

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.

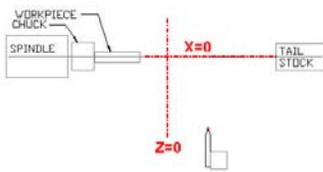


FIGURE 2.1.0

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.

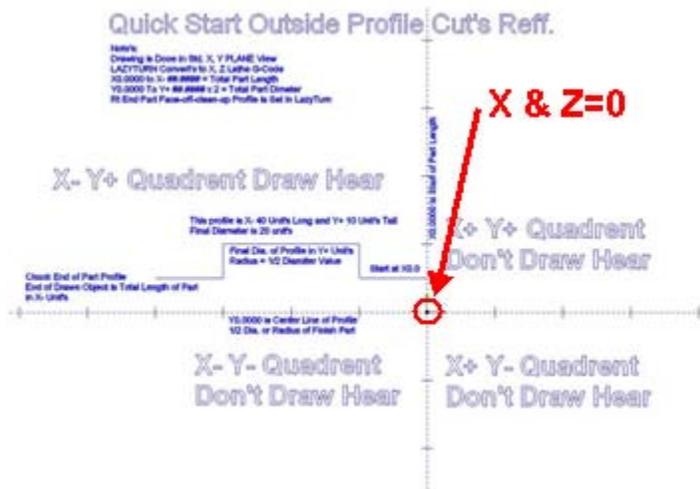


FIGURE 2.1.2

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.

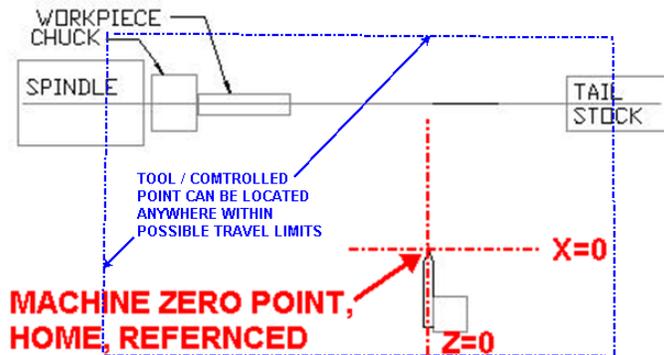


FIGURE 2.1.3

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.

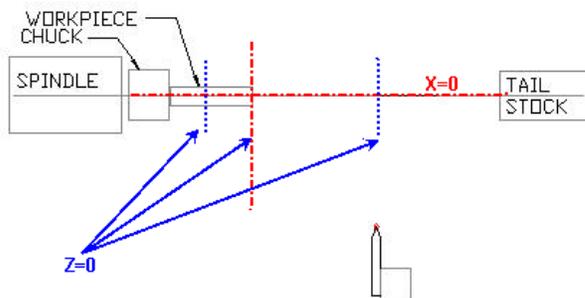


FIGURE 2.1.4

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.

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.

2.1.5 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.

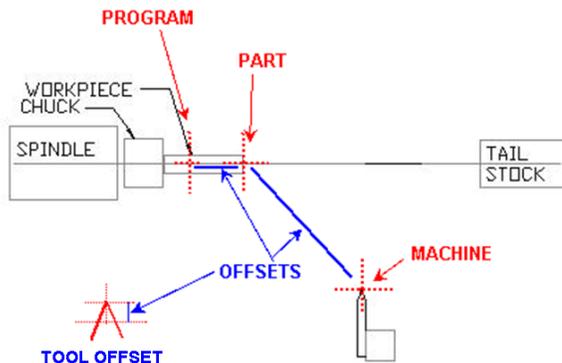


FIGURE 2.15

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.

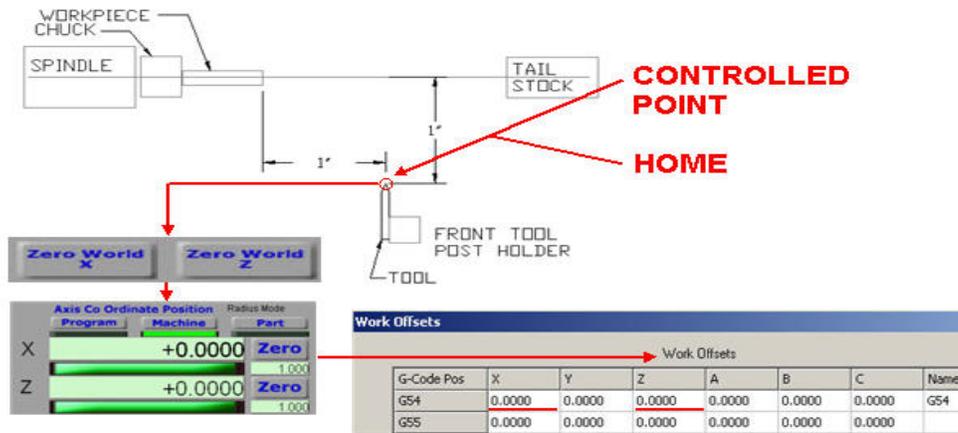


FIGURE 3.0

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.

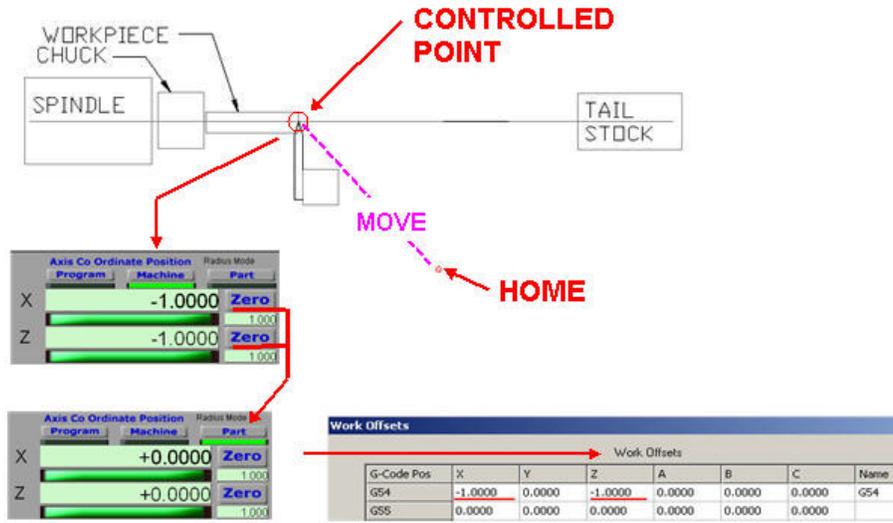


FIGURE 3.1

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.