

**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 will 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.

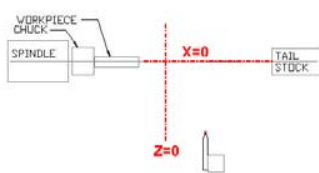


FIGURE 2.1.0

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

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.

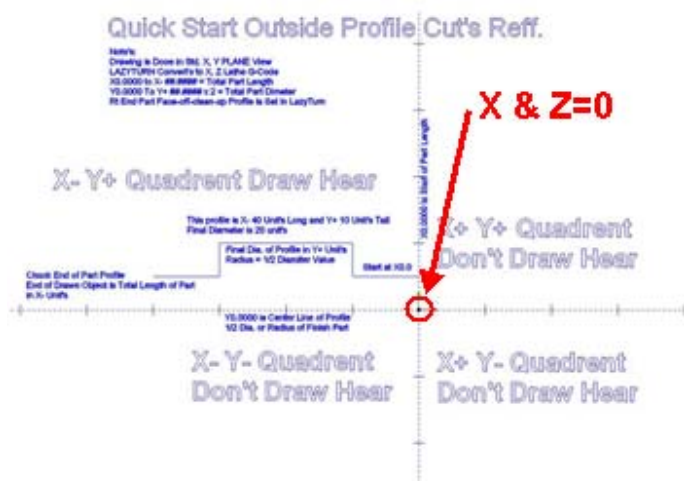


FIGURE 2.1.2

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.

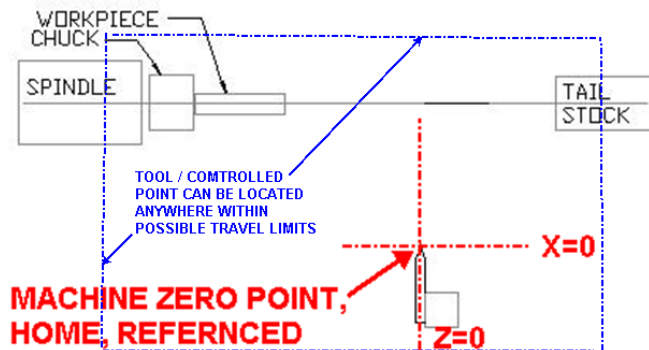


FIGURE 2.1.3

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.

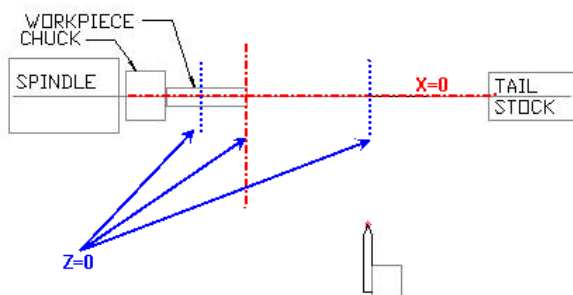


FIGURE 2.1.4

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.

### 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 PART COORDINATES

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.

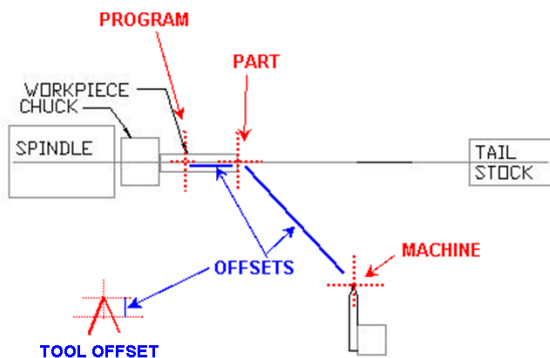
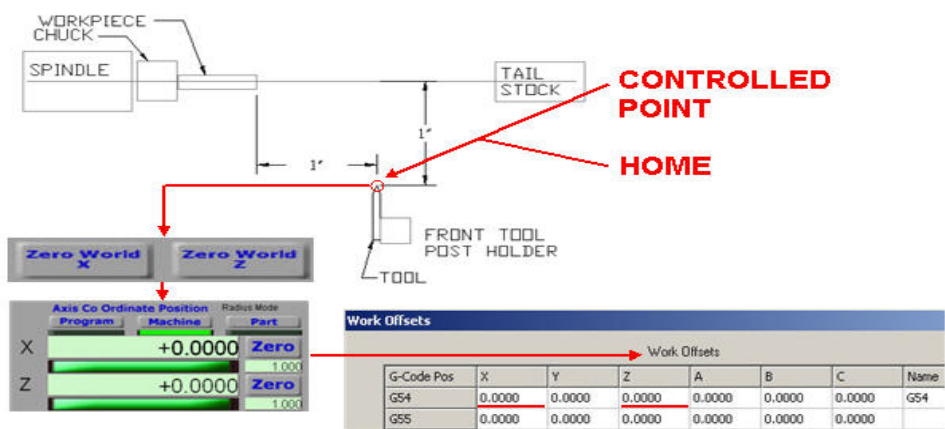


FIGURE 2.15

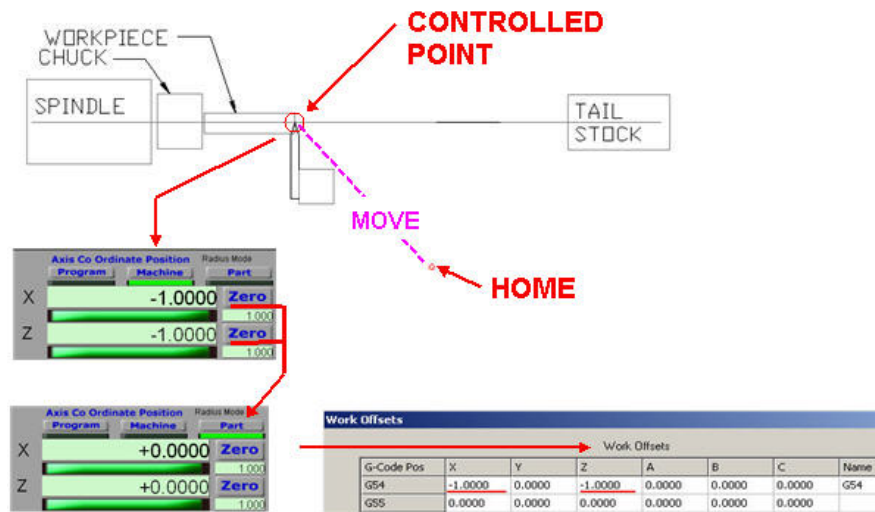
**3.0 ILLUSTRATED EXAMPLE OF THE AXIS COORDINATES**

The figure 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.



**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 below in the Work Offsets table.



**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 lathe 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 below. The master tool height was set to the lathe center.

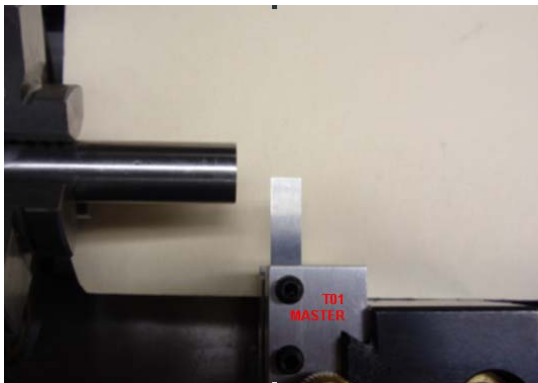


FIGURE 4.1.2

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.

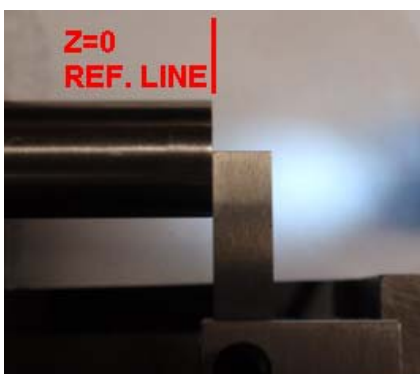


FIGURE 4.1.3

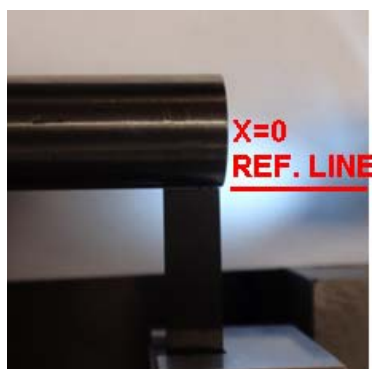
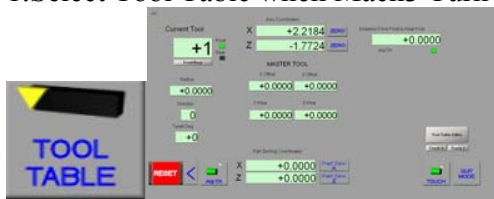


FIGURE 4.1.4

The tool was brought into contact with the face of the reference. You click Zero Z button. The tool was 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.

Here are the steps using the screens:

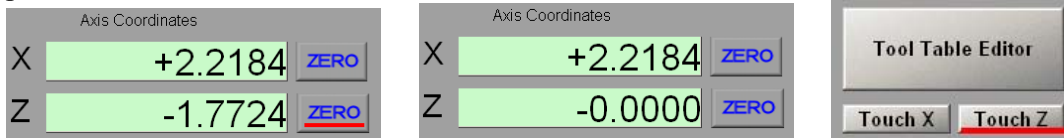
1. Select Tool Table when Mach3 Turn opens which takes you to the tool table screen.



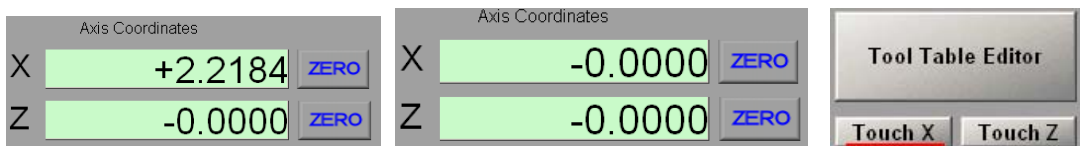
2. Input 1 as current tool and define it as Front by clicking the button below the input. ( 1 will be the master tool )



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.

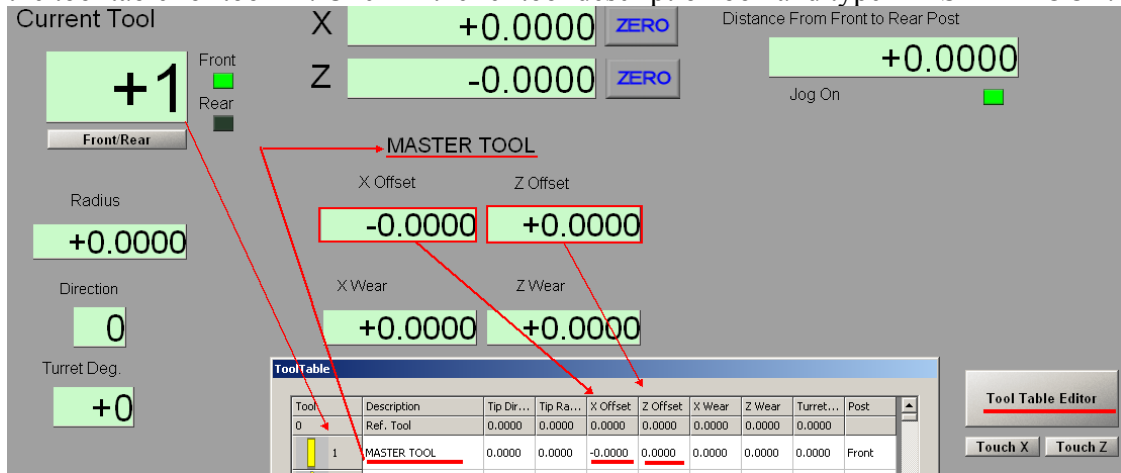


4. Move the X axis to align the tool to the outside surface of you reference and once aligned click the X Zero button and the X DRO will now reset to 0. Then click the Touch X button which puts the current X DRO value into the tool table.





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.

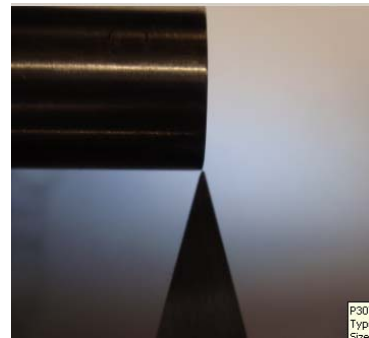
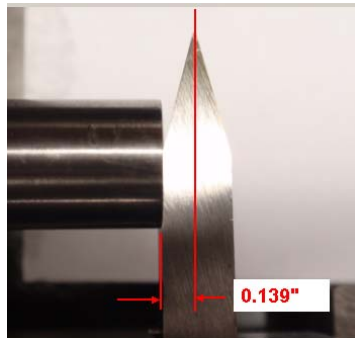


If you open the tool table you will see the X offset for tool #2 and the value shown is absolute based on radius mode.

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:

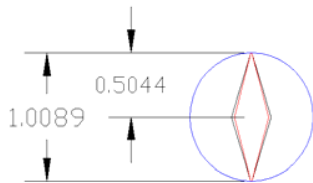
All of the above accomplished defining the offsets of all of your tools as they relate to the master tool. When a tool is called out in a program, Mach will automatically utilize the offsets to control the axis movements as they relate to the program Gcode. Where you define the reference, tool used for 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 manufacturers 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.



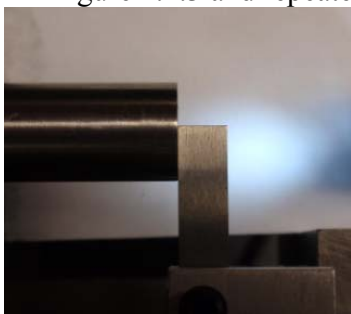
Mach3 Turn uses the offsets in the tool table to adjust tool placement to the part in axis moves. A user can manually copy each of the tools and change the tool number to a series of their choice and change the offsets so that the controlled point is the center of the tool.

#### 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.

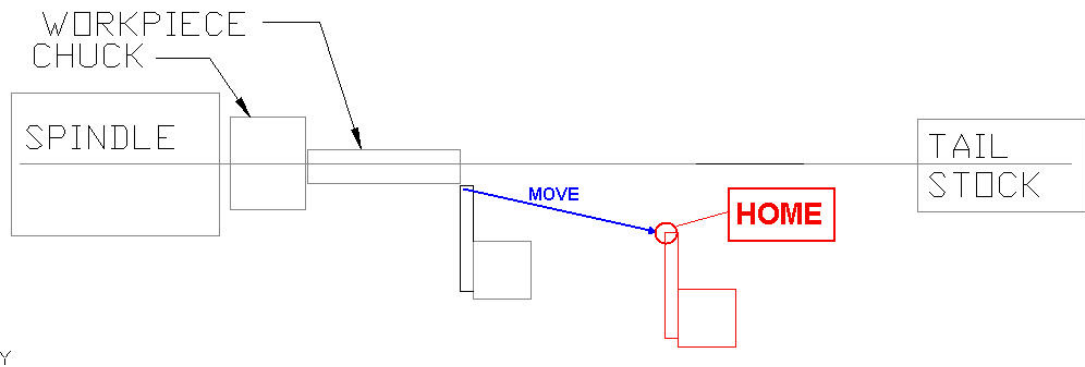
1. The stock is placed in the chuck. The master tool is aligned to the stock such that the edge is placed at the face and center of the stock. ( same as previously shown in Figure 4.1.3 and repeated here)



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

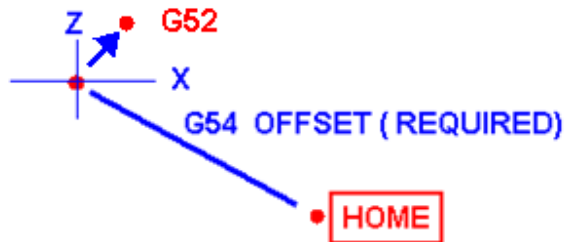
The user would know that series “T20 –T29 are to be used for LazyTurn. Same tool just a different offset. All that really matters is that the user have some kind of logic so the proper tool is defined in the LazyTurn Tool dialog. This would not require modifying the original tool offsets and allow for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

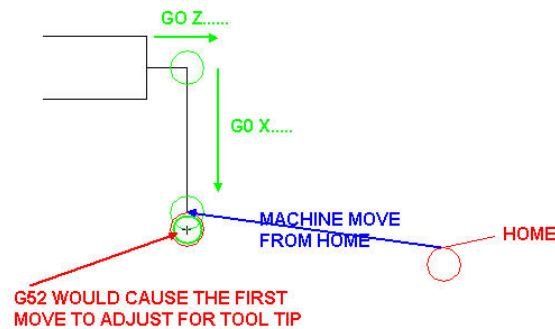
If a tool table was based on the cutting tip, and the user wanted to have a tool based on the center of the tip radius, a G52 command can used to accomplish it.

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**

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



**FIGURE**

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 precompensated 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.*

## 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!*