactures tolerances thus providing true accuracy based on input value as shown for the 35 diamond tip from actual measurements.

Do not confuse the two.

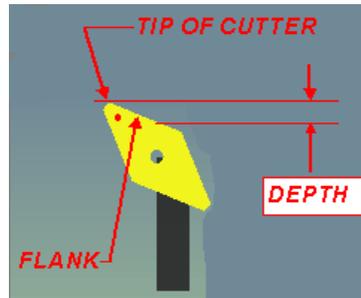


**FIGURE 5.13.2**

**NOTE:**

*A square tool with an inscribed circle of .01 cannot physically have a tool tip radius of .01, they cannot fit in the equation for a square. Hence, the tip is impossible by physical definition, and the tip will not be created. This will cause a fault when the program tries to match a collision. The fault will generate a "tip failure". If you were to try to create the tip with above radius, the tool looks round since its trying to fit the .01 tip radius on each of the 4 corners of a tool which has an inscribed radius of only .01. The tool generator will tell you its impossible. Note. that for the green inscribed circle shown for the 35 diamond tool, that based on that diameter and tip radius, all the other dimensions are by default. Additional comments are made in Appendix "E".*

3. **Tip Radius** – actual radius on the tip end of the tool cutter. Min value is 0.0005”
4. **Flank Cut Depth** – defines how much of the flank side of the cutter can actually cut based on a depth setting. The input value should be from the tip of cutter to the furthest away point along the flank. It should not be as what is deliberately shown in the figure below ( although you can if so desired ).



**FIGURE 5.13.3**

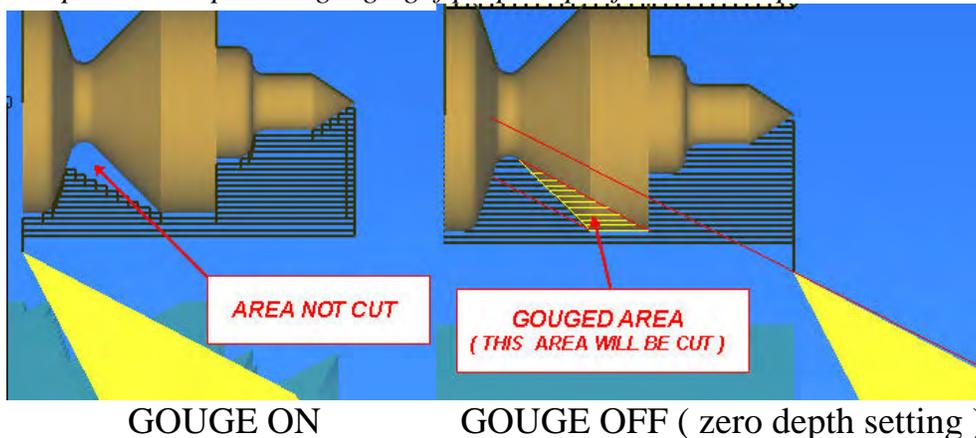
This depth setting can greatly affect your end tool path.

- depth too small and the tool will gouge
- depth too large and the programmed generated path to actual paths will not reflect actual or true cutting edges of the tool.

Gouging occurs when a part of the tool not intended to cut the billet actually does. A value other than “0” is required to turn gouge protection on. See the following figure.

**NOTE:**

*LazyTurn uses the tip information to compare the tip cutting edges to the current shape and will prevent gouging if proper tip information is provided.*



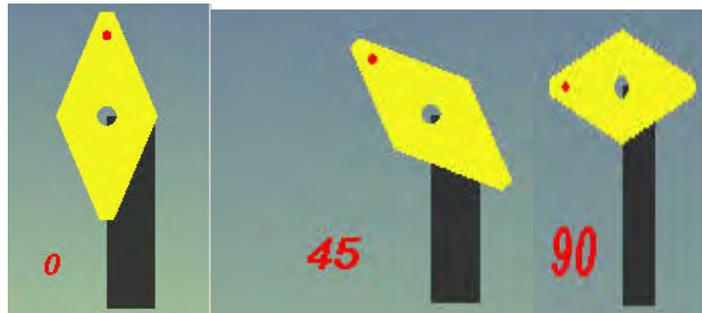
**FIGURE 5.13.4**

You can use the center tool post or use a value of zero if you do not care about stock gouging.

5. **Tool Angle** – This is the angle of the insert about its center and **not** the angle of the complete tool including the holder. For a button tool the angle means nothing and is internally set to zero degrees.

Click the End or Side circular box to rotate the tool 90 degrees.

Examples of tool bit orientation about the left tool holder:



**FIGURE 5.13.5**

Note that only a left or right facing tool can be angled in a tool holder. Additional comments on tools can be found in Appendix “E”.

6. **Feed Rate** - Enter your desired cutting axis feed rate. The feed rate value “F” is included in the Gcode when posted. Note that the F is restricted to the slowest axis F in actual machining as configured in MACH3 Turn. Consult the Using Mach3 Turn manual Section 10 for additional information .

Different values can be used for the rough and finish paths.

7. Enter the **Spindle RPM**
8. **GCode T#** - Enter a defined tool number. The tool number but should not conflict with a same tool number defined in MACH3 Turn. Additional comments on tools can be found in Appendix “E”.

Consult the Using Mach3 Turn Manual Sections 7 to 9 for information.

9. **Tool Holder** – Three choices of tool holder positions are provided namely left, center, and right. As noted before, the cutter bit attaches to the tool holder and its orientation changes accordingly. Left cuts left, right cuts right, center can cut left or right depending on selected cut direction in pass dialog flyout.

10. **Tool Name** - This box allows you to name the tool you are creating or rename /

update an existing name. The name, tip radius, inscribed circle are tabulated on the left side of the lathe tools dialog ( see FIGURE 5.12).

11. **Use Tip Center** - when selected will discriminate between using Mach3's offset registers with compensation, or just to create the code as pre-compensated using the tip radius as that compensation from tip center.

Additional comments on tools can be found in Appendix “E”.  
Consult the Using Mach3 Turn Manual Sections 7 to 9 for information.

12. To **Update** a tool simply select the tool on the left by clicking the left mouse button, modify an input and click the “Update Selected Tool box”.

To **Delete** a tool simply select the tool on the left by clicking the left mouse button and click the “Delete Selected Tool” box.

To **Create** a tool simply fill in all the inputs, update the display if you so desire to see what it looks like, type a name in box and then click the “Create as new tool” box.

13. **Plunge Rate** – Provides for an **X feedrate** that is different than the Z feedrate.

*Here's why: when the tool is moved (X direction) into the stock to begin the next (Z direction) cut, it plunges into the material and initially has to cut a lot of material all at once. After it starts traveling in the Z direction it's only cutting from half of the cutter tip and I've chosen a feed rate that's appropriate for this. But every time it plunges in for another cut it cuts from both sides of the tip simultaneously, removing too much material too fast. If I slow the feed rate to match this X cut, I waste too much time on the Z cuts. I need different, independently set feed rates for cuts in the X and Z directions.*

14. **Update Display** – clicking this button will update the graphic display of the tool.

## 5.14 CUT BUTTONS

LazyTurn provides a choice of how machining of the profile can be done. Normal use would be to make larger roughing cuts to remove the bulk of the material from the billet and then do smaller finishing cuts. You can do multiple roughing and finishing cuts with different tools until the machining is complete. The generated pathing will follow the order of the cuts you create.



### 5.14.1 ROUGH CUT BUTTON

The rough cut button, when clicked, will invoke the pass depth dialog for defining tool path parameters for a roughing pass. Multiple roughing passes are allowed. A tool must exist and be selected prior to using the button. There is choice of diagnostics in the dialog and the default is Visual Intersect Diags if nothing is selected. Diagnostics is covered in section 5.14.3.

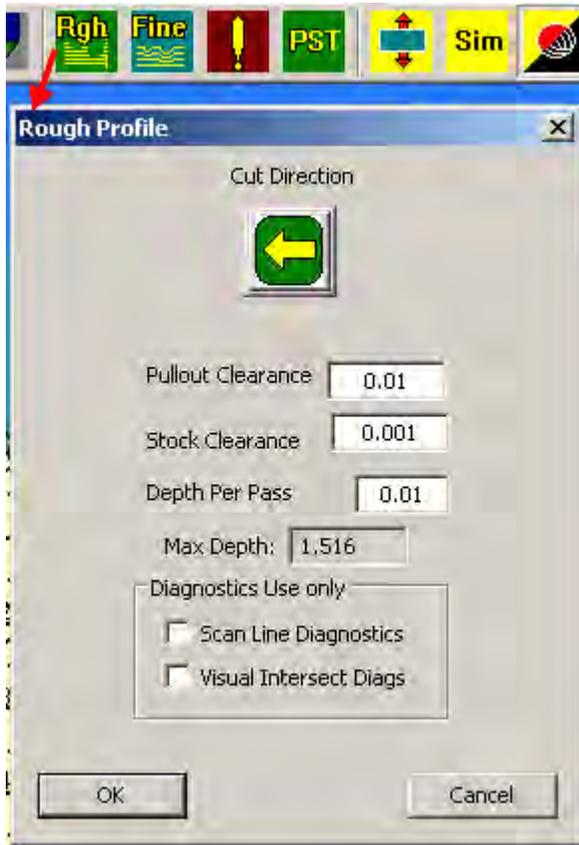


FIGURE 5.14.1

**Cut Direction** – clicking this button will alternately change a middle tools cutting side from left to right. Default is left side. A middle mounted tool can cut in either direction. The button is not applicable for a left or right tool and an error message will appear if clicked.

**Pullout Clearance** - input value defines how far the tool will retract in the X direction during pathing moves. It is not currently restricted.

**Stock Clearance** - input value defines the amount of un-cut stock that remains after a pass.

Max Depth – No input required or allowed.

*The number is simply a limitation, based on the tool, of how deep you can cut. A tool can only be used to cut to its inscribed circle depth, and the end max depth is the inscribed circle, but compensated by diameter or radius mode selection. It stabilizes the operation of the programs volume calculator.*

**Depth per Pass** - is the how deep each individual pass will cut.

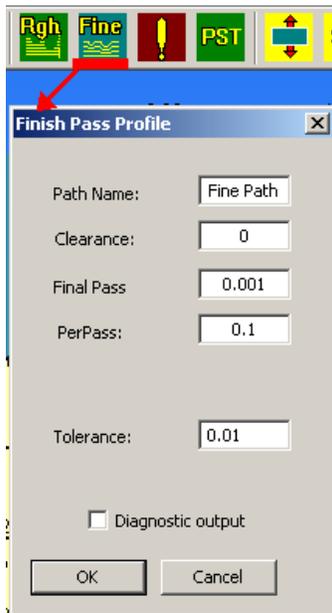
**Cancel** – clicking on the button will cancel the path process

**OK** – clicking on the button will allow the path process to start

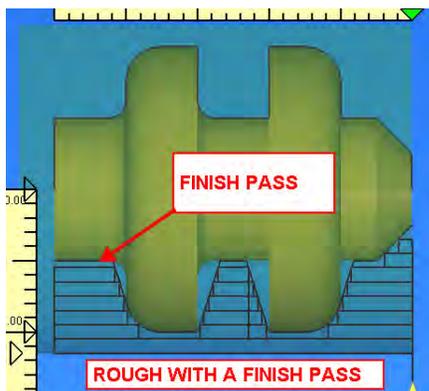
### 5.14.2 FINISH CUT BUTTON

The finish cut button, when clicked, will invoke the pass depth dialog for defining tool path parameters for a finishing pass. Multiple finish passes are allowed. A tool must exist and be selected prior to using the button. A rough cut is not mandatory in order to do a finishing pass, but, the resulting pathing will be different. See Figure 5.14.2.1 & 5.14.2.2 below) The gcode currently posted for finishing passes does not include [G2 or G3 / circular interpolation](#).

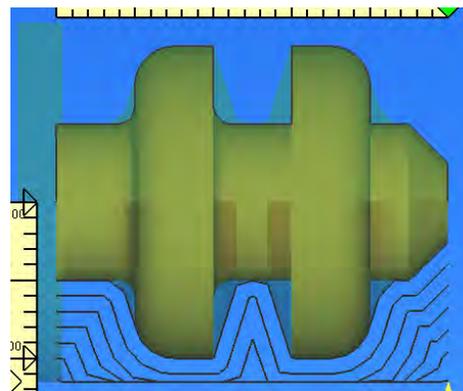
Finish pass flyout is shown below.



**FIGURE 5.14.2**



**FIGURE 5.14.2.1**



**FIGURE 5.14.2.2**

In figure 5.14.2.2, no rough pass done, the finish pass will follow the profile contour. The last finish path is the same but the preceding ones start at the outside diameter of the billet.

**Path Name** – allows for providing a unique name for the finish. The name appears in the Project Info.

**Clearance** – The amount of material to be left on the profile after the finish pass is done

**Final Pass**– The depth of the last path / cut of the finish pass

**Per Pass**– Max pass depth of any pathing associated with the finish pass

**Tolerance**- This setting defines how many times the software will look ahead or along the profile as it is creating the pathing. The range of acceptable input values are 0.0001” to 10. *Currently there are few tolerance issues being tested for tolerances lower than 0.001” and the finish pass may error with values less than 0.001”.* The finer the tolerance the closer the LazyTurn will monitor it’s analysis. The setting will impact how long it takes to generate the pathing. So it will take more time for a long piece of say 30 inches overall length than one of say 6”.

**Diagnostics** – *later*

**Cancel** – clicking on the button will cancel the path process

**OK** – clicking on the button will allow the path process to start

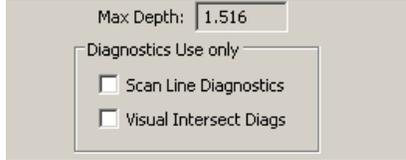
**NOTE:**

As of this update:

- Z Offset enabled for finish passes. ( this may not be optimal, Z Offset is a clipping value, may lead to tool outer flank cutting ). Use of this will be explained later.

### 5.14.3 DIAGNOSTICS

You have two choices in the selection of what diagnostics will be done by clicking one the boxes shown in the figure below.



**FIGURE 5.14.3**

*NOTE: You currently can post code if you use the diagnostics. This WAS NOT allowed in older versions and is being clarified as of this writing .*

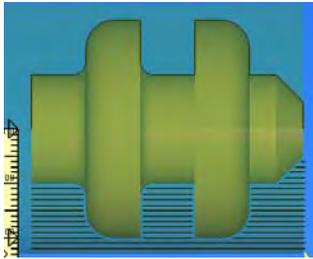
### 5.14.4 SCAN LINE DIAGNOSTICS

The intended use of this diagnostics is as follows:

The resultant image of the path is helpful to find where some types of errors come from.

It is also a very fast way to see a line reduction of the image. No gouging tests are done, and tool geometry is not considered, no real tool path is calculated, it just shows the "before pathing" data from the LazyTurn path engine. Generation of the scan lines are done in an iterative algorithm, so the diagnostics output can show if that is working.

An example of the simplified and rapid display of the tool paths is generated as shown in Fig 5.14.4 below.

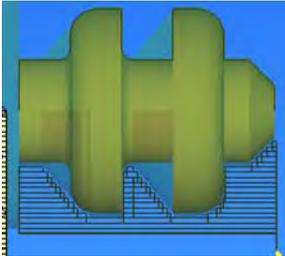


**FIGURE 5.14.4**

### 5.14.5 VISUAL INTERSECT DIAGNOSTICS

The intended use of this diagnostics is as follows:

This diagnostics does full calculated paths based on collision and displayed as shown in figure 5.14.5 below. Notice how actual paths / moves are shown as compared to figure 5.14.4 above.



**FIGURE 5.14.5**

Volume calculations are being done via a SVC ( Swept Volume Calculator ). To assist in diagnosing a possible problem you can set the number of passes, to a value, less than the number of passes normally generated by the SVC. This is done in the flyout that appears after clicking OK in the pass depth dialog ( figure 5.14.5.1).

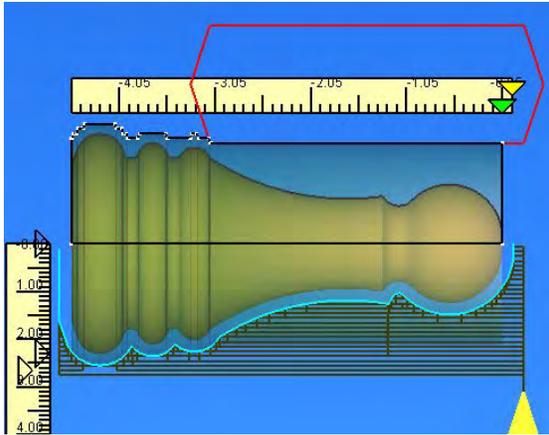


**FIGURE 5.14.5.1**

If there is a problem during calculations, and the pass before the problem pass can be found, it will assist in finding and addressing the problem. The DXF drawing, tool numbers, and pass data should be posted to the forum.

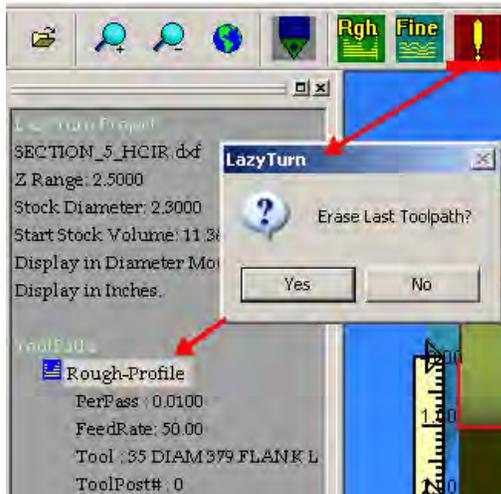
*When you do a toolpath, if the end material is wrong, You can find a particular pass where the shape of the tool is NOT subtracted properly ( this can happen from a wrong inside / outside assumption at some point). If you do get a failed stock, try narrowing in on what pass it failed on. Try 100, if it fails, try 50, if that passes try 75, etc..... eventually you will find a point where the subtraction fails.*

The stock will be chewed down each pass till the max Pass Number you gave was seen. Calculating stops at that point. Figure 5.14.5.2 below shows a few things. The greenline is the amount of offset from the part used in the profile gouging checks, ( just a diagnostic ), the black line on top is the actual stock outline at that point, White dots indicate endpoints of the material stock outline to that point. The large red shape, is the swept tool volume of the last pass done. It is the shape of the tool, swept by the amount of that pass, and subtracted from the stock volume, it was successful on that line as shown by the fact the tool shape is properly subtracted from the material.

**FIGURE 5.14.5.2**

### 5.15 ERASE TOOLPATH BUTTON

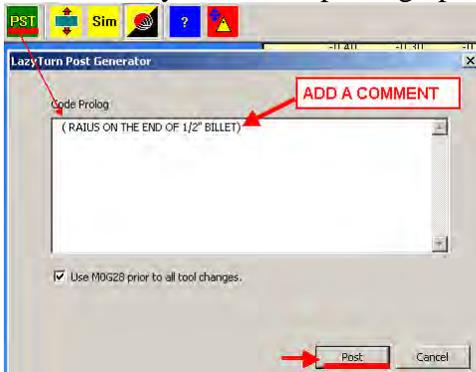
Clicking this button will invoke the flyout where you choose to delete or not delete a path that was created. Note that you first select the path you wish to delete by double clicking on a path (green text) shown in the project information. You cannot replace a path once deleted, so if you delete the very first created pass and want to replace it you will need to delete all passes and start over. You cannot re-order the passes. You can however select a pass and turn its display in the graphics area on or off by double clicking on it..

**FIGURE 5.15**

### 5.16 POST GCODE BUTTON

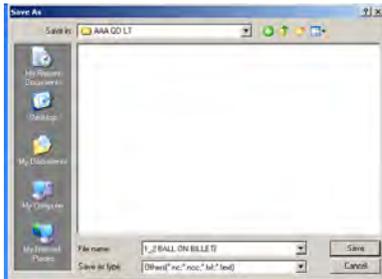
Clicking this button will invoke the Post Generator This is how the Gcode created by paths will be saved.

Click the posting button and add comments as you wish. Additionally select / click the boxes for any additional posting options, then and click post.



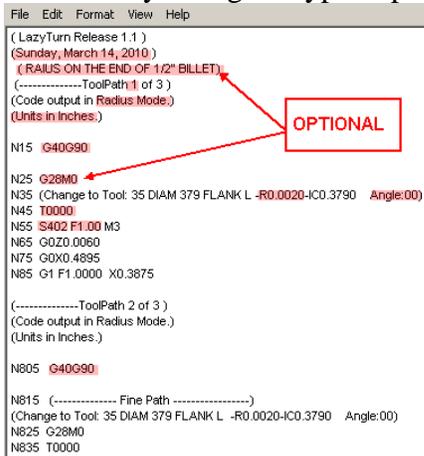
**FIGURE 5.16**

Save the file along with a name to suite your preference. ( typical windows flyout )



**FIGURE 5.16.1**

The figure below shows the saved code, just click File and save the file if you don't wish to make any changes. Typical posted items are highlighted in red.



**FIGURE 5.16.2**

The posted code currently contains the following information and some are optional:

- Version of LazyTurn
- Date file was created
- Optional - Comments you added
- The toolpath #
- Code output basis (diameter or radius mode )
- Units used for the Gcode ( mm's or inches )
- G40G90 at each toolpath posting ( cancels use of tool radius compensation and distance mode is absolute.
- Optional – G28 M0 ( at each tool a change / provides for return home for tool change )
- Tool Angle and tip radius included in tool comment output
- Line numbers on code output ( increments of 5 )

**NOTE:** The posted code will include “all” toolpaths that are currently displayed in the Project Info. If a toolpath is turned off it still will be posted.

- *Z Offset enabled for finish pass. ( this may not be optimal, ZOffset is a clipping value, may lead to tool outer flank cutting ) more explanation later.*

**5.17** The shift stock , view solid and about buttons were previously defined and are buttons for the respective descriptions in the view menu.



The shift button was discussed in [Section 5.5](#)



The simulate button is discussed in [Section 7.5](#)



The view solid button was discussed in [Section 5.5](#)



The about button provides general information about the program.



The debug tool location button and how to use it will be discussed in detail in the [Section 7.5](#)

## 6.0 GRAPHICS AREA

This section will provide a general overview of the Graphics Area. An understanding of the graphic display and manipulation of the profile is required prior to reading Section 7 of the manual. That section defines how the tools you have created will be used to create Gcode paths.

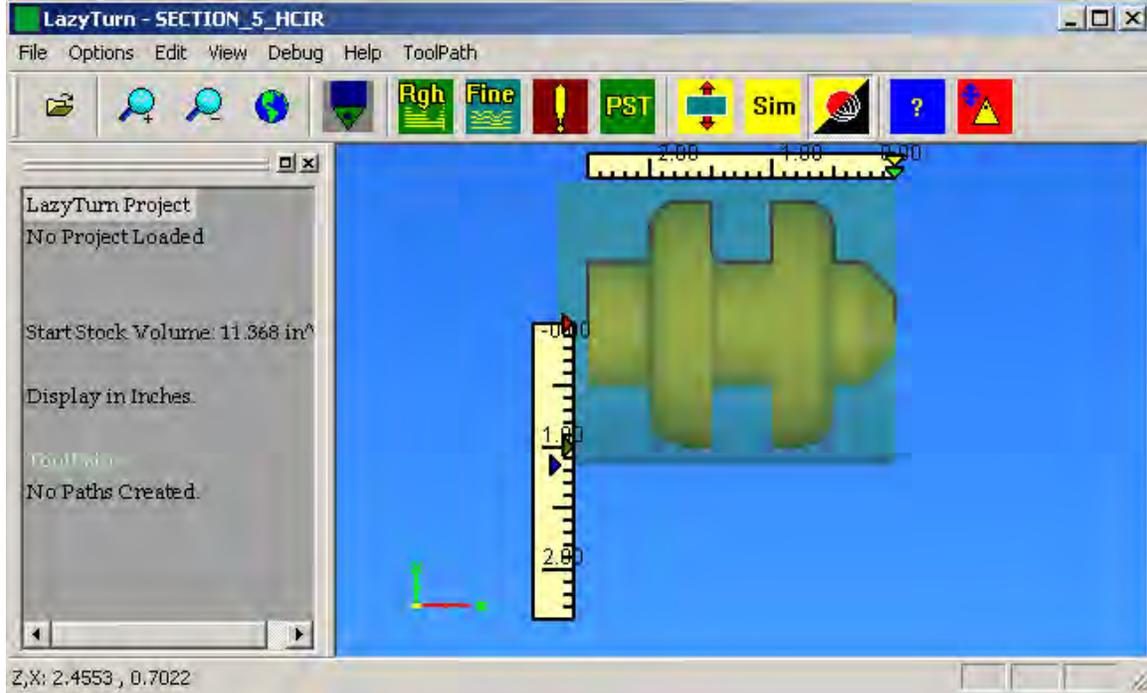
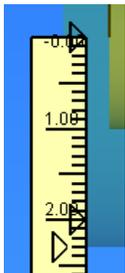


FIGURE 6

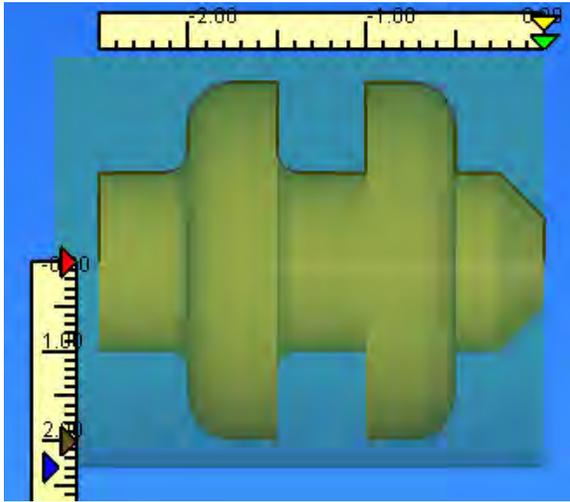
The graphics part of the main screen, delineated by the blue background, provides a plan view of the imported DXF file. Colors are program defaults and can't be changed. The turn object is three dimensional and object manipulation is covered in 6.5 of this section.

Menus and buttons were already discussed and are not repeated here.



*On some computers the pentagon on colors may disappear once a tool is selected.*

## 6.1 PROFILE and RULERS



**FIGURE 6.1**

The black line above the profile delineates the single connected group of entities you created in a CAD program. LazyTurn will swap around and reconnect as it sees fit, so your drawing order can be as arbitrary as you like. In the end though, it must all connect to a single chain. See Appendix “B” and “C” for specific information.

The shaded object represents a 2D view of the profile to be turned. The shaded cylinder around the turn object represents an uncut stock size of diameter or radius ( a default setting ) and length. The change in stock shading is the center of the stock. You can change the program generated stock / billet size by using the scale pentagons on the ruler.

## 6.2 RULERS

The rulers above and along side of the stock are smart, zoom and pan with you, and are programmed to stay on the screen. The rulers divisions will change as you zoom in or out. The divisions are per your default units. The ruler is just a visual rough guide.

### 6.2.1 RULER PENTAGONS

The pentagons on the ruler indicate stock size and length as follows:

- Left side ruler - Stock size ( **blue** )
- Profile diameter ( **brown** )

*Note: Changing the diameter value via the slider will scale the piece up or down equally.*

- Center of stock ( **red** )
- Top side ruler - Z stock location ( **green** )

The bottom green pentagram is for setting the location of the stock Z origin. ( most users prefer to have the Z start at 0 ) and go negative towards the chuck.

- Z start offset ( **yellow** )

- Z Offset enabled for finish pass. ( this may not be optimal, ZOffset is a clipping value, may lead to tool outer flank cutting )

### 6.3 PENTAGON VALUE INPUT

When the mouse pointer is moved over a pentagon the actual value for the feature is displayed. Clicking on any pentagon will allow you to change the value using the flyout shown below. You can input a value other than what the program generated to set a stock size, or even the profile diameter prior to asking for a tool path generation.

The pentagon for the stock will turn brown and overlay the profile pentagon if the values are the same. You can click on the single pentagon and change the values. Note the flyout prompt informing what the change pertains to.



FIGURE 6.3

The stock size graphic will change relative to the profile and is a good visual indication of your values at any time. The example below shows a smaller stock than the profile.

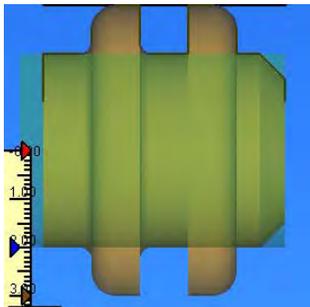


FIGURE 6.3.1

*Note:*

*The program automatically generates stock size values, but you have the choice to adjust the value to suite your actual stock size. Stock size should always be larger than your profile, it need not be, but any paths created will cut off part of your profile. You can't Redefine the Z stock size / length*

*You cannot change the X center stock location of 0.000 ( red colored pentagon).*

## 6.4 CROSS HAIR

The crosshair will move as you move your mouse and its Z,X coordinates are actively displayed in the lower left hand corner of the status bar. Zooming in on the part allows for more precise measurement and inspection of the turn object if so desired.



FIGURE 6.4

## 6.5 MANIPULATION OF THE GRAHIC DISPLAY

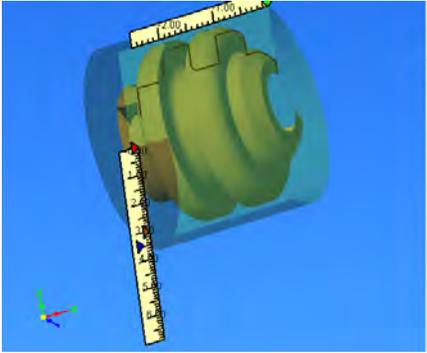


FIGURE 6.5

The three dimension turn object along with the rulers can be viewed from any angle. Note how the coordinate icon ( lower left corner ) also rotates.

Holding the mouse wheel down while moving the mouse allows spinning the object about a point. Release the mouse wheel at any time to freeze the display at any time during rotation.

Double click the left mouse button to reset the display.

**Ctrl** key and **left mouse** while moving the mouse allows you to dynamically move the profile anywhere in the display area as shown below. Releasing one of them will freeze the display at any time.



FIGURE 6.5.1

**Shift key and left mouse** while moving the mouse up and down on the pad allows you to dynamically zoom in and out on the profile as shown below. Releasing one of them will freeze the display at any time.

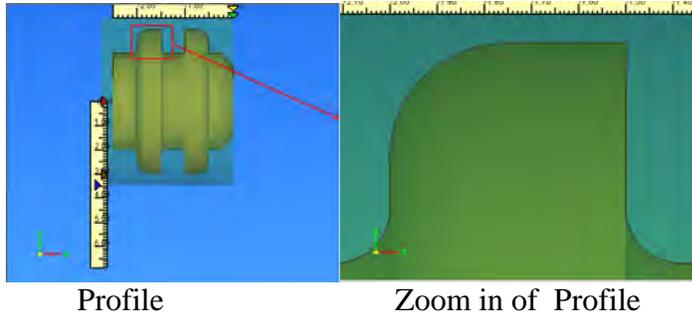


FIGURE 6.5.2

## 6.6 BUTTONS

The buttons shown below are frequently used and duplicate items from the View, ToolPath, and Help menus. They were defined in Section 5. From left to right they are:

Shift Stock, Simulate, View as Solid / Wireframe ( toggles ), Help



FIGURE 6.6

## 6.7 PATH PORTRAYAL

The following is provided so you have basic understanding of what a path is, can relate to the graphic portrayal of the path, and will prepare you for Section 7 in which tool paths are actually created.

Path in LazyTurn is a simple name given to the X and Z individual or combined axis cuts made into the stock. The extent of path movement is shown as a straight black line. The gcode generated from the path will be a plus or minus direction depending on which way the tool travels. The paths don't show direction. Portrayal of the path or paths will vary depending on how the program applies the tool and cut parameters and how your viewing the display area. Below are some examples.

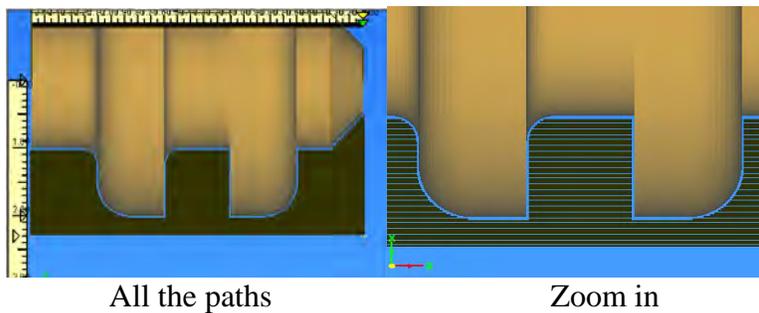


FIGURE 6.7

Zooming way in you will see the single path and any clearance left by unfinished cutting of the profile.

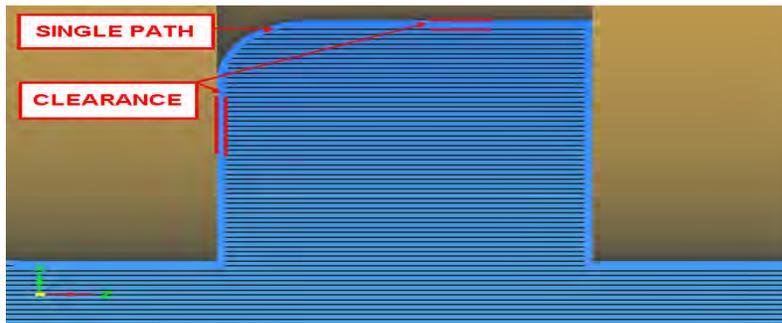
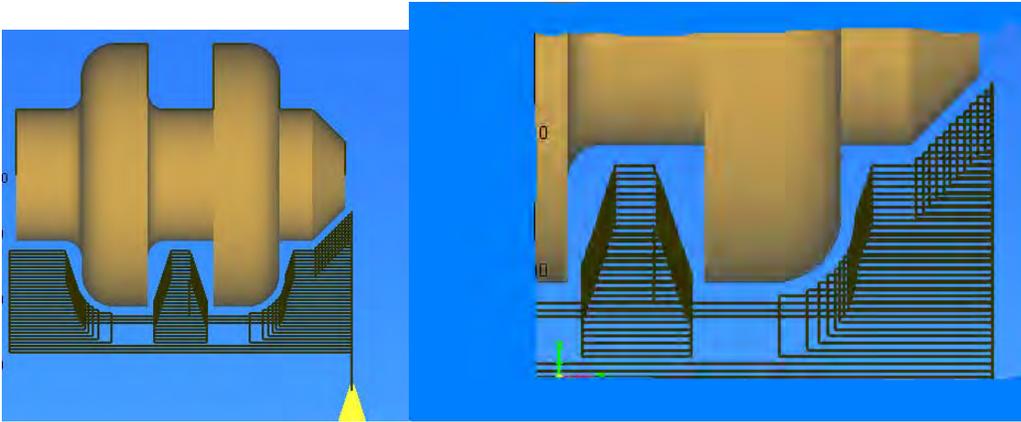


FIGURE 6.7.1

Figure 6.7.2 shows paths made by a different kind of tool with different cutting parameters. LazyTurn provided single and combination paths such that the tool cuts along, retracts ( moves out ), back in, etc. in a continuous efficient pattern to remove stock from the profile. Additionally LT provided paths that would avoid gouging because it was selected by entering a value other than 0 for the flank cut depth value ( see Figure 5.13 ).

**FIGURE 6.7.2**

To dynamically watch the cutting in operation just click the “Simulate” button .  
You will need to create a tool and a tool path before you can Simulate.

## 7.0 TOOLPATH GENERATION

This section describes how paths are created. Program information which was covered in previous sections of the manual will not be repeated although reference to sections may be noted. The Menus, Buttons, along with their associated dialog and flyouts will be used.

Information for the Cut, Delete, Simulate Buttons were provided previous sections.



**FIGURE 7.0**

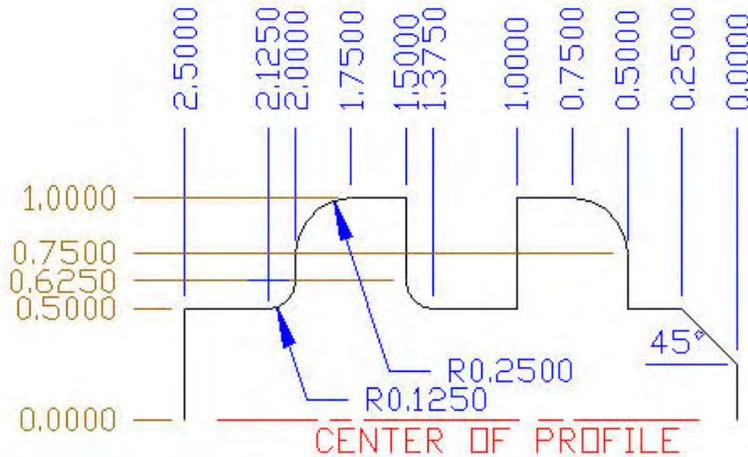
Note that the information presented here is not about teaching you to use a lathe. It is presenting information on how to use LazyTurn. Some of the information may or may not be practical. You may want to review Appendix “E” Practical Machining Considerations.

### 7.1 TOOLPATHS GENERATION OVERVIEW

LazyTurn will import a DXF drawing and generate a solid view of the profile. It calculates the profile volume. The created tools are used to “widdle” the stock down in a progressive manner until the paths for the profile are completed. The progression of cutting down the stock to a finished profile is done by creating toolpaths. The toolpaths are based on the users definition of the tool used. Three passes will usually be required. A rough pass to remove as much stock as possible quickly, a rough-finish pass to clean up the rough pass in preparation of a final finish pass. For each created pass the volume is computed and noted along with other specifics in the Project Information. You can simulate the pass at any time and if doesn’t meet you expectations or there is a problem you can delete it and try a different approach until in the end only the volume of the profile is left.

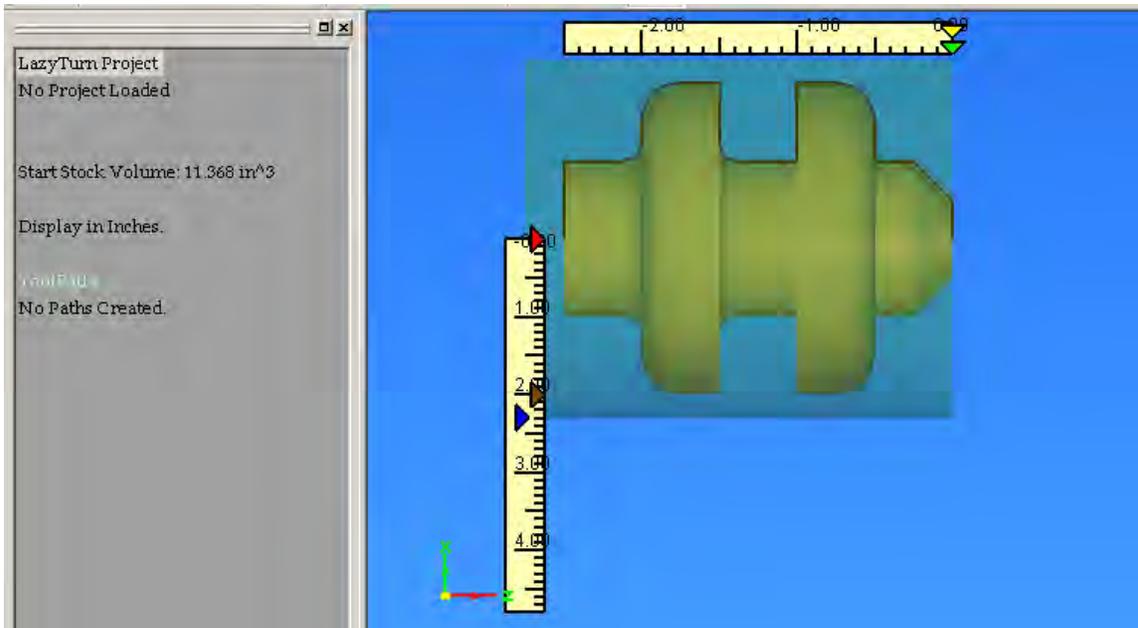
**7.2 DXF DRAWING**

You will need to create a drawing of your profile in your CAD program and export it as a DXF. It is highly suggested that you review Appendix “B” and “C” before creating a drawing. You can download the DXF file ( SECTION\_5\_DIMHCIR.DXF ) used in this section of the manual from the MACH3 Forum. It is located in the Member Docs. 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.



**FIGURE 7.2**

LazyTurn will convert the drawing into a solid as shown below. Tools used in pathing can now be created. LazyTurn remembers all the tools you created and they will be available when you open a file.



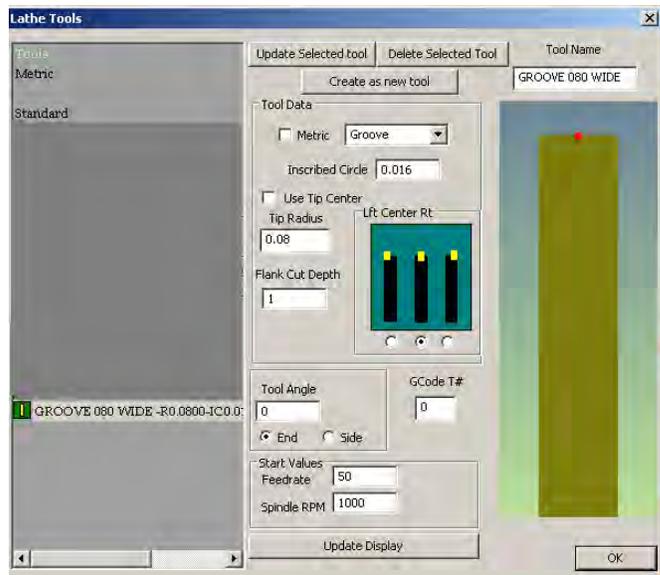
**FIGURE 7.2.1**

### 7.3 TOOL CREATION

Section 5.12 provided complete information on tools and their creation. Information is not repeated, the manual builds on each section, so refer to the appropriate section of the manual.

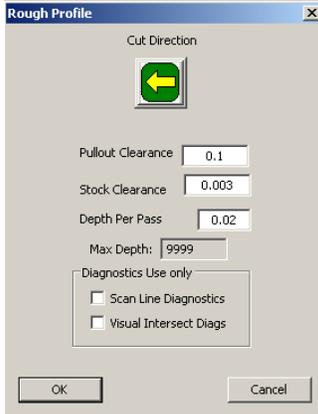
Lets take a moment and think about the tool we will create and how we want to machine the stock down to the profile. Depending on the complexity of the profile multiple rough, rough finish, and finish cuts may be required. For this example, a single tool will be used for the rough. The practical application of a single tool may not be appropriate for the actual cutting, but we will only use only one tool for illustration purposes. The one tool approach, right or wrong, will be used. LazyTurn only knows what you tell it!

The tool we will create is the grooving tool shown in Figure 7.3.1, ground on both sides and the face and ground deep enough that it can move all the way to the profile.  
( ref sect 5.12 & 5.13 )



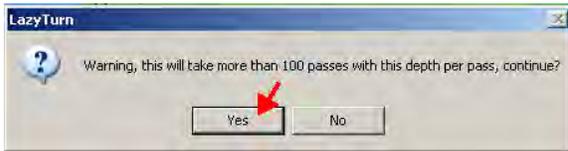
**FIGURE 7.3.1**

Once the tool is created we can select that tool ( by clicking on it and clicking OK) and the define how paths will be generated as shown in Figure 7.3.2.



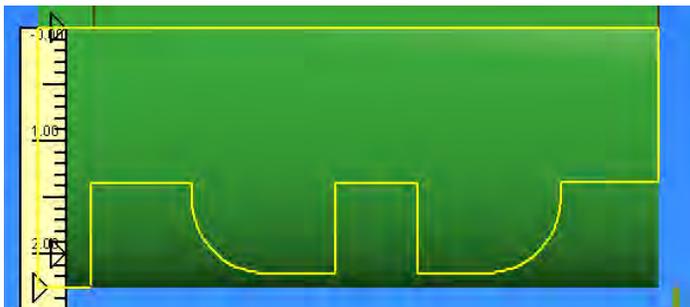
**FIGURE 7.3.2**

Click yes when this screen appears and LazyTurn begins the path creation.



**FIGURE 7.3.3**

LazyTurn graphically shows the path creation.



**FIGURE 7.3.4**

Then the following screen shows the actual paths if the path creation was successful.

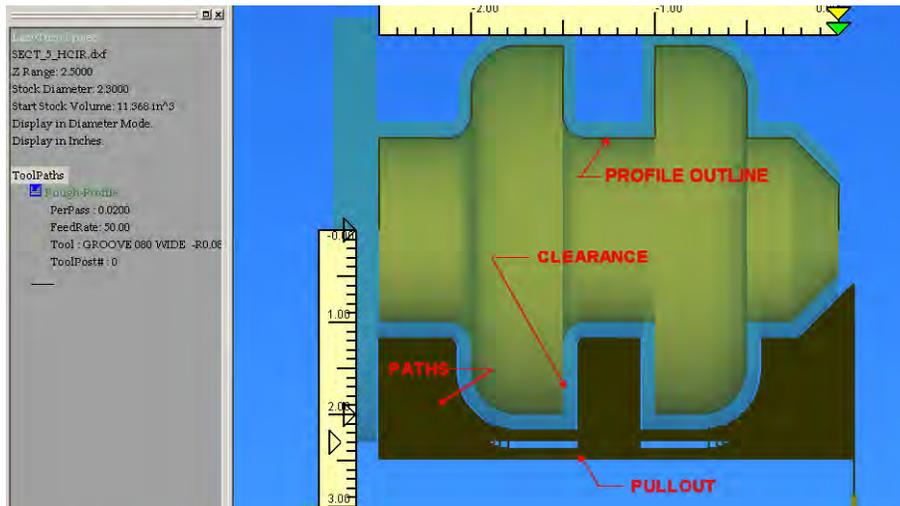


FIGURE 7.3.5

A rough path was created. We left 0.003 clearance to be removed with a finish pass. We change our minds and we decide to only allow 0.002" for the finish path. The same tool is selected, and the stock clearance changed. LazyTurn informs that the first rough cut was all that was needed to go on to doing a finish path. No additional rough pass is created. You can delete the pass and redo the rough pass but there really is no need to do so.



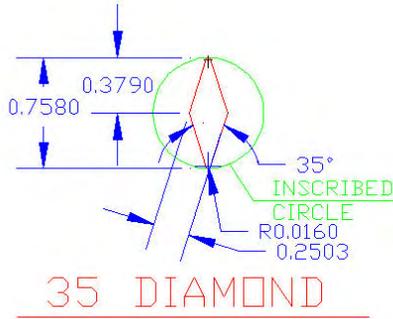
FIGURE 7.3.6

Click OK.

Finishing path creation is covered in the next section 7.4.

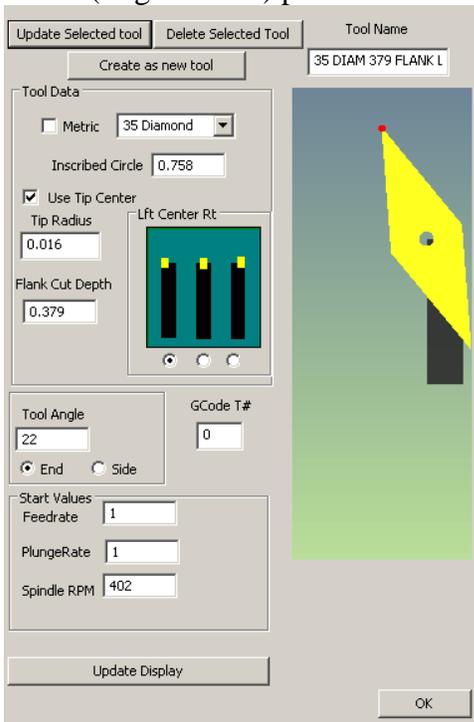
### 7.4 MULTIPLE PATHS AND TOOLS

This is a different illustration and utilizing advanced LazyTurn features for creation of paths. The same file is used, different tools are used, multiple paths will be required. A 35 Diamond tool will be used which is defined per figured 7.4.1 below. The tool could be used to create one rough pass if gouging was turned off (see Figure 5.13.4), but, the intent is to build on prior information for this illustration.

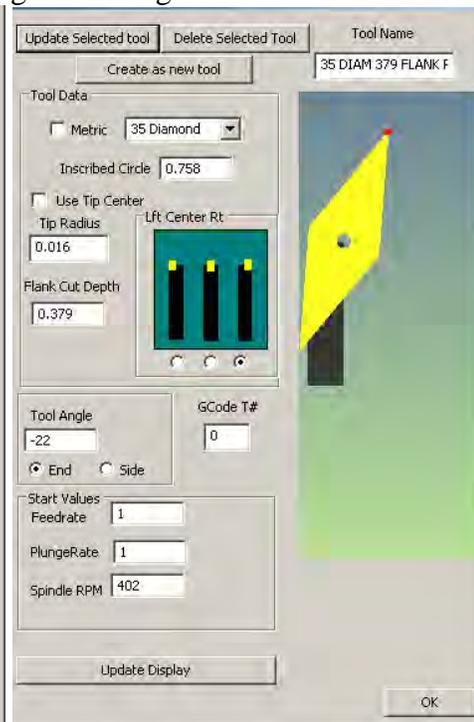


**FIGURE 7.4.1**

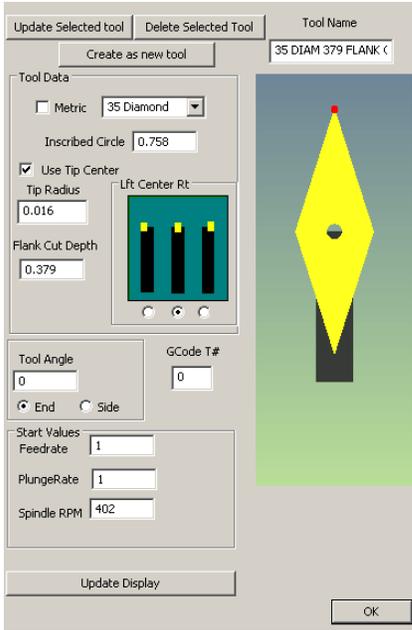
Create three 35 diamond tools, left ( Figure 7.4.2), right ( Figure 7.4.3), center ( Figure 7.4.3) per the following tool dialog screen shots.



**LEFT  
FIGURE 7.4.2**

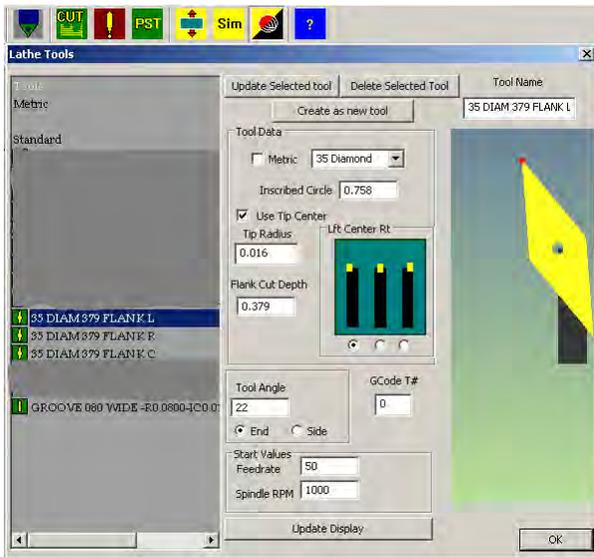


**RIGHT  
FIGURE 7.4.3**



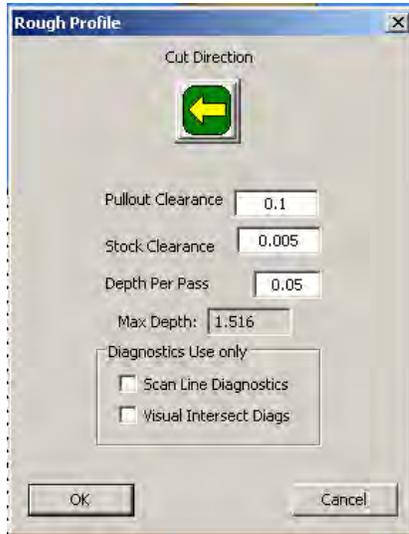
**CENTER  
FIGURE 7.4.4**

The tools will appear in lathe tool table along with the groove tool you created previously.



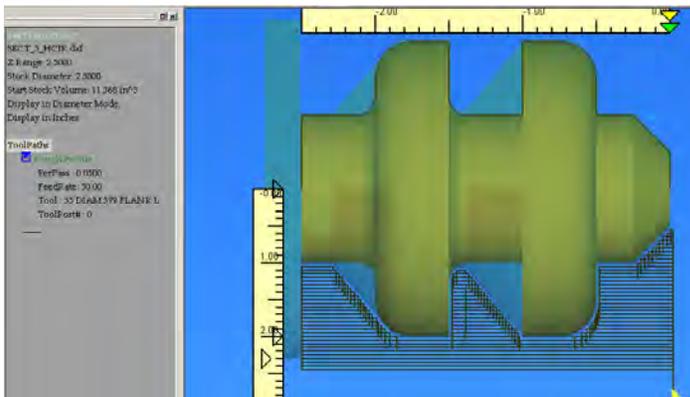
**FIGURE 7.4.5**

The inputs for all three paths will use the same information as shown in Figure 7.4.5 below. Note that a left tool will cut left, a right tool right, and the center tool can cut either way. Clicking the Cut Direction will change the center tool direction.



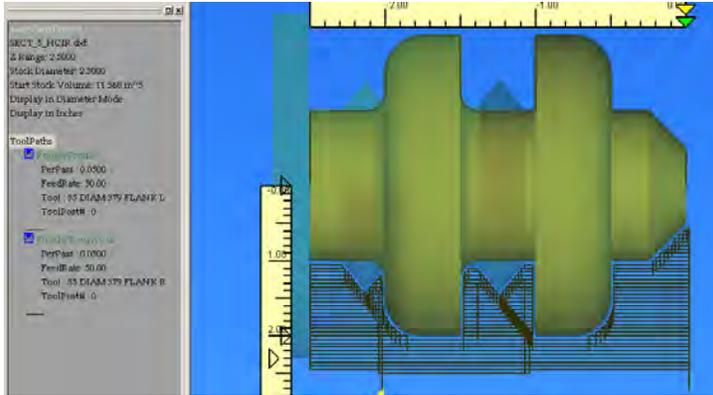
**FIGURE 7.4.5**

First rough paths created using the left tool.



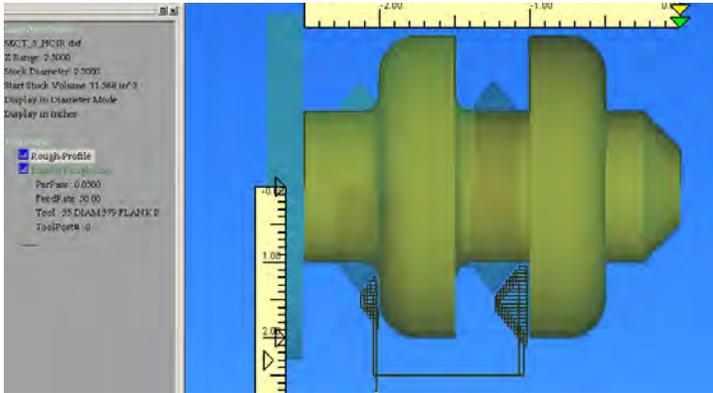
**FIGURE 7.4.6**

Using the right cutting tool the following additional paths are created and added to the displayed pathing.



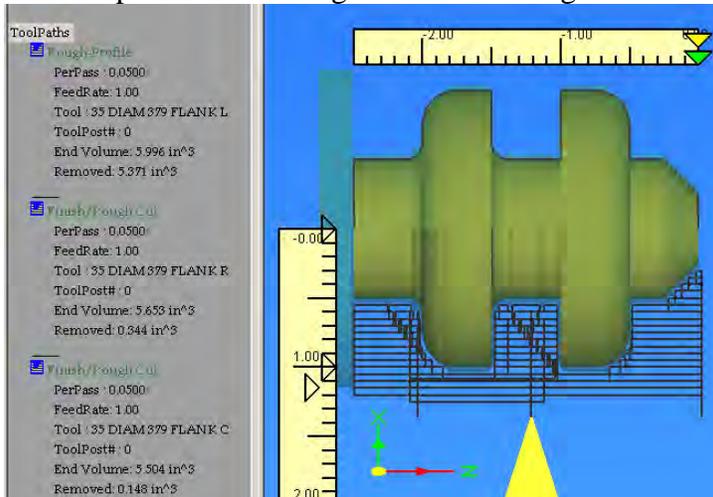
**FIGURE 7.4.7**

If you turn off the first pass, by double clicking on it, then only the second created cuts are shown.



**FIGURE 7.4.8**

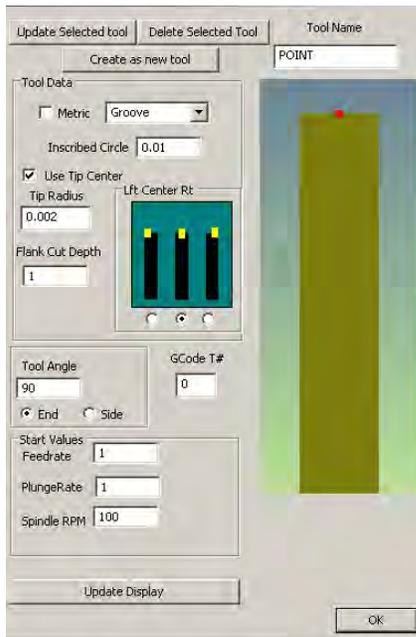
Another pass created using the center cutting tool.



**FIGURE 7.4.9**

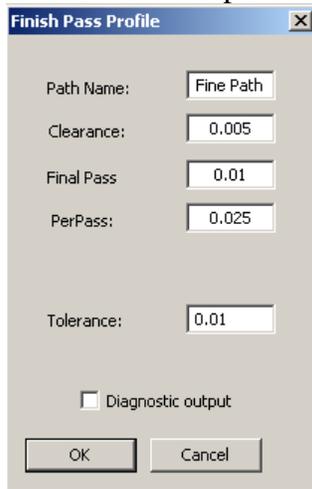
If you zoom in on the piece at this point in the process there is still material to be removed. The starting volume, volume of material removed by each pass, and final volume left can be found in the Project Info. A finish pass is now required to clean up the remaining material. The finish pass we are going to do will use a fictitious tool as it is almost just a point, but although not *usually* practical, it is used for demonstration and discussion of the finish pass.

Create the following point tool. Notice the tip radius is only 0.002" along with a deep flank cutting depth. This will allow the pathing to get very close to the profile. You can do multiple finish passes.



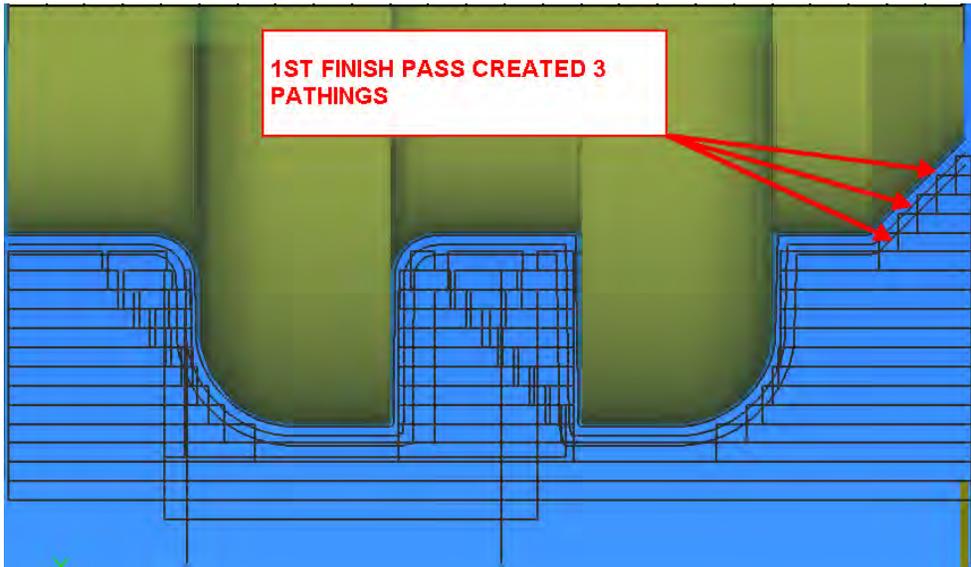
**FIGURE 7.4.10**

For the 1<sup>st</sup> finish pass use the following settings:



**FIGURE 7.4.11**

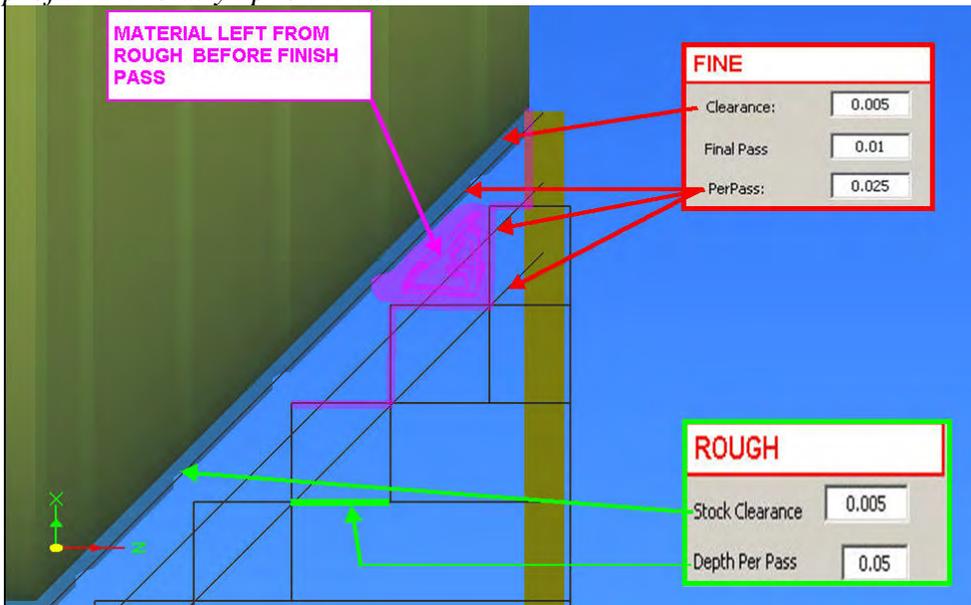
Here is the created pathing.



**FIGURE 7.4.12**

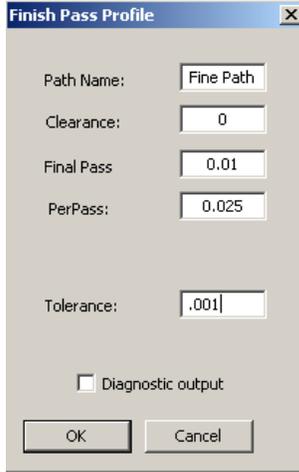
**DISCUSSION:**

*Study the following figure. You can restrict how paths are created by a finish cut. The finish pass inputs were selected such that the finish pass would just remove material left from the rough passes. The finish pathing starts from the outside and removes material to the defined clearance. Material left for the next finish pass will provide for constant volume removal. The choices of how to apply rough, rough / finish, finish to machine a profile are totally up to the user.*



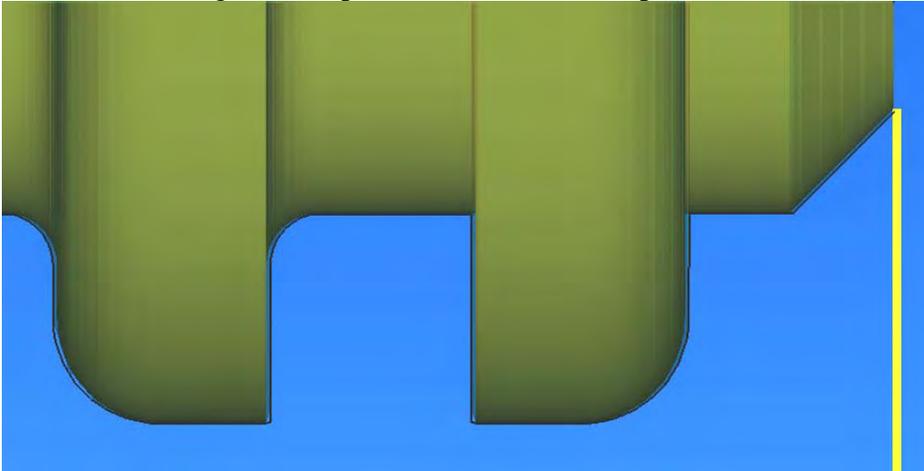
**FIGURE 7.4.13**

A 2<sup>nd</sup> finish pass will now be made to remove the remaining 0.005” material.  
Note that there is no clearance, the depth removes all the remaining material, and the tolerance was increased to more closely follow the profile during path generation



**FIGURE 7.4.14**

The resultant single finish pass which follows the profile is shown below.

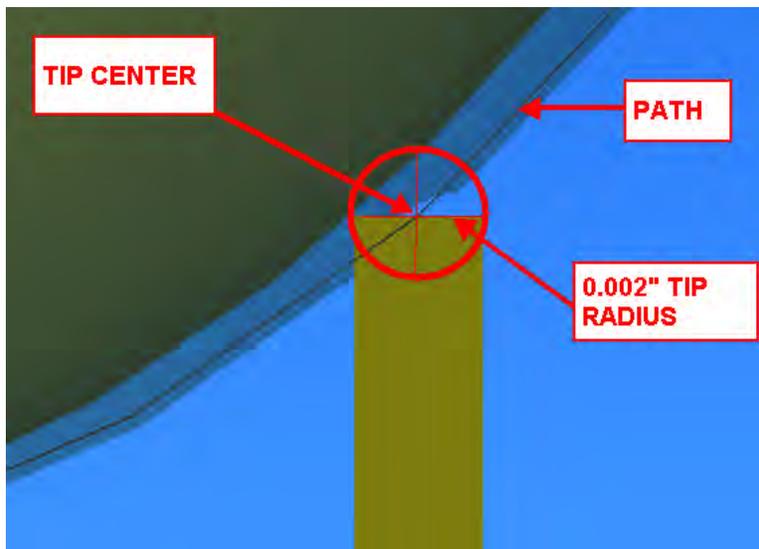


**FIGURE 7.4.15**

## 7.5 DEBUG TOOL

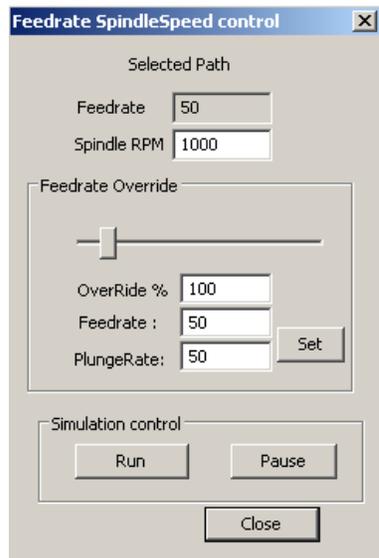


The debug tool can be used at any time to compare the tool to the profile or along a pathing. The current tool will be used and the user then just clicks on the location and it will be placed. The placement is based on the tools tip center and reflects the tool parameters used in it creating it. It can be invaluable when deciding how much to rotate a tool so it will cut the profile and any undercutting will be immediately apparent.



**FIGURE 7.5**

## 7.6 SIMULATION OF THE TOOL PATHS



**FIGURE 7.6**

**Feedrate** –

**Spindle RPM** –

**Feedrate Override** – a slider which will change the feedrate

**OverRide %** -

**Plunge Rate** – X axis move rate

**SET** – click after any changes to the feedrate inputs to be used for the simulation

**Run** – starts the simulation

**Pause** – stops the simulation

**Close** – Closes the dialog box

**NOTE:**

*TO Simulate the cut in real time ( showing true feedrate of the tool ), make sure the paths are expanded in the tree ( collapsing a path de-selects it ),and hit the SIM button, it will show you what the path is calling for and gives a more visual clue about the path to be cut. The speed of the display is the feedrate you commanded in the tool selection, so its a bit of a visual clue if you screwed up the feedrate, I will add a way to change that quickly soon. All expanded paths are simulated one by one when SIM is selected. Rapid speed is loosely defined as 10 times the feedrate, ( just so you don't have to watch all the rapids in slow motion. )*

-BLANK PAGE-

**APPENDIX "A"  
LAZYTURNS QUICK AND DIRTY**

This a quick and dirty step by step procedure to use LazyTurn.

The following is a summary of required steps:

1. Create the profile in CAD and save / export it as a version 12 DXF.
2. Run LazyTurn, and when it opens, open / import the created DXF file.  
The turning object is now shown.
3. Select a tool from the Lathe Tools menu. If none are shown on the left side create one as follows:
  - A- Use the pull down to select the type,
  - B- Define the tool by filling in the information,
  - C- Provide a Tool Name,
  - D- Click the Create as new tool box.,
  - E- Click ok and the display closesReopen the Lathe Tool menu and select the tool.
4. Define a rough profile and click the OK box. The paths are now shown below the profile. Continue doing rough or finish passes as required to accomplish complete machining of the profile.
5. Post the created lathe Gcode by saving it as a text file.
6. Open Mach3Turn and open the saved LazyTurn text file.

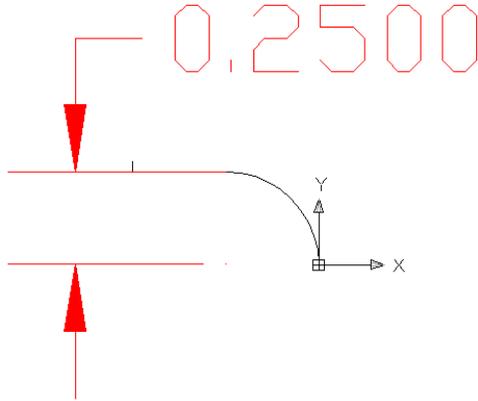
It is highly recommended that you read the manual as detailed information is provided about all the steps.

**STEP 1**

Create the profile in CAD and save / export it as a version 12 DXF.

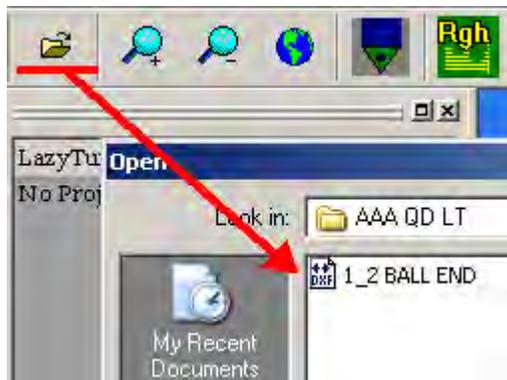
The following shows a drawing for making a ball on the end of a 1/2" diameter piece.

Note that the radius is 0.25".

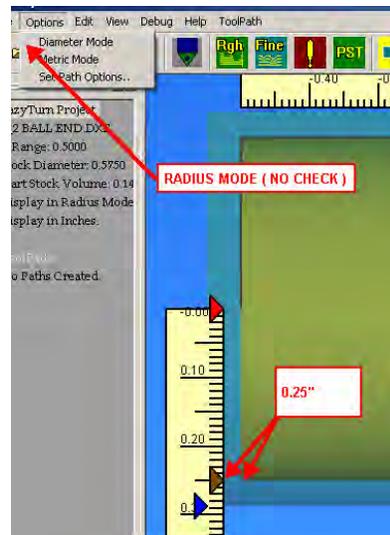


**STEP 2**

2. Run LazyTurn and open the DXF drawing. Note that for this particular example the profile was drawn in inches, exported, imported, and will be machined using radius mode. Do not mix modes and units during the process.

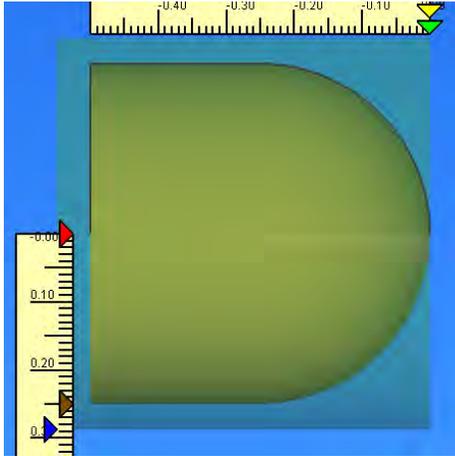


IMPORTING THE DXF



CHECK OF MODE

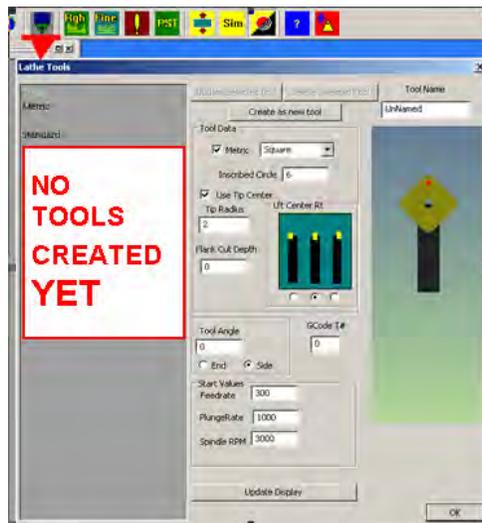
Here is the billet shown after being imported by LazyTurn.



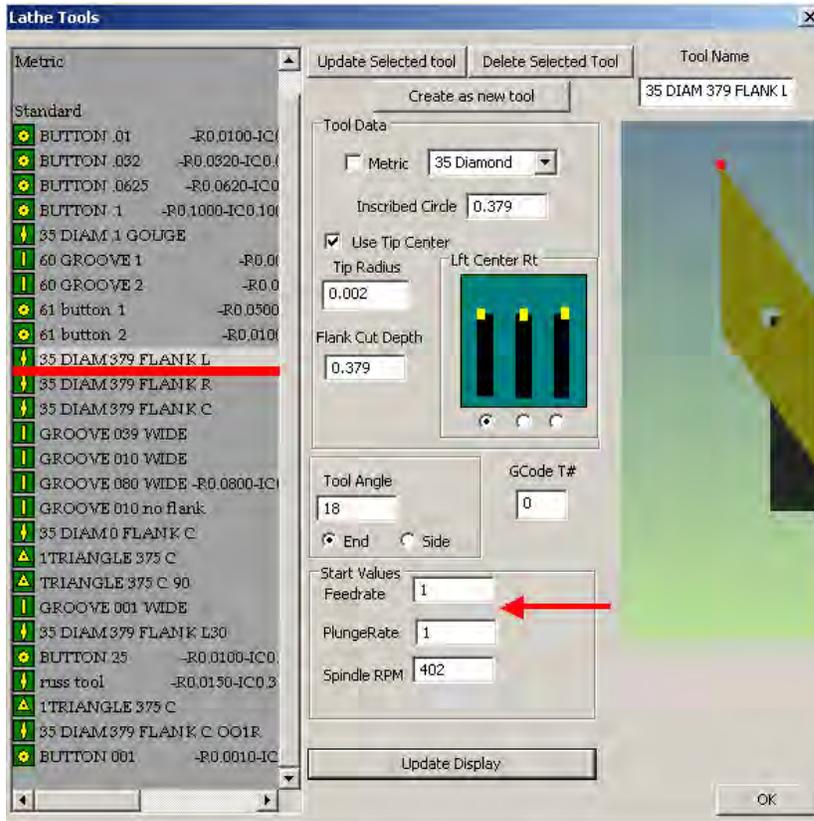
### STEP 3

Select a tool from the Lathe Tools menu. If none are shown on the left side create one as follows:

- a. Use the pull down to select the type,
- b. Define the tool by filling in the information,
- c. Provide a Tool Name,
- d. Click the Create as new tool box.,
- e. Click ok and the display closes



The following 35 diamond tool will be used. Note that the tool is rotated slightly. Careful on your feed and plunge rates verses spindle rpm.



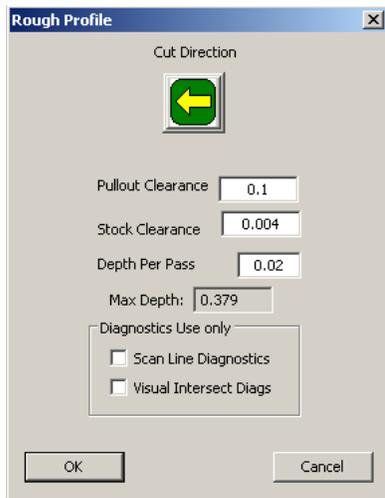
Here is a picture of the actual 35 degree diamond tool aligned to X & Z=0 on the billet.



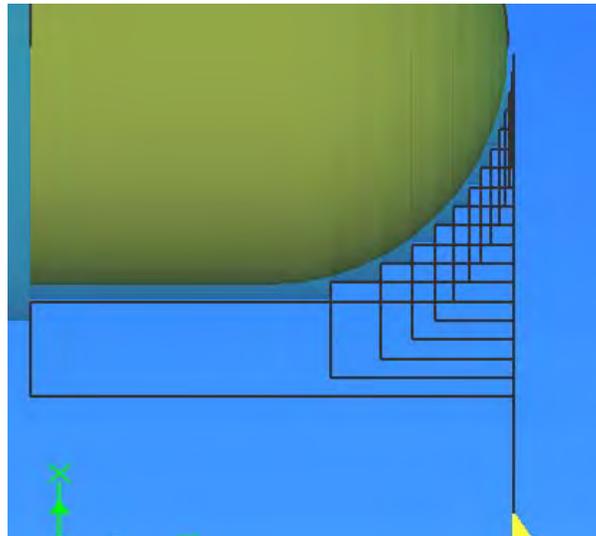
**STEP 4**

Create rough and finish passes as required to completely machine the billet.

A rough pass will be made to quickly remove the bulk of the material. The created pathing is shown to the right.

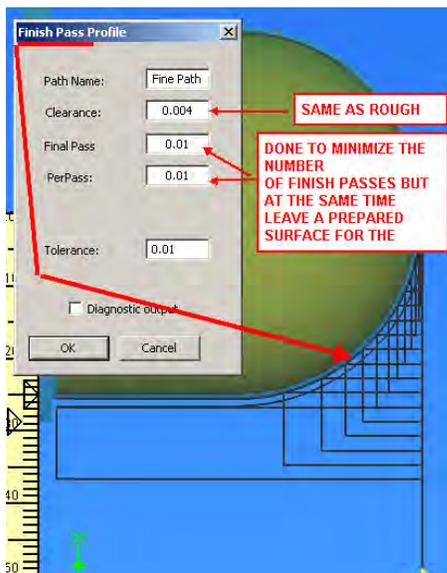


**ROUGH SETTINGS**



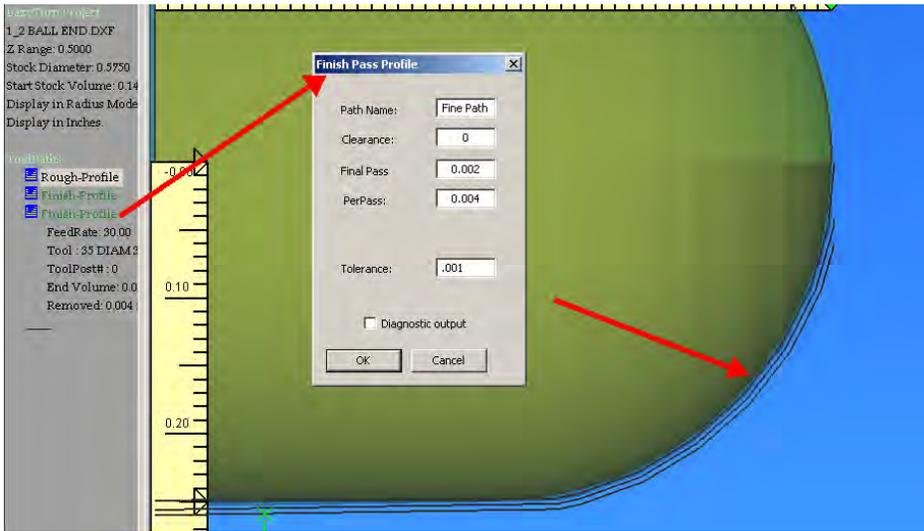
**ROUGH PATHING**

A finish pass will now be done to smooth out the rough notches left around the ball by the rough pass. The passes are selected to minimize the number of finish passes created.

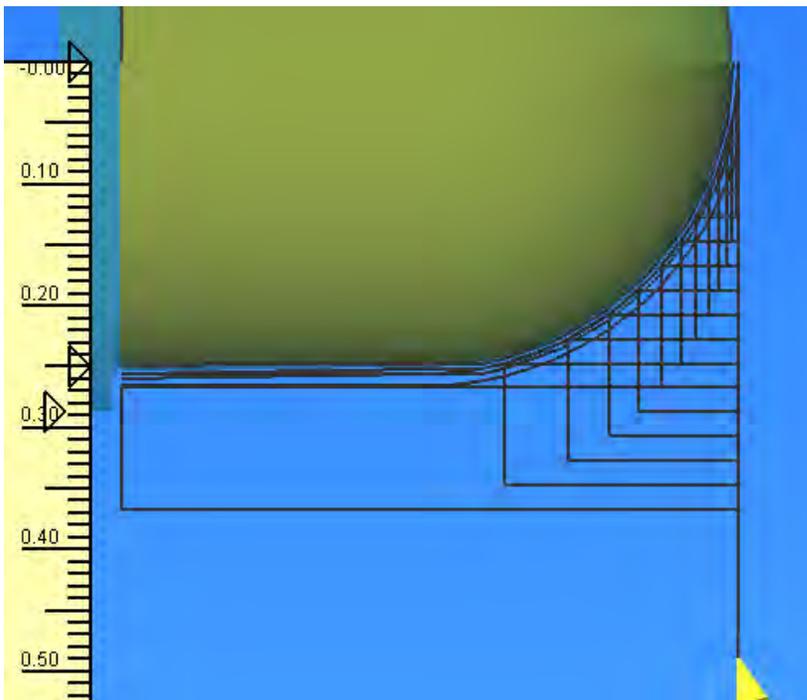


**1<sup>ST</sup> FINISH PASS**

Another finish pass is done cut with finer settings. Note the tolerance increase.  
(The rough and 1<sup>st</sup> finish pass are turned off in the figure below.)



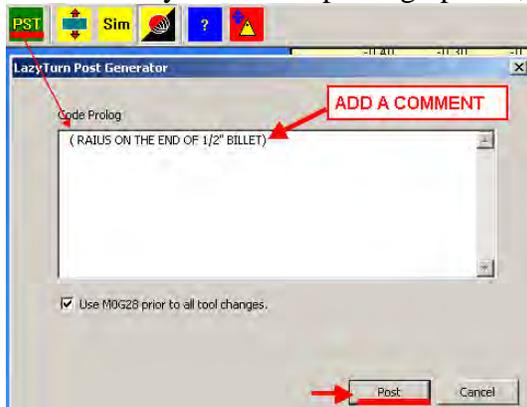
With all the pathing now created the file is posted for cutting in Mach3 turn.  
Here is a picture of all the pathing which will be done.



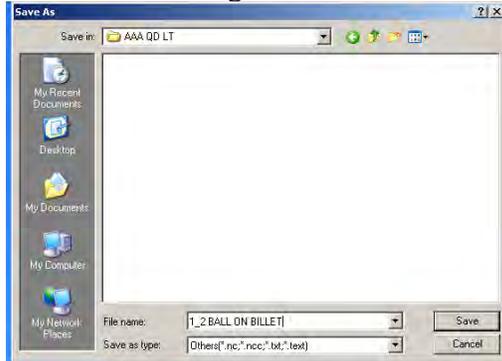
**STEP 5**

Post the created lathe Gcode by saving it as a text file.

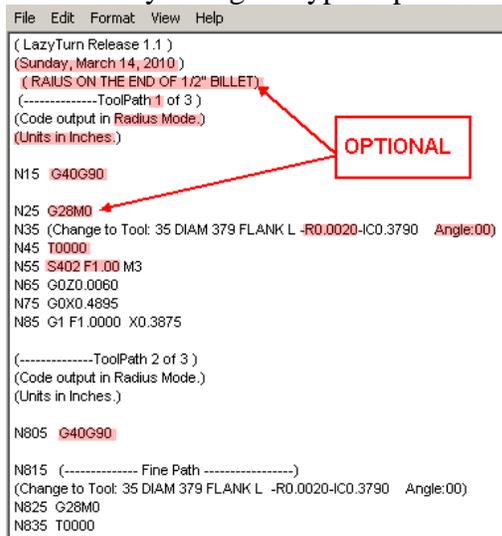
Click the posting button and add comments as you wish. Additionally select / click the boxes for any additional posting options, then and click post.



Save the file along with a name to suite your preference.



The figure below shows the saved code, just click File and save the file if you don't wish to make any changes. Typical posted items are highlighted in red.

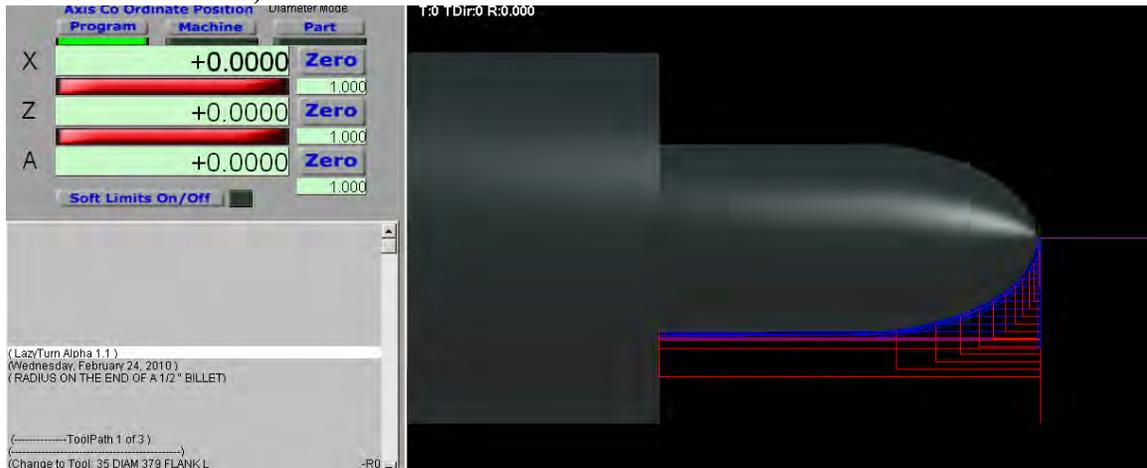


**STEP 6**

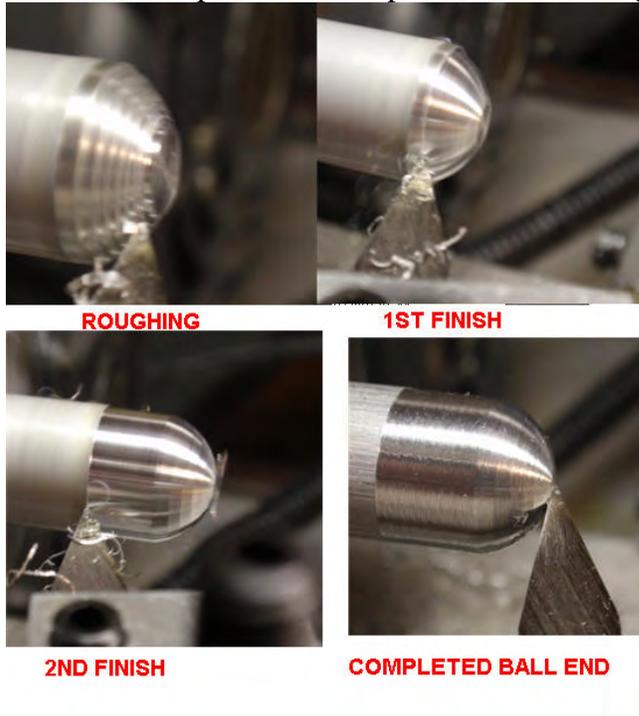
Open Mach3Turn and open Load the file.



Here is the file loaded into Mach3 Turn. ( The screen below is not the standard Mach3Turn screen )

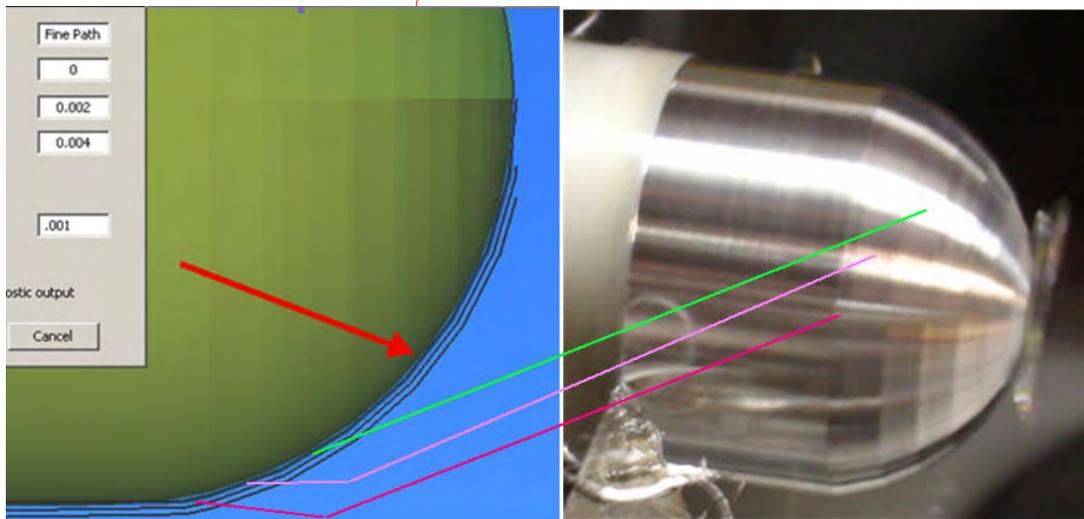
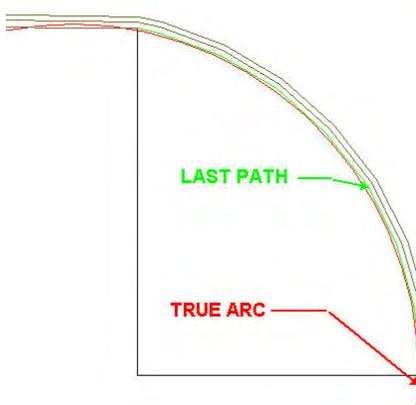


Here are some pictures of the piece as it was being machined.



Just a note on the generated code:

The gcode generated for the finish passes do not contain any arc coding ( G2 / G3 ) as of this writing.. The finish pass is stepped as shown in the figures below and relative to the defined tolerance. In the near future the posted code will be different. It is important to note, that depending on your "Lathe System", your machining results will be different as compared to what is shown bellow.



## 1.0 PREFACE

1.0.1 The intent of this appendix is to provide generic drawing guidance in a way that:

- the basic profile of the object to be turned is understood
- basic program rules are understood

- the drawing imported into LazyTurn will not contain common CAD errors  
So you will create a drawing of the profile which is importable into LazyTurn and the learning curve is shortened .

1.0.2 This appendix is very focused on the intent and recommend the other appendixes be reviewed for PC requirements, DXF file creation, practical profiles that can be machined, etc..

1.0.3 This appendix will not teach you how to use any particular CAD / CAM program. There are probably 100 CAD programs out there with varying complexity in how they accomplish creating entities ( line, arc ), how they joint them, manipulate them, and finally save them. Add to the above CAM / 3D programs and it's overwhelming.

1.0.4

## 2.0 LAZYTURNS PROFILE

2.1 The profile in the drawing must be a connected series of entities.

2.2 One series only.

2.3 Think of it as a drawing showing ONLY the profile.

2.4 The 2D drawing can represent a radius or diameter (Y).

2.5 The drawing is done such that "Y" is vertical and "X" is horizontal.

2.6 It can be in metric or imperial units or even just units,

2.7 LazyTurn converts Y / X to X / Z in the program.

## 3.0 RULES

3.1 The drawn profile must allow for tracing the profile in a single continuous move over its complete length.

So if you were to trace over your profile with a pencil and need to lift your pencil point off the line, to go to some part of the profile or touch something like a line off the profile, then it is not a complete continuous line. Then you may have problems trying to use the file in LazyTurn.

3.1.1 Analogy:

Your driving a paved white line divided road and want to cross the river ahead, but the paved road ends and there is no bridge, you can see another bridge but can't back up, so you take the no white lined unpaved road just off the side and ahead of you, your car goes along but gets stuck in a pot hole. Drive the correct road the first time and don't get stuck. End of story!

3.2 LazyTurn needs to be told what your profile represents on import.

3.3 There are basic differences between a turning, facing, boring profile so for now all writing applies to turning.

3.4 LazyTurn has built in features to reject or fix reasonable CAD errors created by you in creating the profile.

3.5 Draw your profile in the correct quadrant as shown in Figure 3.4.1 below. Draw wherever you want, but move the profile when done.

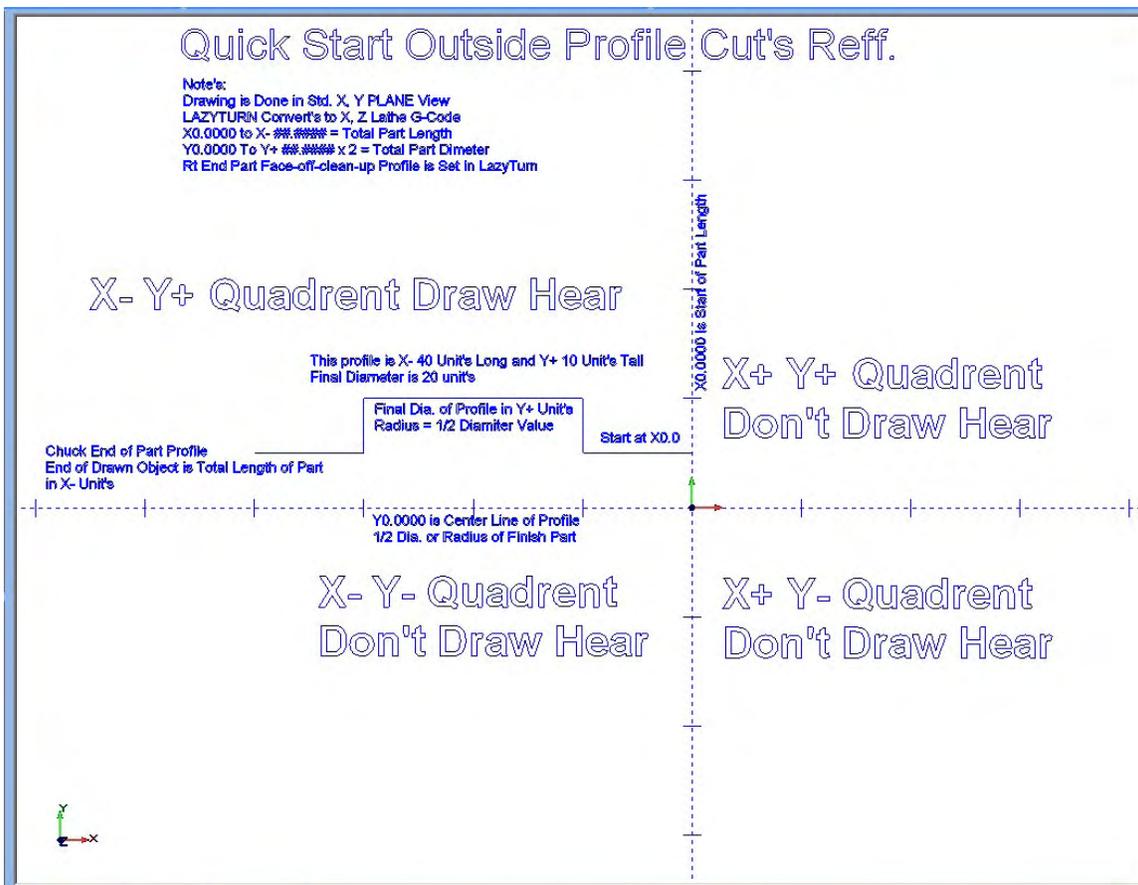


FIGURE 3.5.1

- 3.6 Do not draw more than the top part / upper half of the profile.  
You will get an error message "No appropriate drawing profile was found" if the full profile is drawn.
- 3.7 Do not draw / leave a center line or any line which would represent the the profile center line of the object.
- 3.8 DO NOT ...DO NOT draw, extend any line, any part of the profile, leave any line / dot / pixel / or what ever below the  $Y=0$  which represents the center line. See FIGURE 3.5.1
- 3.9 Do not draw / leave a center line or any line which would represent the the profile center line of the object.
- 3.10 Do not leave a vertical line going up or down at the end of the profile. Although accepted it is not required or recommended.
- 3.11 Do not create multiple drawing entities , one existing under the other, as this will cause an error " self intersection in master file" and no rendered graphics will be displayed.
- 3.12 Do not leave vertical or parallel lines not connected to the profile ( very close or far away from the profile ). Although ignored they should not be in the drawing.
- 3.13 Do not leave a "space" very small or large in or between any element which should be joined and be one continuous part of the profile. Common spaces will usually occur are as follows:
- line to line
  - end of a line crossing the end an adjoining line
  - line to an arc
  - line tangent to an arc
  - arc to arc
  - arc end crossing a matching arc end
- 3.14 Do not leave a line / circle / dot or any element in the area below the profile and the center line /  $Y=0$ .
- 3.15 LazyTurn will reject lines in the lower half of the profile including lines drawn through the profile and if they exist you will get an error message telling you so.
- 3.16 Do not start the profile from a point and progress with the profile in a X positive direction or said differently a dip in the face . This would be considered a facing operation which is not implemented yet.



## 5.0 GENERIC CAD TIPS

- 5.1 Use any tool available in the program that will assist you in avoiding spaces in the profile. Snap, chamfer, fillet, break, trim, extend are just a few by word and may be called something different by individual programs.
- 5.2 Utilize the level or layer ability of the program. Name or create just one for the profile only and keep dimensions or what ever on the others.
- 5.3 Always export only the profile turning off the other layers.
- 5.4 Zoom way in when connecting elements or snapping to a point or line. This will eliminate most space and connection issues associated with the profile.
- 5.5 Use a properties box to compare start and end points of mating elements to assure that they connected.
- 5.6 Connect tangentially to circles and provide smooth transitions by zooming in and confirming the drafting.
- 5.7 When moving a profile to a start point use precision input to define the point (like 0,0,0 ) it is going to.
- 5.8 CAD programs can be very complex and do amazing tasks, yet to generate even a complex profile, only basic commands are utilized. So discipline yourself to use that 5% of the program in an expert way.
- 5.9 Compose the profile from simple elements.
- 5.10 Don't use Polylines
- 5.11 Don't use points, constructions lines, invisible lines on the profile layer.



## 1.0 PREFACE

- 1.1 The intent of this appendix is to provide information about the transition of a drawn profile into LazyTurn.
- 1.2 Lazyturn only requires an importable DXF file preferably Version 12. Only DXF files are supported as of this writing.

## 2.0 DXF HISTORY

- 2.1 DXF stands for Drawing Exchange Format and was created by Autodesk approximately 20 years ago. AutoCad is just one of it's products. DXF is nothing more than a common proprietary language providing a method of exchanging ( import and export ) drawing file information within AutoDesk products. DXF is a generic language which AutoDesk controlled without having to give out their own proprietary format.
- 2.2 The evolution of AutoDesk products have required revision of DXF. Since it was propreitary and not shared with other CAD products in the market place the other products interpreted the DXF to allow file exchange between competing products. Intergraph and later Bentley Microstation were two such products. Information could be exported / imported by these products but at times the conversions left something to be desired because of how the other products interpreted DXF.
- 2.3 The DXF had to address new complexities such as add-on packages, CAM, 3D etc. thus additional information was required to define the drawing via DXF. The DXF changed to address the complexity and here is a listing of versions.

AutoCad Release / DXF Version

12 / 12

13 / 13

14 / 14

2000 / 15

2004 / 16

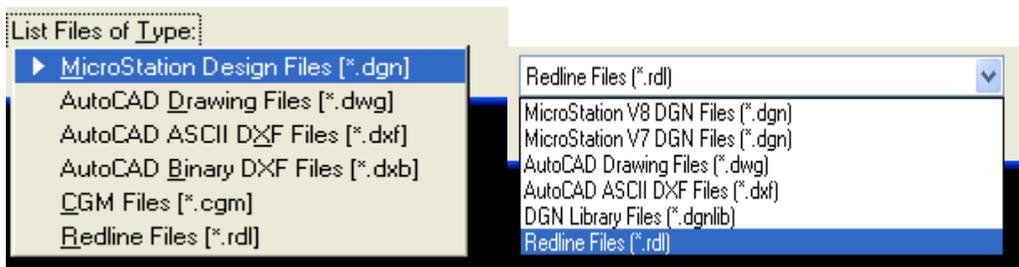
2007 / 17

2008 / 17.1

2009 / 17.2

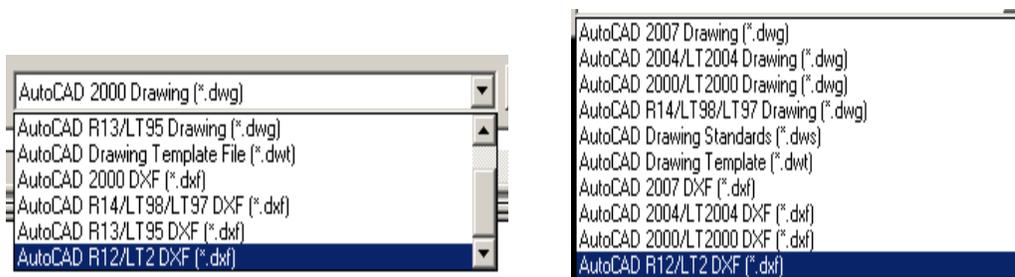
- 2.4 AutoCAD 10 and up support both ASCII and binary forms of DXF. Prior versions support only ASCII versions of DXF. You can read ASCII files by opening the file with a text editor like Notepad since they contain text that can be read. The binary form of DXF files are made up of more than just plain text and have hex characters / bytes that are intended to be interpreted as something other than text. The binary file could more easily represent complex stuff like images, meshes, or whatever. It is more compact, takes less file space and can be read and written more quickly by the computer.
- 2.5 The ASCII and binary DXF files contain a complete description of the AutoCAD drawing. Now if an “1” is interpreted as a “I” when imported or exported to or from another program problems can occur. The exchange has matured over the years but it is still not perfect. Now suppose there are 100 CAD programs out there and each interprets the exported and imported data ever so slightly or differently. Suppose one program doesn’t even allow or recognize what it is receiving? There certainly can be problems as all CAD programs are not created equally.
- 2.6 There are accuracy issues as ASCII dxf files trade-off size and accuracy but binary *dxf* files preserve all of the accuracy.
- 2.7 There are other formats but they will not be discussed.
- 2.8 LazyTurn addresses a lot of issues when importing a drawing of a profile. The program has built in safeguards to avoid problems and confusion. Appendix “B” addresses common issues which a user can eliminate in the beginning and help make LazyTurn’s job of importing easier.
- 2.9 Many CAD programs provide options on what drawing information should be exported / contained in the DXF. It can be a challenge to define how the data should be defined on export in a program. Frankly it should not be required.
- 2.10 Here are just a few options some offer:
- level to layer ( or whatever name it’s called ) changes inclusive of all the associated parameters
  - Entity conversion of all associated parameters
  - Entities only, such that the resulting DXF file contains only the profile section based only the objects you select
- 2.11 Turning off / disabling / freezing the layers / level and keeping entities unrelated to the profile like boarders etc. will provide a more acceptably defined DXF . It just minimizes the interpretation talked about earlier.

- 2.12 Data from conversion to conversion between programs can be carried over into the next DXF thus complicating it.
- 2.13 Convert the file from a 3D to 2D drawing prior to DXF.
- 2.14 Clean, optimize, compress the drawing if provided by the program prior to DXF export.
- 2.15 Exporting / importing between different programs may cause the profile location relative to X=0 Y=0 to change. Always check in your program prior to creating a DXF. Some programs may actually reverse the profile 180 degrees.
- 2.16 The turning profile you provide via DXF for LazyTurn import is very basic in it's elements and only requires fundamental descriptions.
- 2.17 Most applications provide for drawings to be exported out as a Version 12 DXF. Some versions were not as stable as others and new versions continue to this day. Version 13 & 14 seems problematic. Version 12 seems most acceptable even though more recent versions will also work.
- 2.18 Here are some screen shots of AutoCad and MicroStation Options for saving a file. If there is any doubt on the selection, select the one with “ASCII” in the wording. Remember ASCII imports and Binary saves as a drawing to be opened by CAD. You want to save as a AutoCAD ASCII DXF file (\*.dxf) or AutoCAD R/12 (\*.dxf)



MICROSTATION SE &amp; J

MICROSTATION V8XM



AUTOCAD 2000

AUTOCAD 2007

### 3.0 PROGRAM DXF'S

3.1 The following programs provided successful DXF files for LazyTurn.

Turbo Cad

Vcarve

Turbocad V8.1, V9, Dekux 14

TurboCAD Deluxe 14.

BobCad,v18,v21,v22,

Rino v2.0,

CutViewerTurn

STDCAD

D2NC

MICROSTATION – all versions

AutoCad V12 and above

### 3.2 NOTED PROBLEMS & QUESTIONS

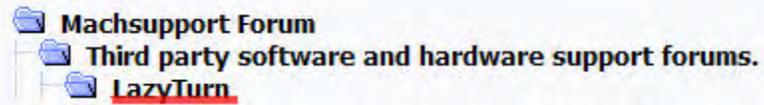
*QCad.*

*Q1- I can not get this to load, it is just lines and arcs, no splines etc. It doesn't seem to care*

*whether the DXF is x,y or x,z. based or where it's drawn in the Cad Quadrant layout  
(adjust to 0,0 after loading)*

*R1- The column.dxf also has those annoying lights in it. When removed, it loads fine. I'm not sure how to sense those automatically, they are stored as lines and arcs, so the program sees them as normal entities.. What I may do is just ignore the softest chains, delete them, and keep the longest chain, that should solve many of these kinds of errors.*

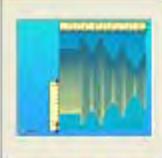
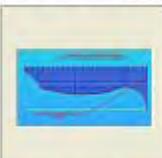
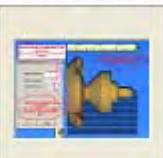
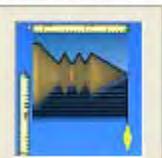
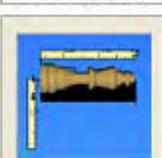
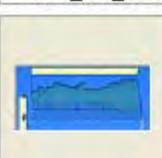
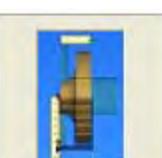
The following is a listing of posted DXF files along with the associated reply# to the LazyTurn thread. The higher the reply # the more recent the reply. A thumb nail image of each file is also provided.



REPLY#	FILE NAME	COMMENTS
13	A-----new_test.dxf_E	
19	A-----new_test_G	
32	King	
35	A-----mach3Cannon1_bigbiglimbo_I	
39	A-----King_Grahm_A_Rev_normal	
47	Column	
47	Pulley	
48	Valve1	
198	richTHREE_BALLS	
203	test_v21_inlet_7	
203	test_v21_inlet_8	
208	DXF12_2000_1B_C4	
208	DXF12_2000_2_C4	
208	DXF12_2000_3_C4	
221	DrawingA4TODO	
226	5_DXF_HCIR	
240	half_elisp_no_Leads	
243	king_grahm_scaled_to_2_50_inch_AA	
244	2d_12DXF_chip_243	
247	8	
258	swivel_socket_d	
287	Cylinder	
321	Flanged_Spindle_INCH	
325	10DXF_HCIR	
353	11_6DXF_HCIR	
366	4_tman_lead_in_2_00	
387	2PROFILE_HANK	
389	connrod_rodend_BIG_Hank_G	
397	20_DXF_HCIR_EGGHEAD_Rich_A	
402	20_DXF_MAN_REV1_VERTICAL	
404	20_DXF_MAN_REV1	
404	20_DXF_MAN_REV1A	
428	Flanged_Spindle_INCH3	
514	Arc.3_D_Good	
514	Arc.3_E_Good	

REPLY#	FILE NAME	COMMENTS
514	Arc.3_F_Good	
555	Allison_Crush_Cyl	
564	Balconspijl-willem	
583	Roller_Test_Master_000_10_units	
602	Copy_of_Roller_Test_Master_000_10_units	
608	40_DXF_02SAME_HCIR	
663	4pwn_dxf	
669	4pwnRC_	
693	CadStdLite	
695	CadStdLiteArcs	
697	BPIN	
704	bpin_new	
722	pulley1	
771	1file_dxf	
777	kandelaar40RC	
789	woodenptxt	
791	GOOD_Copy_of_king_grahm_scaled_to_2_...	
795	01_16_09_Was_GOOD_9_DXF_HCIR_Rich	
805	apin	
820	Schachfigur_Dame	
820	Schachfigur_Dame-4	
857	Connecting_rod	
860	Crosshead_guide_rev1	
878	1x1.5x4_Long_Roller_Test_Master	
893	Flywheel3_D	
898	toolholders	
911	Save_Inside_Inlet_New_Test_L	
935	BPinapin	
938	Test_AA	



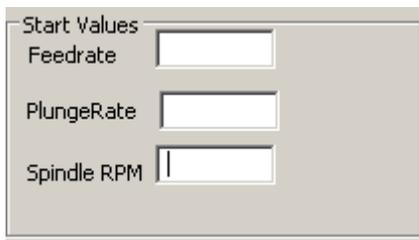
				
47 Pulley	47 Column	469 Chip_131...	428 Path1	404 20_DXF_H...
				
404 20_DXF_H...	402 20_DXF_H...	397 Chip_075...	39 Chip_016_...	389 Chip_061...
				
387 HANK2	366 Chip_036...	353 11_6DXF_...	35 Chip_015_...	325 10C_HCIR
				
321 ScreenHu...	32 LZT_sim_001	287 ScreenHu...	258 Chip_017...	247 8_DXF_HCIR
				
244 vmr_chip_...	243 Chip_008...	240 Chip_029...	226 5_C_HCIR	221 TODO
				
208 LT_SCREE...	203 Chip_012...	203 Chip_011...	19 Chip_012_...	13 Chip_009_...

## 1.0 PREFACE

1.1 This intent of this appendix is to provide general machining considerations as they apply to using LazyTurn.

## 2.0 LATHE SPEEDS - FEEDRATE – CUT DEPTH

For each tool created an associated feedrate, plunge rate, and spindle rpm is specified. The information is contained in the posted Gcode.



The image shows a screenshot of a software interface with a light gray background. At the top left, the text "Start Values" is displayed. Below it are three vertically stacked input fields, each with a label to its left: "Feedrate", "PlungeRate", and "Spindle RPM". Each label is followed by a white rectangular input box with a thin gray border.

The feedrate is in inches per minute. Normally feedrates are expressed in feet per minute measured on the circumference of the work. Note that in Mach3 Turn your axis velocity is defined in inches or mm's per min. The calculation for SFM ( Surface Feet per Minute ) is as follows:

$$\text{SFM} = \text{RPM} \times \text{diameter} / 3.82 \quad \text{RPM} = \text{Revolutions per Minute of Spindle}$$

Diameter = diameter of piece in inches

$$\text{Feedrate in inches per min} = \text{SFM} / 12$$

The chart below can be used find an approx SFM relative to a diameter and rpm. Just divide the SFM numbers by 12.

TABLE OF CUTTING SPEEDS

Diameter of Work Inches	Surface Speed in Feet Per Minute											
	30	40	50	60	70	80	100	120	150	200	300	500
1/16.....	2072	—	—	—	—	—	—	—	—	—	—	—
1/8.....	805	1270	1270	2072	2072	2072	—	—	—	—	—	—
3/16.....	685	805	805	1270	1270	1270	2072	2072	—	—	—	—
1/4.....	418	685	805	805	805	1270	1270	2072	2072	—	—	—
5/16.....	266	418	500	685	685	805	805	1270	1270	2072	—	—
3/8.....	266	266	418	418	500	685	685	805	1270	1270	2072	—
7/16.....	164	266	266	418	418	500	685	685	805	1270	2072	—
1/2.....	164	164	266	266	418	418	500	685	805	1270	1270	2072
1.....	112	164	164	266	266	266	418	418	500	805	1270	2072
1 1/4.....	83	112	164	164	164	266	266	418	418	685	805	1270
1 1/2.....	70	112	112	164	164	164	266	266	418	500	805	1270
1 3/4.....	70	83	112	112	164	164	164	266	266	418	685	1270
2.....	45	70	83	112	112	164	164	266	266	418	500	805
2 1/2.....	45	70	70	83	112	112	164	164	266	266	418	805
3.....	45	45	70	70	83	112	112	164	164	266	418	685
3 1/2.....	28	45	45	70	70	83	112	112	164	164	266	500
4.....	28	45	45	45	70	70	83	112	164	164	266	500
5.....	28	28	45	45	45	70	70	83	112	164	266	418
6.....	28	28	28	45	45	45	70	70	83	112	164	266
7.....	28	28	28	28	45	45	45	70	83	112	164	266
8.....	28	28	28	28	28	45	45	45	70	83	164	266
9.....	28	28	28	28	28	28	45	45	70	83	112	164
10.....	28	28	28	28	28	28	45	45	45	70	112	164

Example: For a 1/2 " diameter stock at 400 rpm the SFM is 50/12= 4 for a tool feedrate input.

Cutting of metal on the lathe is not limited to just feedrate and spindle speed. Depth of cut for a rough and finish pass is also related to the machinability of the material. Additionally, what is a proper depth of cut for one lathe may not be practical for your lathe as it takes horsepower to keep the spindle rotating as the cut is done. The axis motor ability also comes into the equation along with rigidity of the machine.

As the material being cut gets softer the cutting speed can usually be increased. Likewise as the cutting tool material gets harder the cutting speed can also be increased. Use of a cutting fluid and special coatings on the cutting tool also affect the cutting speed and depth.

Below is a table which shows the machinability of different materials:

MACHINABILITY OF MATERIALS

Carbon steels:	Index Percent Relative to 1212 Carbon Steel		
1212	100%		
1015	72%		
1018	78%		
1020	72%		
1022	78%		
1030	70%		
1040	64%		
1042	64%		
1050	54%		
1095	42%		
1117	91%		
1137	72%		
1141	70%		
1141 annealed	81%		
1144	76%		
1213	136%		
12L14	170%		
1215	136%		
		Alloy steels:	Index Percent Relative to 1212 Carbon Steel
		2355 annealed	70%
		4130 annealed	72%
		4140 annealed	66%
		4142 annealed	66%
		41L42 annealed	77%
		4150 annealed	60%
		4340 annealed	57%
		4620	66%
		4820 annealed	49%
		52100 annealed	40%
		6150 annealed	60%
		8620	66%
		86L20	77%
		9310 annealed	51%

Stainless Steels and Super Alloys:	Index Percent Relative to 1212 Carbon Steel		
302 annealed	45%		
303 annealed	78%		
304 annealed	45%		
316 annealed	45%		
321 annealed	36%		
347 annealed	36%		
410 annealed	54%		
416 annealed	110%		
420 annealed	45%		
430 annealed	54%		
431 annealed	45%		
440A	45%		
15-5PH	48%		
17-4PH	48%		
A286 aged	33%		
Hastelloy X	19%		
		Gray Cast Iron:	Index Percent Relative to 1212 Carbon Steel
		ASTM class 20 annealed	73%
		ASTM class 25	55%
		ASTM class 30	48%
		ASTM class 35	48%
		ASTM class 40	48%
		ASTM class 45	36%
		ASTM class 50	36%
		Aluminum and Magnesium Alloys:	Index Percent Relative to 1212 Carbon Steel
		aluminum, cold drawn	360%
		aluminum, cast	450%
		aluminum, die cast	76%
		magnesium, cold drawn	480%
		magnesium, cast	480%

The depth of cut should vary with the different materials. The rough cut should be within your lathes capability and appropriate for how the material is being held in the lathe. Roughs are just meant to remove as much material as possible in the shortest period of time. A rough-finish may be done to prepare for a finish cut which will be of small depth. A prepared finish, when cut, with a finish pass, will avoid tool chatter and springing of the material providing for more accurate end results.

All of the above become important when inputting values for the depth of cut and number of passes for the finish and roughs.

### 3.0 TOOL COMPENSATION

LazyTurn provides Gcode which is pre compensated based on the tool tip radius and tool shape. The created tool should reflect the tool tip radius as accurately as possible along with a flank cut depth cut that reasonably resembles the standard tools provided. There should be no need to use G40 / G41 / G42 along with using tool tip radius compensation in Mach3 Turn. ( REF: Section 9 & 10.7.11 in the Using Mach3Turn manual rev 1.84 ).

LazyTurn provides collision detection when generating the gcode, thus using Mach3 Turn compensation, would void the collision protection which was provided and the odds are very high that the user would not implement the required coding changes properly.

### 4.0 TOOL SELECTION

- 4.1 There are books written about lathe tooling. Inserts and holder dimensions vary between manufactures even if the descriptions are similar. Specialty shapes, nose angles, inscribed circular tolerances, relief angles, corner / tip radius, and cutting edges are just a few parameters which define how the insert along with the holder could clash with the profile. Take into consideration hand or custom ground tools and the variance in how the tool insert shape relates to clashing with an object is overwhelming.
- 4.2 Different lathe tools are primarily used for specific cutting actions. LasyTurn currently doesn't restrict what tool you choose to do a particular operation. The same tool can be used for rough through finishing operations and cut a complete profile.
- 4.3 Common sense regarding selection can be subjective depending on the user. The program only knows what the user tells the software. Currently there are no built in safeguards in tool selection and frankly provides for unrestricted use of the software which in the end provides for flexibility.
- 4.4 It is up to the user to select or define a tool to do the operation. CNC only implements the cutting more accurately and timely as compared to doing it manually. The user needs to think in terms of clashing or following the profile with another object and apply common sense.

The tool you select or create must have a form which will allow the tool to follow the profile. If the tool can't follow the profile, then only restricted pathing will be generated. (ie: expect a .2" tip radius to cut a square corner or a lesser radius)

**1.0 PREFACE**

This intent of this appendix is to assist in getting a profile turned on a lathe using LazyTurn in combination with Mach3Turn software. It is not intended to teach you how to operate your lathe or provide for a detailed explanation of using Mach3Turn.

**2.0 MACH3 TURN OVERVIEW**

The following is just a quick overview of Mach3 Turn and it is suggested that the reader consult the Using Mach3 Turn Manual. It is assumed that you don't have any homing or limit switches, that a front tool post in the carriage is located away from the work piece towards the tailstock on startup of Mach3 Turn, and tool changes will be done manually. There are numerous ways to setup and run a lathe. I have chosen the above since that is rather the norm for many novice users.

**2.1 LATHE DRO DISPLAY AND ZERO POINTS**

The lathe has two axes, namely X and Z for this discussion, and Mach displays three different coordinate positions as shown below. They are Program, Machine and Part and a general understanding of each is important along with their respective zero points.

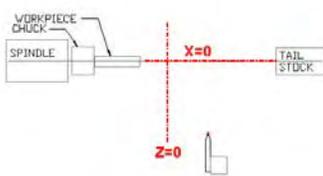


FIGURE 2.1.0

Axis Co Ordinate Position		Diameter Mode
	Program	Machine
X	+0.0000	Zero
Z	+0.0000	Zero
		1.000
		1.000

FIGURE 2.1.1

Program zero point – where X and Z=0 as it relates to the CAD coordinate system used in locating the profile in a DXF file. X and Y are converted to X & Z on import into LazyTurn. X=0 is considered the lathe and profile center line. Z=0 is relative to the end of the stock since the profile can be negatively offset from Z =0. The profile end is normally at X and Z=0.

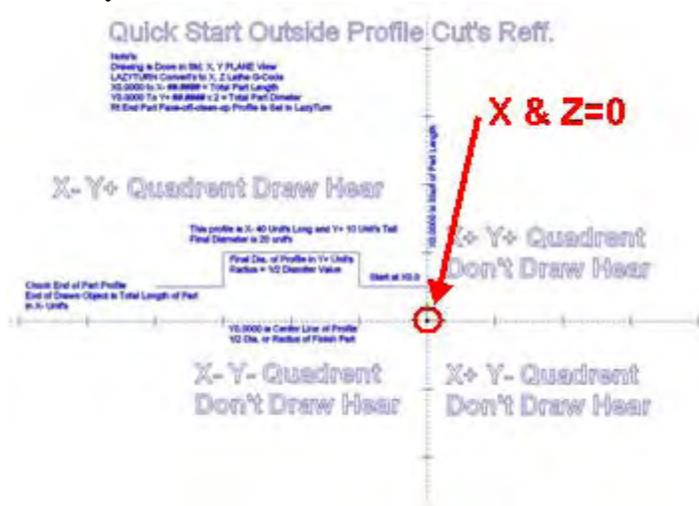


FIGURE 2.1.2

Machine zero point – where the tool ( controlled point ) is located within the possible travel limits along the lathe bed. X & Z =0 can be anywhere.

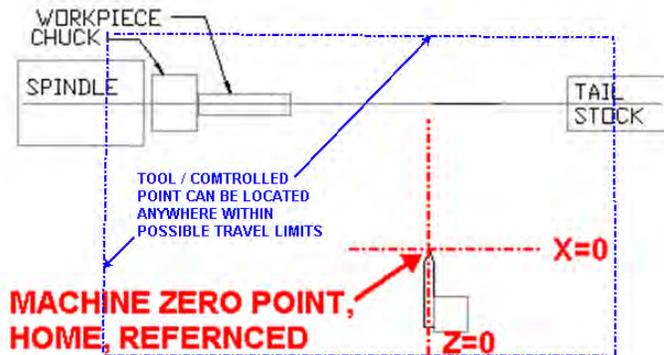


FIGURE 2.1.3

Part zero point – where the part or workpiece is located relative to the lathe axis. X =0 by default but Z can be anywhere along the lathe bed.

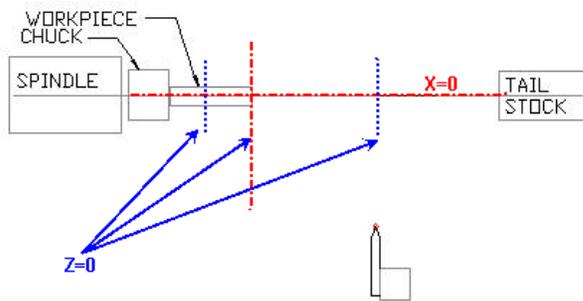


FIGURE 2.1.4

Program, machine, and part zero points can generally be summed up as follows:  
 Program zero is normally at the end of the profile, machine zero can be anywhere but normally it is located away from the workpiece, and the part zero is normally the end of the stock. The controller can correlate the different zero points and adjust the machines axis movement when setup appropriately.

In general, machine coordinates are used to define how the machine is set up to the cnc controller, part coordinates are used to define where the part is to the cnc controller, and the program coordinates associate additional program coding to the cnc controller. The cnc controller is the Mach3 Turn Software.

### **2.1.2 PROGRAM COORDINATES**

The “Program” contains all the Gcode for machining. Thus the LazyTurn program will provide all the axis moves necessary to machine the part. In general, there will be an initial x and z move, then all the machining moves, and a final axis move to some point. The Program Coordinates “DRO” shows where the tool is relative to the machine coordinates. The program coordinates are based on a coordinate system and in many cases there will be no difference between the program and part coordinates since initial setup will correlate the two via a work offset ( G54). Usually only one offset is required during setup and after that additional offsets may be used when multiple pieces using the same gcode will be done. It is suggested that the user review the Gcode section of this appendix on offsets.

### **2.1.3 MACHINE COORDINATES**

Mach3 Turn keeps track of axis movements from a user defined point. You need to define that point to Mach since it has no idea where the carriage or tool is relative to the axes of the lathe. That point can be anywhere, but using common sense, that point should be within the available axes travel. Once that point is defined via actual axis movement ( you can’t change machine coordinates directly via DRO input ), any axis move away from the point will be shown in the machine coordinates DRO. They will be absolute values and always show the X distance in terms of a radius even though you may be in diameter mode. X axis moves away from the lathe axis are plus values and Z axis moves towards the tailstock are also plus values for this overview. Simply put, Machine Coordinates are Mach3 Turns way of “absolutely” knowing where your tool is along the lathe bed and are used for defining where the tool is relative to the travel limits.

### **2.1.4 PARTCOORDINATES**

The software needs to know where the part is relative to where the tools are currently located. Part is the piece in the headstock to be machined as discussed here. The part can be located anywhere along the z axis. The part has coordinates associated with it and normally the center of the part is on the lathe x=0 axis. The software can’t control the machining of the part if it doesn’t know where it is. SO part coordinates are used to relate or define where the part is relative to the machine coordinates.

### **2.15 COORDINATE SUMMARY**

There are three different coordinate systems, each related to a specific zero point, but all are related to each other “after” setup. When alternately selecting one of the DRO’s the controller will show the differences ( if there are any ) or relationship between them. Figure 2.15 shows three different zero points as they relate to their associated coordinates prior to setting up. Setup involves defining to the controller how they are related which are done via offsets.

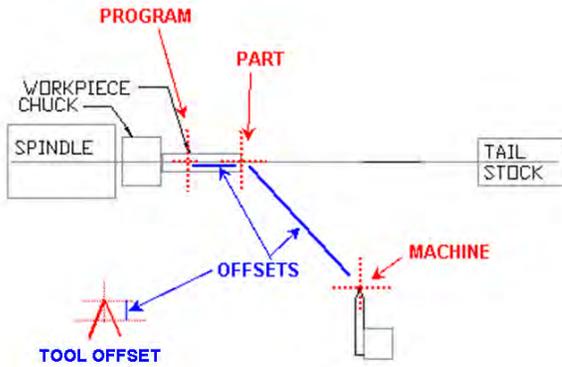


FIGURE 2.15

**3.0 ILLUSTRATED EXAMPLE OF THE AXIS COORDINATES**

Figure 3.0 below shows the tool in a holder located a distance away from the work piece. Clicking the Zero World X & Z buttons will delete any stored work offsets and resets the the X & Z dro’s to zero. There are six basic work offsets ( G54 TO G59 ) available but usually only one of them will be used Additionally clicking the Set Home X & Z buttons reaffirms that you are home and that your home off settings in limits is X and Z=0. Mach now interprets that the tool ( controlled point ) is located at X & Z=0 as displayed in the Machine coordinate DRO’s. Work offsets are nothing more than how far the workpiece is from the tool. Mach will keep track of the moves and at any time you can just click the Home All button and the axes will move back to the home position of x & z =0.

G-Code Pos	X	Y	Z	A	B	C	Name
G54	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	G54
G55	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	

FIGURE 3.0

If the axes are moved such that the tool ( controlled point ) was placed on the center of the lathe axis and at the face of the stock, the Machine DRO’s will display the position away from home. If the user clicks the X and Z zero, this will define to Mach where the work piece is relative to home. If you click Part or Program buttons the DRO’ s are 0.0000 but relative to home in terms of Machine coordinates, which are absolute distances from a defined point on the lathe (ie; home ), they are -1.0000” in the example. The measured distance is from machine zero / home position to the part / program zero which is negative. Mach stores this information by creating a Work Offset from the home position as shown in Figure 3.1.

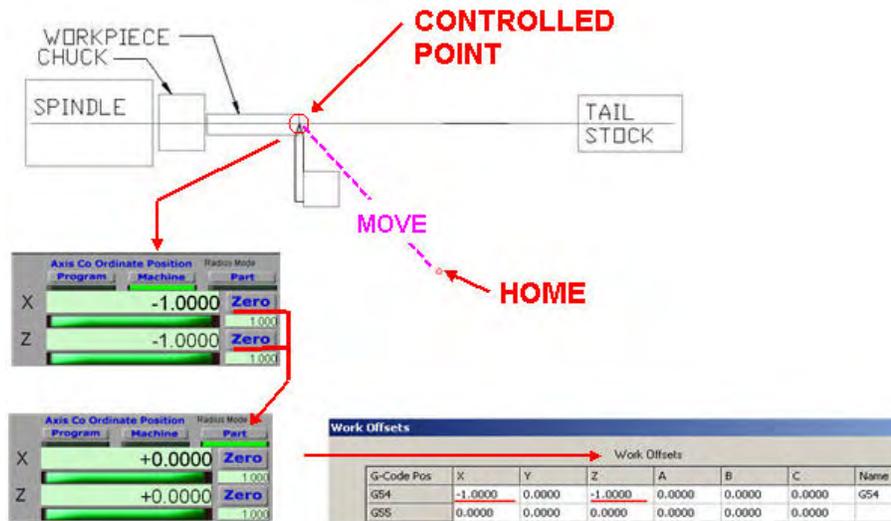


FIGURE 3.1

At this point of writing, you have defined to Mach where the starting point of the tool is and also defined to Mach how far away the work piece or part is from the tools starting point via an offset.

#### 4.0 TOOL TABLE SETUP FOR THE LATHE

This is quick overview of how to setup up a tool table in Mach 3 Turn if you use a quick change tool post.

Mach uses a tool table to store tool information. The table is just a listing of offsets. The controller can then apply the different offsets of a selected tool when controlling axis movements. Tool #1 can be considered a master tool and all the other tools are related to it. This is accomplished by defining how the other tools are dimensionally offset from the master tool. There are two ways to fill in the tool table, namely, measure the differences with a device like a tool setter and just fill in the table, or, use the Mach Tool Table Screen to assist in getting the measurements and filling in the data.

#### 4.1 USING MACH TOOL TABLE SCREEN

Mach needs to know where the tool tip, more precisely the point where it will just start cutting material, is located for each of the tools placed in the tool post. It will be assumed that the tool (includes the holder) can be inserted, removed, and reinserted into the post and the tip location will be retained. Let's assume you have three individual tools in holders, namely tool #1, #2, and #3. Each tool, when placed into the post, will be located a different distance from a fixed / reference point. Mach will associate the distance differences of the other tool tips by relating those different distances to the master tool and populate the table accordingly.

#### 4.1.1 REFERENCE POINT / LINES

A reference is nothing more than a line parallel to the lathe X axis and a line representing the face of a fictitious part or location along the Z axis. The reference lines are used as a fixed location from which you measure to set your tool table up. A master tool is used to define where the reference lines are so the other tools can be related to it. The most accurate way of setting up a reference would be to turn a piece of stock to a known outside diameter and face the stock, thus, the outside of the piece would be parallel to the lathes X axis and the face ( Z reference ) would be perpendicular to the X axis.

#### 4.1.2 SETTING THE MASTER TOOL

For discussion purposes, instead of turning down a piece, a ground rod of known diameter with a ground perpendicular face is mounted in the chuck. The master tool is a piece of ½ round carbide held in tool holder #1. It is a good idea to mark your tool holders. Both are shown in the Figure 4.1.2. The master tool height was set to the lathe center.

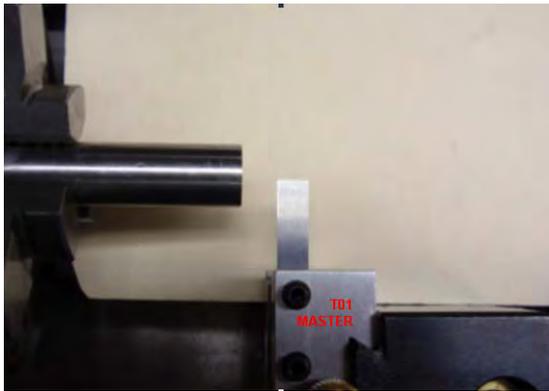


FIGURE 4.1.2

The master tool is aligned such that it is flush with the face of the reference, thus the tool holder and post are now perpendicular to the X axis and the tool edge is at Z=0 reference in Figure 4.1.3. The tool is then aligned to the OD of the reference, thus the master tool is at a known reference location X =0 as shown in Figure 4.1.4.

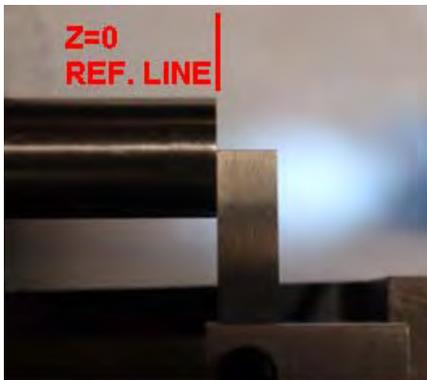


FIGURE 4.1.3

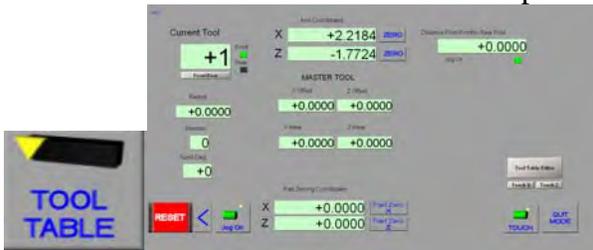


FIGURE 4.1.4

Here are the steps using the screens:

The tool is brought into contact with the face of the reference ( touched off ). You click Zero Z button. The tool is brought into contact on the OD of the reference. You click Zero X button.Mach machine coordinates display X and Z =0. Any movement away the reference will be shown in the DRO’s. So..... if you move away from the reference, remove tool #1 and then insert the next tool, tool#2, align to the face reference ( Z=0 ), the Z DRO will display the dimensional difference or offset of tool #1 to tool #2. Like wise for the X axis if you align to outside diameter.

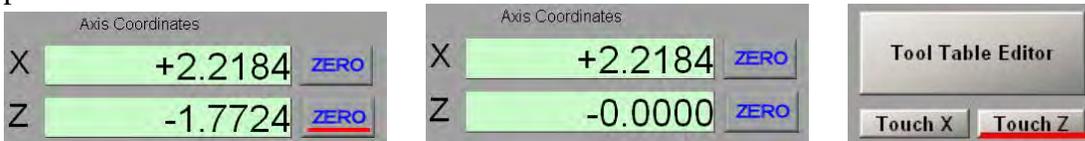
1.Select Tool Table when Mach3 Turn opens which takes you to the tool table screen.



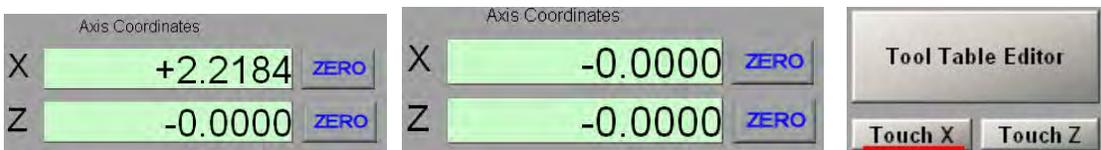
2. Input 1 as current tool and define it as Front by clicking the button below the input.  
( 1 will be the master tool )



3. Move the Z axis to align the tool to the face of your reference and once aligned click the Z Zero button and the Z DRO will reset to 0. Then click the Touch Z button which puts the current Z DRO value into the tool table.

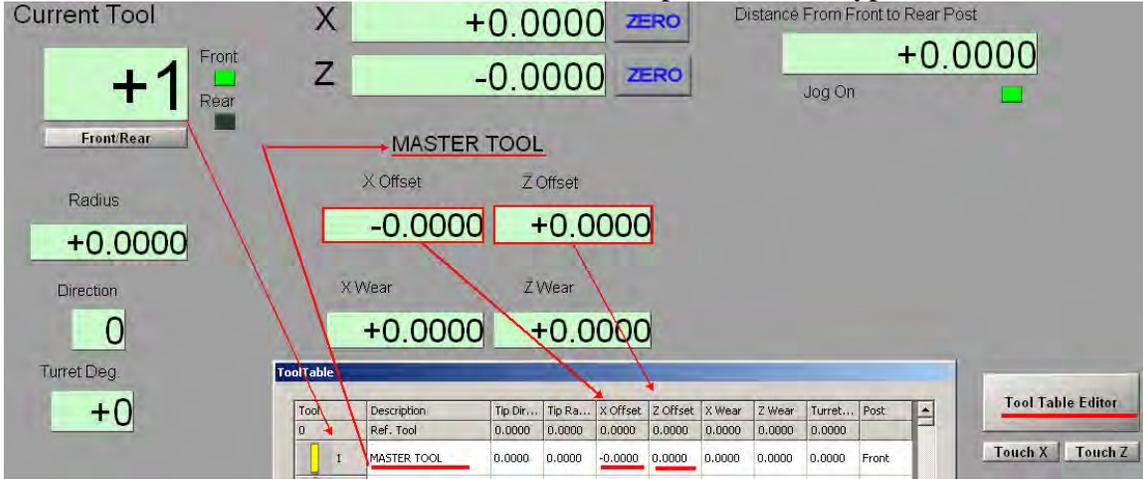


4.Move the X axis to align the tool to the outside surface of you reference and once aligned click the X Zero button and the X DRO will now reset to 0. Then click the Touch X button which puts the current X DRO value into the tool table.



Clicking the x & z Zero button is only required once so you can use the DRO to display the differences.

5. Click the tool table editor to open the tool table. Note that there is no X or Z offset in the tool table for tool #1. Click in the ref tool description box and type MASTER TOOL.



6. Close the tool table flyout. The reference point has coordinates of X & Z = 0. You now move the axes far enough so you can change tools. Move tool #2 so you can align it to the same reference used for tool #1. **DO NOT CLICK ZERO BUTTON FOR ANY AXIS**

### 4.1.3 SETTING THE OTHER TOOLS

1. Input 2 as current tool and define it as Front by clicking the button below the input.



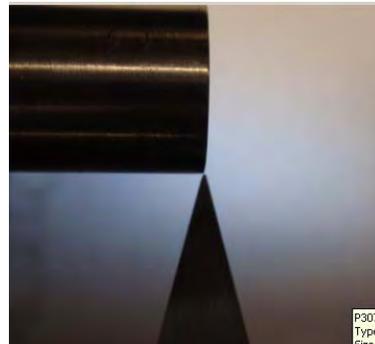
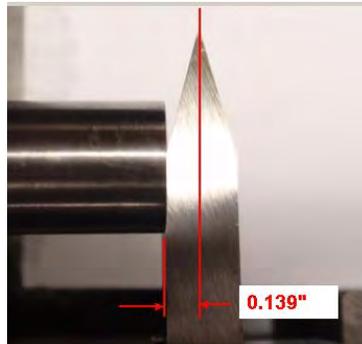
2. Move tool #2 to align it to the outside diameter of the reference ( same as in figure 3 but now it's tool #2 ). You will notice that the X DRO is no longer 0.0000. (**DO NOT CLICK the X ZERO button** ) Mach has kept track of the location tool #2 is at, and upon clicking Touch X, Mach automatically calculates the offset from T01 and puts the offset into the X offset in the tool table for tool #2. The X axis DRO is reset to zero.



If you open the tool table you will see the X offset for tool #2 and the value shown is absolute based on radius mode.

Tool	Description	Tip Dir...	Tip Ra...	X Offset	Z Offset
0	Ref. Tool	0.0000	0.0000	0.0000	0.0000
1	MASTER TOOL	0.0000	0.0000	0.0000	0.0000
2	30 DEG 010 R 278 W	0.0000	0.0000	0.5488	0.0000

3. Now move the tool to align it to the Z axis reference point. Once aligned you again just click the Touch Z button. Note that you need to align the tool such that the center of the tool tip is aligned with the z reference and not the side of the tool as was done with the master tool. Example, you can align the side of the diamond pointed tool as shown in figure 1, measure the width and divide by 2 to get the center ( if the ground tool is truly symmetrical / ground equally from each side ) and then make a step move to center it on the face as shown in figure 2. Offsets of all the tools from the master tool is only as good as the care put into alignment of the tool and is also influenced by the accuracy of your lathe's system.



4. Duplicate the setting procedure done for tool # 2 for all the remaining tools.

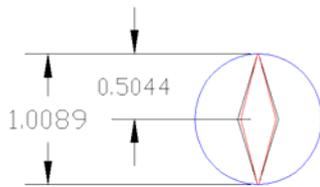
#### Summary and comments:

All of the above accomplished defining the offsets of all of your tools as they relate to the master tool. When a tool is called out in a program, Mach will automatically utilize the offsets to control the axis movements as they relate to the program Gcode. Where you define the reference, tool used for the master, alignment method used, etc are user choices to be made. The Using Mach3 Turn Manual defines the other tool parameters you will see in the tool table. You only need be concerned with them if using a turret and your post is rear mounted. The wear offsets, tip radius, and tip direction are not used because LazyTurn pre-compensates the code. Do not have a mixture of front and rear tool posts, one or the other, but not both.

***NOTE: All the tool offsets are relative to a single tool but all the tools are also related to each other.***

#### 4.4 CONTROLLED POINT

The tool table offsets in last few section were determined using the tool tip which is defined as the controlled point ie; just where the tip of the tool will start to cut the material. LazyTurn uses the center of the tool tip radius for the controlled point. The tool radius for different inserts varies and manufactures specify the radius along with tolerances. When a tool is manually ground then the radius needs to be known. The picture below shows a diamond shaped 30 degree tool with a tip radius of 0.009" as measured with a toolmakers microscope. The measured radius should be used in the LazyTurn Tool Dialog. The more accurate the better.



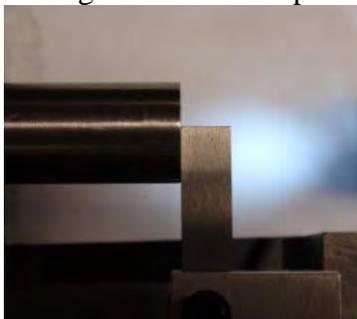
Mach3 Turn uses the offsets in the tool table to adjust tool placement to the part in axis moves. A user can manually copy each of the tools and change the tool number to a series of their choice and change the offsets so that the controlled point is the center of the tool.

#### 4.5 SETTING UP THE LATHE USING THE MASTER TOOL

The user must set the lathe up before machining the piece. Basic setup involves placing the piece to be machined into the chuck, aligning the master tool to the piece, and then creating a work offset such that the tooling will be associated to the piece. This is done using a number of the Mach 3 Turn screens.

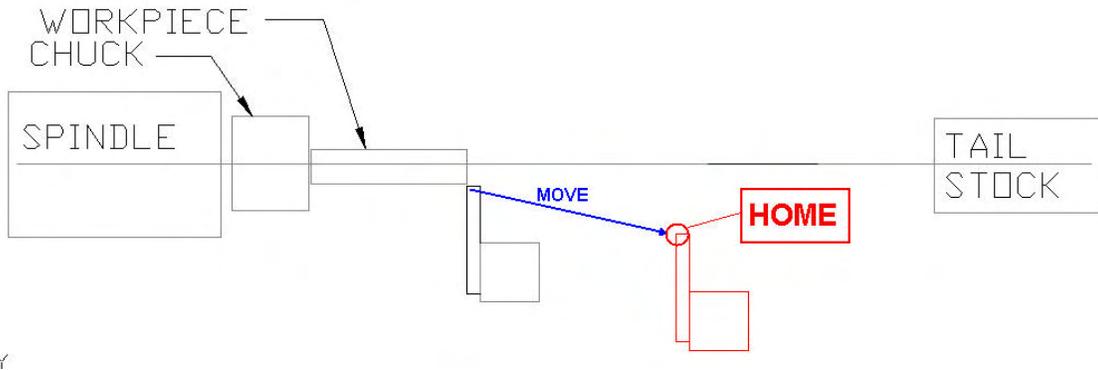
The following example provides for a setup where the tool post is located such that you can go to a repeatable location away from the piece called home, set a work offset to use your quick change tool holder, machine the part, and then return home.

1. The stock is placed in the chuck. The master tool is aligned to the stock such that the edge is placed at the face and center of the stock. ( same as previously shown in Figure 4.1.3 and repeated here)



The Z World X & Z buttons are clicked which removes any work offsets and sets the DRO's to zero.

2. You now move the axes to a location on the lathe such that you can change your tools. Click the Set Home X and Z buttons. ( they are only found in the Manual screen ) This defines home as the location of the tools ( master tool was used so all the other tool tip locations are known by Mach.



3. Move the master tool back to the same point it was aligned to in step 1 above. You will notice that the Dro's display how far you are from the home position. This is the x and z values that will be used to create the work offset to the part from home. You click the x & z PART ZERO buttons and a work offset is created.
4. The tool is located at the part, so.....click Home All button or type G28 in the MDI line, and you will go to the home position.

--- NOW READ SECTION 5.0 ---

## 5.0 LAZYTURN PRECOMENSATED GCODE – TOOL COMPENSATION

The following discussion is provided to the user so they can relate their tool use to pre-compensated Code.

### 5.1 GENERAL

LazyTurn assumes the tool ( the controlled point / center of tool tip radius ) used for a rough or finish pass is set in the tool post in agreement with the tool description given in LT's tool dialog. For example, if rotated on some angle, then it is rotated to that angle. LT also assumes that the center of tool tip is related to X & Z=0. Not like what was shown for Mach3 Turn where the tip cutting point is at 0,0.

Consider the following:

The tool post is fixed to the carriage. You can rotate the cross slide on the carriage when setting the tool to the master tool, thus the offsets for the tool would be correct. How accurately you can turn and then return the cross slide back to the pre turned position can affect all the offsets.

If your lathe does not have a rotatable cross slide, then once you loosen the tool post holding screw, well, practically speaking, all the tool table offsets are no longer correct.

Lathes with automated tool indexers don't have the repositioning problem noted above, but, they may need to have a special ground bit which satisfies a locking position. Most users don't have them to begin with!

You can have a custom holder, but, most users will simply want to use their typical insert repositioned. Most users / hobbyist do one of a kind stuff which is a very different situation than setting up tooling etc. for a production run.

So simply said, tool setup and accurate tool positioning can be a problematic. If standard right, left / center position tools can be used, or the same tool will be used then use of the tool changer works rather well. Should you need to use something other than standard tools during the passes then resetting is required.

### 5.2 THE PRECOMPENSATED CODE

Mach 3 Turns tool table provides for additional inputs namely tip direction, tip radius, wear offsets and were not covered because they do not apply.

LT pre-compensates based on the center of the tip radius. Mach bases the cutting on the tip where it will just start cutting. When you use compensation in Mach you need to define the tip radius and angle in the tool table. Then you need to apply and cancel the compensation accordingly to different sections of the profile which is not an easy task in coding. Thus with pre compensated code none of manual coding is required or to use it

you would be generating code out of high end program. So there is no need to define those items in the tool table. LT provides for a G40 before each pass in case you use a tool number which has that compensation.

## 6.0 OPTIONS

### 6.1 TOOL TABLE

As of this writing or until Mach3 Turn is updated a few comments are noteworthy:

1. Do not use front and rear tools definitions for your tools. Choose one or the other but not both.
2. I don't recommend using the other settings such as tip direction, diameter, and wear offsets. Frankly, I have never used them and not sure if they are buggy, but irrelevant you won't be using them because of pre compensated code.
3. The tool table lists 253 tools, but you can only use 99 of them because of the the tool numbering format in Mach3 Turn is based on industry standards of 99 tools.
4. Tool "0" is not accepted as a call out in some programs.

Lathes use the format TAABB to define the tool number and the offset.

AA is the tool number – T01 to T99

BB is the offset number – TAA01 to TAA99

The offset can be associated to a tool. IE; you have 9 tools with a specific offset based on the tip radius as follows:

T0101

T0202

T0303

T0404

T0505

T0606

T0707

T0808

T0909

Then you can also have another set based on center line of the tip radius by just manually changing the number and adjusting the offset by the radius in the tool table as follows:

T2001 or T0111 or T0121

T2102

T2203

T2304

T2404

T2505

T2606

T2707

T2808

T2909

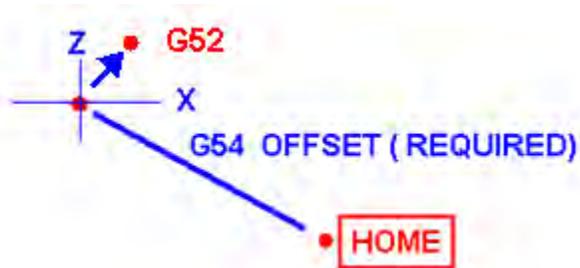
The user would know that series "T20 –T29 are to be used for LazyTurn. Same tool just a different offset. All that really matters is that the user have some kind of logic so the proper tool is defined in the LazyTurn Tool dialog. This would not require modifying the original tool offsets and allow for the use of the same series of tools to be used with a different offset.

The controller ( Mach3Turn) applies offsets in strict order so first part offset, then the program offset, then the tool offset and other modifications as defined by the program.  
( *confirm order* )

## 6.2 USING G52 – TEMPORARY OFFSET

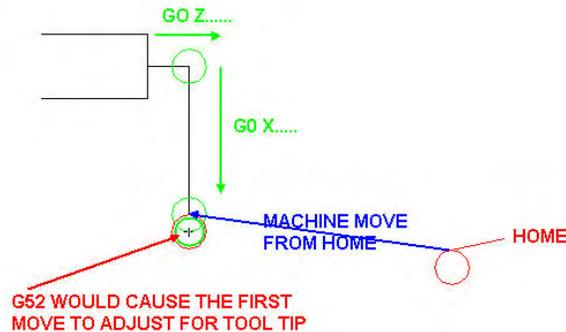
If a tool table was based on the cutting tip, and the user wanted to have a tool based on the center of the tip radius, a G52 command can used to accomplish it.

A G52 shifts the part zero within the current work offset by a specified amount. The G52 requires an active work offset. The G52 X....Z..... is placed before the two G0 moves and a G52 X 0.0 Z0.0 is placed after the moves to cancel the shift.



**FIGURE 6.2.1**

The temporary shift in the axis would cause the tool to be aligned upon the first axis move from home.



**FIGURE 6.2.2**

G52 EXAMPLE FOR X TOOL ALIGNMENT "only": TOOL RADIUS =.010"

G52 X-.010 ( provides for adding the radius to the actual length traveled in the G0x.....move)

G0 Z .....

G0 X.....

G52 X0.0 ( cancels the G52 and returns to work offset)

G.....no difference from here as the code is still precompended including the clearance.

#### COMMENTS:

1. If the same tool radius is used for all passes then only need to implement the G52 one time and home posting is only required once.
2. A G52 is always required on the first pass, and a G52 is always required if the tool radius changes from the previous one.

### 6.3 MANUAL TOUCHING EACH THE TOOLS

The user can simply select T0000, ( which has no X,Z offsets), zero the new tool, then Select the new tool, at which time the X,Z offsets in the table take effect. Of course this means no length offset being used since the X is being zeroed for each tool....the table would simply have X and Z offsets equal to that tools radius..

### 6.4 PROGRAM AND MACHINE COORDINATES THE SAME

"Is it possible to shift the X origin (zero position) to the outer diameter of the starting stock so that setup operations whether using wizard or LazyTurn G-code are similar? "

Most all programs have X=0 as the origin for their code because it can be a repeatable place to easily find and reference to. There are a lot of ways of accomplishing what you ask. Depends on how "you" want to work and your understanding of Mach3 Turn.

A suggestion for when in MACH3Turn:

Zero world X and Z which ( cancels any G54 so there are no offsets ) and set home somewhere away from the piece, move the tool "controlled point" to where the gcode for the part was created, click part zero X & Z, ( now machine=program=part cords at the same point ) move the tool to where you want to start cutting, cycle start the program and all should work just fine.

*NOTE: If you change tool #0 offset then all the other tool get changed and they may longer be correct.*

*Note that if tool #0 is called out in the Auto mode that there will be no offset so you would just set the tool to the end of the stock, put the start move from the G code program into the axis locations leaving the part location a zero and ready to cut but that's a different story on how to apply the tools to a program.*

**6.5 GCODE – FIRST MOVE OF THE MACHINE**

LT’s code provides an initial G0 / rapid move from X & Z =0 to a Z which is the center of the tool radius and then to an X value beyond the stock size. So if you touch off the tool edge to the face of the part then it is in the pre-compensated position in the Z. Well not quite exactly as the tip position may also extend past X=0 ( remember your tool setup was based on the cutting tip).

The first physical axis move from a home position will be not be to 0,0, but rather, to the Pre-compensated location away from the piece.

*NOTE: LAZYTURN PROVIDES A CODE POSTING OPTION TO PUT A G28 AT THE END OF EACH PASS.*

*If you are going to use the same tool for all the passes then there is no need to check it. If you are going to use different tools to accomplish passes and the tools in your tool table agree with the tools selected in LT for the pass then you probably still want to go home to change tools. Now if you use say a diamond shaped tool which needs to be set on some angle other than “centered”, then you will need to come from home and again create an offset to use that tool. You can’t accurately loosen the post nut, rotate the post to place the tool at the required LT and use it without touching off that tool. Rotating the tool voids the initial part zeroing / offset.*

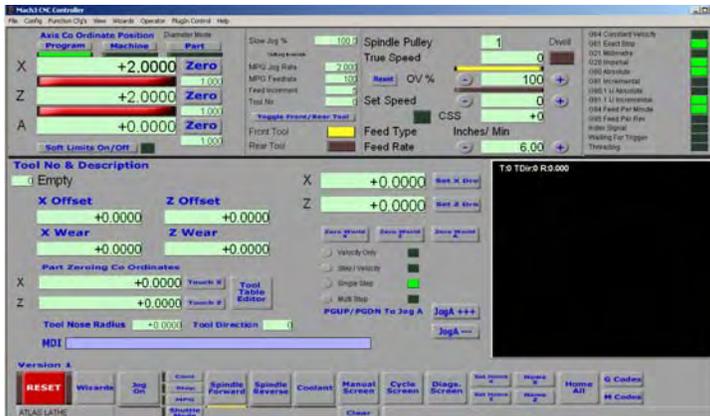
*The home position, ideally, should be such that it will move to the start of the pathing. There is an initial move from X&Z =0 to some a common starting point for all the passes. You can’t start at 0,0 with some shaped tools since they would run into the end of the stock.*

*In summary, LT allows for generating code using some strange tooling positions ,but practical setting up with those tools is a challenge left to the user!*

**6.6 MACH 3 TURN OPTIONAL SCREEN SET**

The user may choose to use a different screen set than the standard one currently provided with Mach3Turn. Until Mach3Turn is updated and a new screen set provided, the one shown below is highly recommend. The screen combines the Tool Table and Manual Screen into one screen and additionally adds a number of nice features not included in the original screen set. It eliminates the jumping back and forth between screens, provides an enlarged graphics area, can see all dro’s on one page, a axis, wizard buttons, status at a glance, and additional buttons for ease of use.

Here is the link for info on the screen set which includes how to install it.  
<http://www.machsupport.com/forum/index.php/topic,13548.msg88932.html#msg88932>



MANUAL SCREEN



CYCLE SCREEN

## 6.7 GCODE OPTIONS

LazyTurn provides all the code necessary to machine a turning. The user may find the following of value in setting up the lathe. Offsets were mentioned in prior sections of this appendix. This just provides some insight in offsets.

G54 – Work Offsets can relate two coordinate systems

G52 – Temporary Offsets work within the work offsets and can temporarily shift the G54 offset

G10 – Tool Offsets allows selection of a tool and will permanently change the G54 work offsets based on the tool

G92 – a legacy Gcode somewhat replaced by G52 and tells the controller where the current tool tip is from program zero. The values used are from program zero to current tool position along the axes.

G53 – Provides for a tool change position

### G10 Tool Offset

The G10 will permanently change the work offsets ( G52 will not as it can be cancelled ) and provides for selection of tool offsets. Note that the tool tip radius and wear offsets may be problematic in Mach ( so I have read and frankly I don't use them in the tool table ).

#### Mach Code description:

To set the offset values of a tool, program

G10 L1 P~ X~ Z~ A~, where the P number must evaluate to an integer in the range 0 to 255 - the tool number - Offsets of the tool specified by the P number are reset to the given. The A number will reset the tool tip radius. Only those values for which an axis word is included on the line will be reset. The Tool diameter cannot be set in this way.

To set the coordinate values for the origin of a fixture coordinate system, program G10 L2 P~ X~ Y~ Z~ A~ B~ C~, where the P number must evaluate to an integer in the range 1 to 255 - the fixture number - (Values 1 to 6 corresponding to G54 to G59) and all axis words are optional. The coordinates of the origin of the coordinate system specified by the P number are reset to the coordinate values given (in terms of the absolute coordinate system). Only those coordinates for which an axis word is included on the line will be reset.

It is an error if:

“ the P number does not evaluate to an integer in the range 0 to 255.

If origin offsets (made by G92 or G92.3) were in effect before G10 is used, they will continue to be in effect afterwards.

The coordinate system whose origin is set by a G10 command may be active or inactive at the time the G10 is executed.

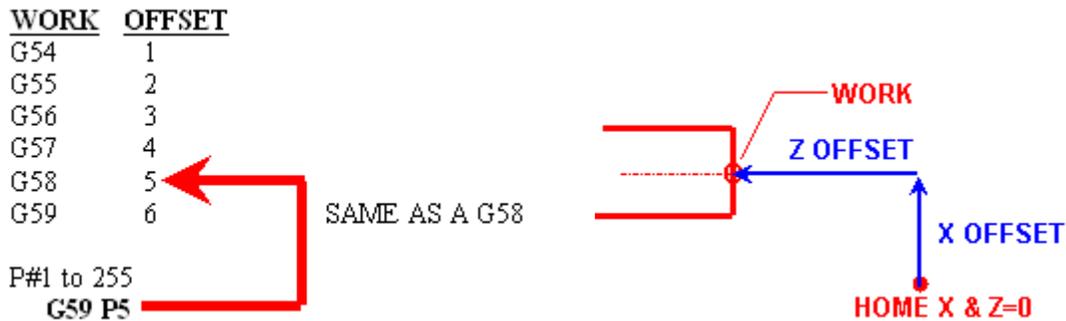
The values set will not be persistent unless the tool or fixture tables are saved using the buttons on Tables screen.

Example: G10 L2 P1 x3.5 y17.2 sets the origin of the first coordinate system (the one selected by G54) to a point where X is 3.5 and Y is 17.2 (in absolute coordinates). The Z coordinate of the origin (and the coordinates for any rotational axes) are whatever those coordinates of the origin were before the line was executed.

**G54-G59 and G59 P1-254 Work Offsets**

The G54 – G59 offset is the distance from “machine zero to program zero” and can be used to relate two coordinate systems. ie; Home position ( machine coordinates ) to part position. This was shown in Section 3.0 of this appendix. G54 is the default system. Not to confuse, but, the reader may also find work offsets called fixture or work coordinate systems. Most projects on the lathe are one of a kind and datum shifts are not usually required but can be useful when doing multiple turnings.

Here is my simplified version of Mach’s description:



Another example of use would be making multiple turnings using the same code by providing an offset to do the other turnings. ie; G54 offset used for the first turning and then G55 Z ( offset amount ) such that the cutter would start at a different point along the Z axis for the next turning.

Mach Code description

To select work offset #1, program G54, and similarly for the first six offsets. The system-number-G-code pairs are: (1-G54), (2-G55), (3-G56), (4-G57), (5-G58), (6-G59)

To access any of the 254 work offsets (1 - 254) program G59 P~ where the P word gives the required offset number. Thus G59 P5 is identical in effect to G58.

It is an error if:

- .. one of these G-codes is used while cutter radius compensation is on.

**G52 Coordinate System Offset**

Use of G52 was covered in Section 6.2 of this appendix providing for a temporary offset within an offset. It basically provides a way of offsetting the original offset ( G54 ) by some value and can be cancelled, thus, if cancelled the original work offsets are not changed as would happen with G10. I have read that it basically replaced the G92.

Mach Code description:

To offset the current point by a given positive or negative distance (without motion), program G52 X~ Y~ Z~ A~ B~ C~ , where the axis words contain the offsets you want to provide. All axis words are optional, except that at least one must be used. If an axis word is not used for a given axis, the coordinate on that axis of the current point is not changed.

It is an error if:

- .. all axis words are omitted.

G52 and G92 use common internal mechanisms in Mach3 and may not be used together. When G52 is executed, the origin of the currently active coordinate system moves by the values given. The effect of G52 is cancelled by programming G52 X0 Y0 etc.

The axis offsets are always used when motion is specified in absolute distance mode using any of the fixture coordinate systems. Thus all fixture coordinate systems are affected by G52

**G92,G92.1,G92.2 & G92.3 Offsets**

See the chapter on coordinate systems for full details. You are strongly advised not to use this legacy feature on any axis where there is another offset applied.

To make the current point have the coordinates you want (without motion), program G92 X~ Y~ Z~ A~ B~ C~ , where the axis words contain the axis numbers you want. All axis words are optional, except that at least one must be used. If an axis word is not used for a given axis, the coordinate on that axis of the current point is not changed. It is an error if:

- .. all axis words are omitted.

G52 and G92 use common internal mechanisms in Mach3 and may not be used together. When G92 is executed, the origin of the currently active coordinate system moves. To do this, origin offsets are calculated so that the coordinates of the current point with respect to the moved origin are as specified on the line containing the G92. In addition, parameters 5211 to 5216 are set to the X, Y, Z, A, B, and C-axis offsets. The offset for an axis is the amount the origin must be moved so that the coordinate of the controlled point on the axis has the specified value.

Here is an example. Suppose the current point is at X=4 in the currently specified coordinate system and the current X-axis offset is zero, then G92 X7 sets the X-axis offset to -3, sets parameter 5211 to -3, and causes the X-coordinate of the current point to be 7.

The axis offsets are always used when motion is specified in absolute distance mode using any of the fixture coordinate systems. Thus all fixture coordinate systems are affected by G92.

Being in incremental distance mode has no effect on the action of G92.

Non-zero offsets may already be in effect when the G92 is called. They are in effect discarded before the new value is applied. Mathematically the new value of each offset is  $A+B$ , where A is what the offset would be if the old offset were zero, and B is the old offset. For example, after the previous example, the X-value of the current point is 7. If G92 X9 is then programmed, the new X-axis offset is -5, which is calculated by  $[[7-9] + -3]$ . Put another way the G92 X9 produces the same offset whatever G92 offset was already in place.

To reset axis offsets to zero, program G92.1 or G92.2. G92.1 sets parameters 5211 to 5216 to zero, whereas G92.2 leaves their current values alone.

To set the axis offset values to the values given in parameters 5211 to 5216, program G92.3.

You can set axis offsets in one program and use the same offsets in another program. Program G92 in the first program. This will set parameters 5211 to 5216. Do not use G92.1 in the remainder of the first program. The parameter values will be saved when the first program exits and restored when the second one starts up. Use G92.3 near the beginning of the second program. That will restore the offsets saved in the first program.

### **G53 Move in ABS Coordinates**

G53 is not related to any program but is often used as a way to position the carriage or cutting point to accomplish a tool change. Use in absolute coordinates, any current offset will not be cancelled, and additionally only used in a block where it is specified.

#### Code description:

Absolute machine coordinates: G53 - move in For linear motion to a point expressed in absolute coordinates, program G1 G53 X~ Y~ Z~ A~ B~ C~ (or similarly with G0 instead of G1), where all the axis words are optional, except that at least one must be used. The G0 or G1 is optional if it is in the current motion mode. G53 is not modal and must be programmed on each line on which it is intended to be active. This will produce co-ordinated linear motion to the programmed point. If G1 is active, the speed of motion is the current feed rate (or slower if the machine will not go that fast). If G0 is active, the speed of motion is the current traverse rate (or slower if the machine will not go that fast).

It is an error if:

- .. G53 is used without G0 or G1 being active,
- .. G53 is used while cutter radius compensation is on.

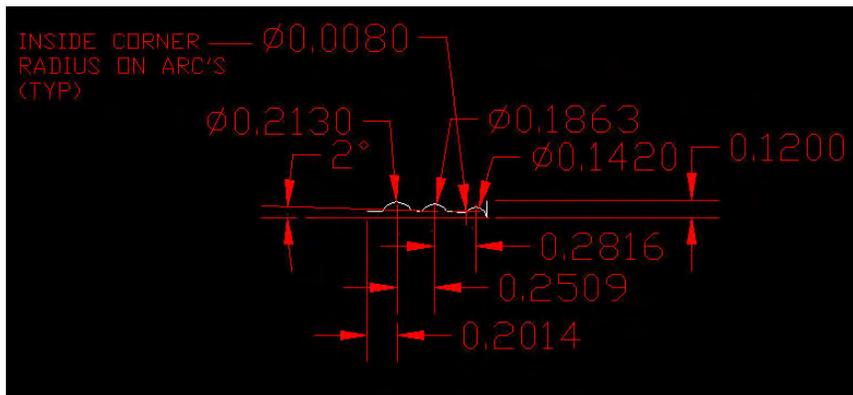


## 1.0 PREFACE

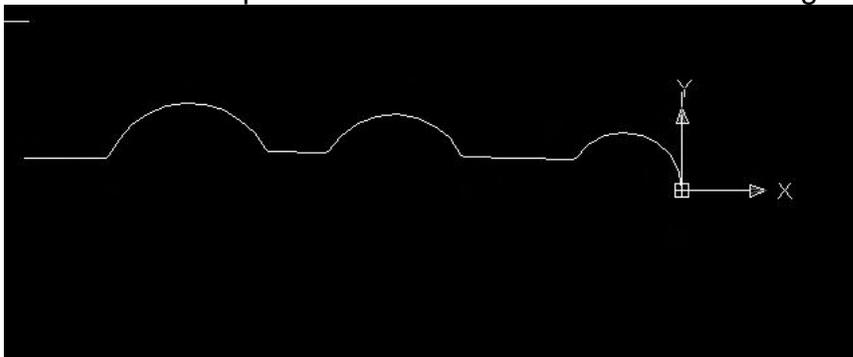
This intent of this appendix is to show the user how to get just the profile Gcode from LazyCam Lathe posted Gcode. In the future it is planned that LazyTurn will provide an option for posting just the profile gcode but until then this option is provided. Note that the code contains arcs, will required modifying the code, and it is assumed that the user has a license for LazyCam Turn ( required to use the lathe module ). Note that the Gcode is strictly the code for the profile thus there is no clash detection, etc.. The info given dose not get into details since there is a manual for LazyCam.

## 2.0 EXAMPLE

The following is the CAD drawing of three balls with a tapered section between them. A DXF file was generated and imported into LazyCam for generation of the lathe code.



Here is what the piece looks like without the dimensioning and as exported.

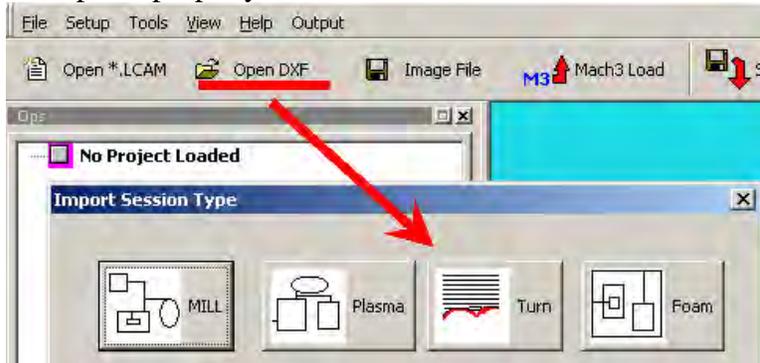


In order to cut the profile from beginning to end along the profile for the above piece the user must use a "point tool" and ground in such a fashion such that it will not gouge.

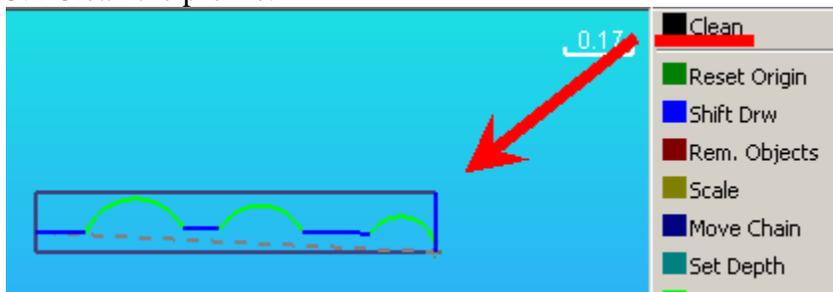
LazyTurns finish cut doesn't have arc coding as of this writing.

### 3.0 LAZYCAM TURN

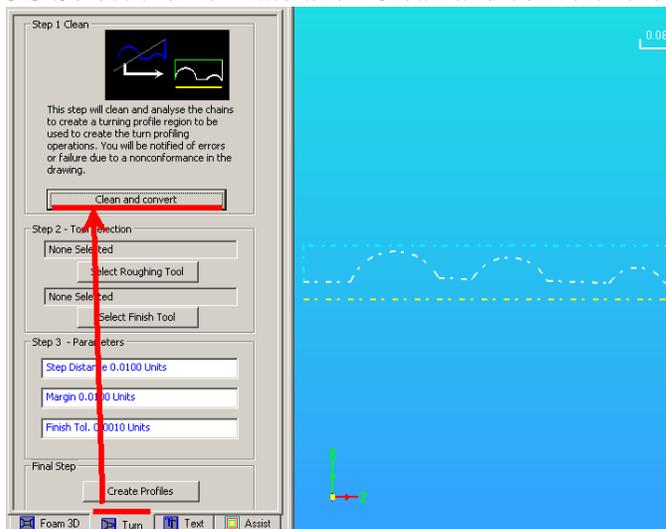
3.1 Open a properly drawn DXF file and select the Turn Module.



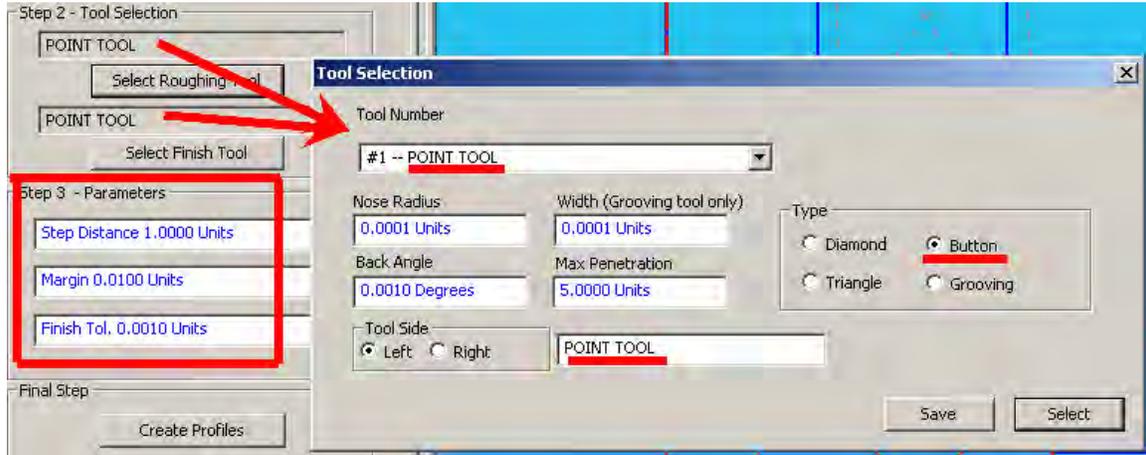
3.2 Clean the profile.



3.3 Select the Turn tab and “Clean and convert” the import.

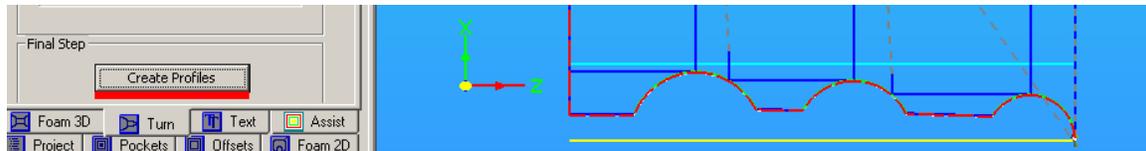


3.4 Create a “point tool” ( it’s an imaginary tool with almost zero radius and width ).  
The Same tool will be used for both roughing and finishing.

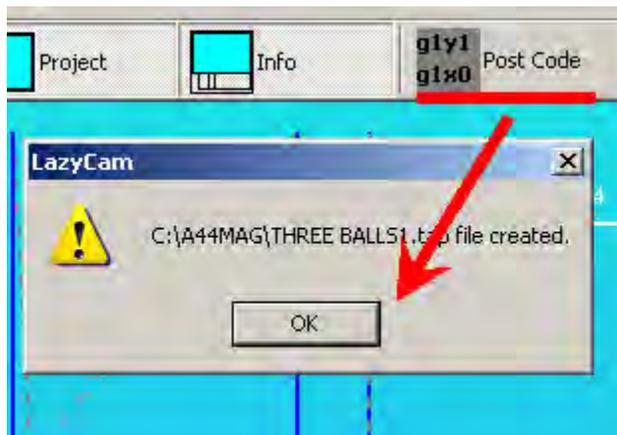


3.5 define the parameters shown in 3.4 above. The step distance is greater than the total depth as we would like a single pass, margin is tool pullout, finish tolerance is very small as shown below.

3.6 Click Create Profiles and profile is highlighted along with any other passes.



3.7 Post the code.



3.8 Exit LazyCam, open the code file with Notepad, delete any code you don't want, the profile code will always be the last number of lines. You will need to add prep moves for tool placement. The code can be appended to LazyTurn generated code if so desired or run by itself after your modifications. Note that you would leave a finish allowance in LazyTurn since this code will cut the profile in one continuous cut along the profile.

```

THREE BALLS1 - Notepad
File Edit Format View Help
N10 G60
N20 G0 X1.1310 Z0.0011
N30 M3
N40 G1 X-0.0038 F60.00
N50 G3 X0.0000 Z0.0012 I0.0038 k-0.0720
N60 X0.0038 Z0.0011 I0.0000 k-0.0721
N70 G1 X0.0114
N80 X1.1310
N90 X0.0038 Z0.0011
N100 G3 X0.0721 Z-0.0709 I-0.0038 k-0.0720
N110 G1 X0.0721 Z-0.2916
N120 G3 X0.0943 Z-0.3524 I-0.0722 k-0.0608
N130 G1 Z-0.5514
N140 G3 X0.1077 Z-0.6034 I-0.0943 k-0.0519
N150 G1 X0.1077 Z-0.8059
N160 X0.2249
N170 G0 X1.1310 Z-0.0709
N180 G1 X0.0721
N190 G3 X0.0387 Z-0.1318 I-0.0721 k0.0000
N200 G1 X0.0426 Z-0.2683
N210 G3 X0.0721 Z-0.2916 I-0.0427 k-0.0841
N220 G1 X0.1056
N230 G0 X1.1310 Z-0.3524
N240 G1 X0.0943
N310 G3 X0.0411 Z-0.7029 I-0.1077 k-0.0000
N320 G1 Z-0.8048
N330 X0.1077
N340 X1.1310
N350 G0 X-0.0038 Z0.0000
N360 G3 X0.0000 Z0.0000 I0.0038 k-0.0710
N370 X0.0038 Z0.0001 I0.0000 k-0.0711
N380 G1 X1.1300
N390 G0 X0.0038 Z0.0001
N400 G3 X0.0376 Z-0.1313 I-0.0038 k-0.0710
N410 G1 X0.0417 Z-0.2689
N420 G3 X0.0462 Z-0.4335 I-0.0417 k-0.0835
N430 G1 X0.0485 Z-0.5084
N440 G3 X0.0401 Z-0.7022 I-0.0485 k-0.0850
N450 G1 Z-0.8049
N460 X1.1300
N470 M30

```

**DELETE CODE YOU DON'T WANT  
THIS IS THE PROFILE CODE**

3.9 View the code in Mach3Turn.. Make sure your configuration is correct.

