

3.1 G and M codes on the Emco Compact 5 CNC Lathes

3.4.1 Summary of Commands

Table 3.1: G and M-codes Available with the Emco Compact 5
CNC Lathes (Extension A6C 114 004)

Command	Function
G00	Rapid Traverse
G01	Linear Interpolation
G02	CW Circular Interpolation (2-d)
G03	CCW Circular Interpolation (2-d)
G04	Dwell
G21	Empty Line
G24	Radius Programming
G25	Sub-routine call-up
G27	Jump Instruction
G33	Threading with Constant Pitch
G64	Feed Motors Currentless
G65	Cassette Operation
G66	RS 232 Operation
G73	Chip Breakage Cycle
G78	Threading Cycle
G81	Drilling Cycle
G82	Drilling Cycle with Dwell
G83	Drilling Cycle, Deep Hole with Withdrawal
G84	Longitudinal Turning
G85	Reaming Cycle
G86	Grooving with Division of Cut (parameter H)
G88	Facing with Division of Cut
G89	Reaming and Drilling with Dwell
G90	Absolute Mode canceled <i>only</i> by G91
G91	Incremental Mode canceled <i>only</i> by G90 or G92
G92	Set Register (Zero Point Offset) Absolute Mode
G94	Feed in mm/min (or in/min)

Table 3.1: Emco Codes (continued)

Command	Function
G95	Feed in mm/rev (or in/rev)
M00	Programmed Stop (Pause)
M03	Spindle ON, CW
M05	Spindle OFF
M06	Tool Length Compensation
M08	Switch exit X62 PIN 15 HIGH
M09	Switch exit X62 PIN 15 LOW
M17	Return Command to the Main Program
M22	Switch exit X62 PIN 18 HIGH
M23	Switch exit X62 PIN 18 LOW
M26	Switch exit X62 PIN 20
M30	End of Program (Must be in Program)
M98	Automatic Compensation of Play
M99	Circle Parameter
Command	Function

3.4.2 G-Code Parameters

Meaning and Ranges of Parameters

Note that when programming on the Emco Compact 5 CNC Lathes, every parameter will have an integer value. (In other words the code is not entered in floating point format.) Also, the machine is set to either English or metric units by setting the switch to either inches or mm. (As you will notice, there is no G-Code to do this.) The scaling (i.e. how many inches or millimeters each unit represents) is shown in Table 3.2, along with the meaning of the various parameters. Additionally, Figure 3.1 demonstrates the direction of the X and Z-axes. Notice that *positive Z* points *away* from the chuck, while *positive X* essentially points "away" from the machine. Although no Y or J parameters exist, the implied direction of the Y-axis is also shown. The significance of this shall be seen when discussing the G02/G03 commands.

Entering Data

When you are keying data directly into the Emco Compact 5 CNC Lathe, you can really only input numbers. Further, all the data you enter goes into one of six columns. The first column contains the program number, and the computer essentially takes care of this for you. (Thus you really only need to worry about entering data in the remaining five columns.) The second column is for the G or M command. By default, the machine assumes you are entering a G-code; to get an M-Code, press the Minus "-" key. The third and fourth columns are used to store the X and Z values for a given move (actually X can also be the pause time), or the I and K distances to the center point (if one is using the M99 command). Note that as one is looking down on the lathe, the positive Z axis points along the spindle axis, away from the spindle and the positive X axis points away from the machine. The fifth column has the widest number of "interpretations," signifying the feed rate, F, the pitch of a thread (which, unfortunately, is also called K), the tool number, T, or the line number, L. Finally, the

sixth column always contains the parameter called H, which generally corresponds to the amount of material taken in a given step in one of the canned functions. Generally, one has to enter a value for every parameter a given function uses. In other words, modality does not really work for most Emco commands.

Table 3.2: Significance and Sizes of Emco Parameters

Parameter and Meaning	Range		Scaling	
	Metric	English	Metric [mm]	English [in]
N Block Number	00-209		N/A	
G Move Command	00-95		N/A	
M Miscellaneous Function	00-99		N/A	
X Coordinate CNC-input	0 - _5999	0 - _1999	0.01	0.001
Z Coordinate CNC-input	0 - _32760	0 - _12900	0.01	0.001
F G94 (per min)	2 - 499	2 - 199	1	0.1
Feed rate G95 (per rev)	2 - 499	2 - 199	0.001	0.0001
I Center Pt X dist.	0 - 5999	0 - 1999	0.01	0.001
K Center Pt Z dist.	0 - 22700	0- > 1999?	0.01	0.001
X Dwell (time) (sec)	0 - 5999	0 - 1999	0.01s	0.01s
J Jump Address	0 - 221		N/A	
T Tool Address	0 - 499	0 - 199	N/A	
H Depth Per Step	0 - 999	0 - 999	0.01	0.001
H Width of Tool	10 - 999	10 - 999	0.01	0.001
H Impulse Edit	0 - 999	0 - 999	0.01	0.001
K Thread Pitch	2 - 499	2 - 199	0.01	0.001

Figure 3.1: Direction of Axes on the Emco CNC Compact 5 Lathe

3.4.3 Detailed G-Code Description

Rapid Traverse G00

Here the X and Z parameters are specified, and the tool moves in a straight line to the destination at maximum speed. As with the DynaMyte mills (and any machine using a G00 code), this is intended for motion in air *only*. Do not cut material with a G00. The Emco lathes also possess both absolute and incremental modes- in fact on the Emco lathes, there are essentially two types of absolute mode. In incremental mode (G91), X and Z represent the signed distances from the current tool point to the final tool position. *Note that the machine default is to incremental mode.* In absolute mode (G90 or G92), X and Z represent the absolute coordinate values of the destination point. However, there is one slight "wrinkle," to absolute mode. By default, any absolute coordinate is assumed to be a *diametral* measurement. In other words the machine assumes that your zero point corresponds to the center point and the X value that you enter is the desired *diameter* to which you wish to move. (So if zero is at the center of the cylinder, you are in inch mode, working in absolute coordinates, and you enter G00 2000 0000, the tool location will wind up being one inch from your zero point.) As

mentioned before, this is the default, but not the only mode. One can specify X to mean a radial value, if one uses the G24 command *before* using the G90 command. Note that for linear interpolation or rapid moves, if *both* X and Z are non-zero, then their ratio must be between 1:39 and 39:1.

Linear Interpolation G01

G01 is used to specify a move in which the tool actually cuts material. As such, the feed rate F must be specified, in addition to the X and Z parameters. X and Z have exactly the same meaning as with the G00 commands. The feed rate F can be used to specify either a feed per minute (G94) or a feed per revolution (G95). If neither G94 nor G95 have been specified, the machine default is to take feed rates as inches (millimeters) per minute. Note that for linear interpolation or rapid moves, if *both* X and Z are non-zero, then their ratio must be between 1:39 and 39:1. (At least this is the ratio for incremental mode. If one is in the "diametral" absolute mode, it may be different.)

Circular Interpolation G02/G03

Circular interpolation works slightly different on the Emco lathes than it does on the DynaMyte mills. If we think of how we defined "Clockwise" and "Counter-clockwise" on the DynaMytes, then G02 and G03 have the same meaning. Recall that a "Clockwise" move was in the direction of the negative axis, while a "Counter-clockwise" move was in the direction of the positive axis. On the Emco lathes, only the X and Z are ever moved, so the "third" axis is the Y-axis. However, the X and Z-axis are set up such that the Y-axis points *down* to the floor. According to our definitions, G02 does result in clockwise motion, while G03 produces counter-clockwise motion. The trouble with this definition is that we always view the lathes from the top, where we see the *negative* Y-axis pointing at us. To an observer viewing the G02 command from above, it may appear to be counter-clockwise, while the G03 appears clockwise. To keep consistent with "standard" G and M code terminology, we shall call G02 *CW* and G03 *CCW*. However, it will again be suggested that you try remembering that G02 is about the negative Y-axis, while G03 is about the positive Y-axis.

Further, on the Emco lathes *all* circular interpolations are taken as *single* quadrant moves (arcs of 90_ or less). (There is no multi-quadrant mode on the Emco lathes.) In order to enter the center point of the circle, the Emco lathes require an additional command, M99, to *immediately* follow the G02 or G03 commands. Note that the I and K parameters that one enters again describe the *relative* distance from the current tool position to the center of the circle, *regardless of whether one is in incremental or absolute mode*. (In this sense, I and K work the same as on the DynaMytes.) Since only single quadrant interpolation is allowed, I and K are *unsigned* parameters. Note that one does not always need to specify I and K, coordinates with circular interpolation. For 90_ arcs one can simply specify the X and Z parameters. If the values of X and Z are such that no 90_ arc is possible, but, one has not used an M99 line following the G02/G03 command, an error will result. X and Z have the same meaning and usage as in G00 and G01. Since G02/G03 are intended to cut material a feed rate must also be specified.

Thus the syntax for a 90_ arc in *incremental* mode is as follows.

G	X	Z	F
(G) 02	$\pm \Delta$	$\pm \Delta$	fff

Whereas for an arc that does *not* run through a full 90°, a little more is required.

G	X	Z	F
(G)02	$\pm \Delta$	$\pm \Delta$	fff
(M/-)99	iii	kkk	

Figure 3.2 illustrates the eight possible motions for 90° circular interpolation with the same radius. These correspond to different *signs* on the Δ (radius) and to G02/G03 commands. (On Figure 1.8, G03 and G02 interpolations are labeled. Additionally at each end point, one sees the *signs* on the X and Z parameters, respectively, displayed in parenthesis.) Notice that if one has the same X and Z values (and I and K parameters, if applicable) for a G03 command as for a G02 command the endpoint will be the same. The only difference is the path that is traveled between the start and endpoints.

Figure 3.2: Eight Possibilities for 90° Circular Interpolation

Delay G04

The delay command requires only one parameter, X, to express the duration of the delay in *hundredths* of a second.

Empty Line G21

This command does absolutely *nothing* to the program when executing. It is sometimes useful as a "placeholder," for future expansion. Primarily, though, it is "left over," from a time when one could not insert blank lines and delete lines. (Now, one can insert blank lines by *simultaneously* pressing the tilde key, ~, and the input key, INP, or delete lines, by *simultaneously* pressing the tilde key, ~, and the delete key, DEL.)

Radius Input with Absolute Values G24

As already mentioned the default "move" mode of the machine is *incremental* mode. When one enters *absolute* mode, using either G90 or G92, the machine will, by default, interpret any X value as a diameter. Thus any move made will really only be "half" as far from the zero point as the X coordinate appeared to specify. (Note that the Z coordinate functions "normally.") If one does *not* want the absolute X coordinates taken as diameters, one can use the G24 *before* using the G90 or G92 commands, and then X will be taken as a radial value.

Subroutine Call-Up G25

To call a subroutine, one simply enters the G25 command, along with a line number, L, to which the program should branch. (Note that the L is entered in the F T L K column.) Make sure that the value in L is a line number that either starts or is in the middle of a subroutine. At the end of the subroutine, one needs to have an M17 command to return control to the line immediately following the original subroutine call. Note that subroutines can be nested up to 5 levels deep.

Jump Command G27

Using a G27 tells the machine to branch to the line indicated in the L parameter.

Threading with Constant Pitch G33

Essentially, all G33 does is tell the tool to move along the Z-axis of the lathe, however it does so at a *constant* pitch (which it verifies by watching the spindle position/speed). As such, the G33 command requires only two additional parameters, the length of thread, Z, and the pitch K (again entered in the F T L K column). Note that Z is a signed quantity (allowing right and left handed threading).

Feed Motor Currentless G64

When you initially start the Emco Compact 5 CNC Lathes, the motors are currentless. However, as soon as you move any slides- in either hand or CNC mode- power to the motors is enabled, and *remains* enabled. If one is not actively using the machine but still leaves the motors to "sit" under power, the motors will get *very* hot. To cut off the power to the feed motors, one should use the G64 command. Note that the G64 command executes as soon as one pushes the input key, INP, and then "clears itself" from the memory immediately after executing. (At that point any commands that were deleted from the given line are restored.) Further note that there is *no* command to *enable* power to the motors, as this *will happen automatically*. Finally note that this is an important command- *use it*.

Threading Cycle G78

This is very similar to the G33 threading motion. However, as G78 is a cycle, the machine can be made to perform several passes, to cut deep threads in a series of shallower passes. To use the G78 command, one needs all the parameters. Here, X describes the "final" depth of the thread, while Z still describes the total length of the threads (again X and Z are in either absolute or incremental coordinates). K continues to signify the pitch of the thread, but for the first time, we also need the parameter H to describe how deep of a cut one wishes to take on each pass. If one enters a larger value for H than the total amount of material that the X values indicates, the machine will simply go to the value indicated by X in a single step. Similarly, if H is set to 0, the machine will go to the value indicated by X in a single step. Finally, if the amount to be removed (as indicated by X) is not an integer multiple of H the machine will take as many steps of H as it can (without exceeding the total amount to be removed), then take a final "clean up" pass.

G78, just like G84 and G88, implements a full cycle that *always* finishes with the tool at its *original starting point*. Further, all these commands have the same basic steps. First the tool moves rapid to the desired "cutting depth," then the tool moves at feed rate, across the workpiece for the correct length

of cut. The tool then retracts, at feed rate, moving in the opposite direction to the first move. Finally, the tool moves rapidly, backtracking the distance it covered in the second move. Then, another rapid move to the correct cutting depth begins a new sub cycle.

Longitudinal Turning G84

G84 is another canned cycle that allows one to take a series of straight (i.e. purely along the Z axis) cuts at progressively "deeper" cuts. Again, it requires that one enter all 4 parameters: X, Z, F, and H. G84, just like G78 and G88, is a full cycle that *always* finishes with the tool at its *original starting point*. Further, all these commands have the same basic steps. First the tool moves rapid to the desired "cutting depth," then the tool moves at feed rate, across the workpiece for the correct length of cut. The tool then retracts, at feed rate, moving in the opposite direction to the first move. Finally, the tool moves rapidly, backtracking the distance it covered in the second move. Then, another rapid move to the correct cutting depth begins a new sub cycle. Figure 3.3 depicts the effect that different signs in the value of `_x` and `_z` have upon the cycle. Additionally, Figure 3.4 further illustrates how the "stepping" works.

3.3: Effects of Sign of `_X` and `_Z` on G84: Here, the step, h has been set to approximately 1/3 of `_X`. Dotted lines are rapid, solid lines are feeds. (Note that *Positive X is Down* and *Positive Z is Right*.)

Grooving Cycle G86

G86 is somewhat similar to the other canned functions (G78, G84, and G88), but there are some differences. To start with, the "sub-cycle" that forms the basis for the G86 command really only has 3 steps.

1. The tool plunges into workpiece (X motion at feed rate)
2. The tool retracts (X motion at maximum speed)
3. The tool advances for next cut (Z motion at maximum speed)

On the final pass, however, instead of advancing for another cut, the tool instead moves along the Z-axis to the original start point. Just like all the other canned functions, G86 *always ends at the same point it began*. Another subtle difference between G86 and the other canned functions is that one enters the width of the tool, not the "step" in the H parameter. The step will always be 10 Z units (i.e. 0.010" or 0.10mm) less than the tool width. Moreover, since one does enter the *actual* tool width, the Z value that one enters is the *total width* of the groove that *will be created* and *not* how far the tool moves.

Figure 3.5 illustrates a typical sequence of moves for a G86 command. Note that the machine was in *incremental* and *inch* mode when the command was executed.

Figure 3.4: Further Examples of G84- Effects of Different Step (h) Values. Machine is in (default) *Incremental* mode. Dotted lines are rapid, solid lines are feeds. (Note that *Positive X* is *Down* and *Positive Z* is *Right*.)

Absolute Value Programming G90

On the Emco Compact 5 CNC Lathe the default mode for all moves is *incremental*. You can change this, by using either the G90 or the G92 commands. If you use the G90 command, the point where the tool is sitting at the instant the command is issued becomes the new zero point. (Note that *unlike* the ah-ha! Artisan controllers on the DynaMytes, the Emco Compact 5 CNC Lathe controllers do *not* let one define a workpiece zero *outside* of the program.) Recall that in absolute mode, the default is to interpret all X coordinates as diametral values. To change this, one must enter G24 *before* entering the G90 command. If a G90 command is processed *after* a G24 command, all X coordinates will be interpreted as radial values.

Incremental Value Programming G91

Entering G91 puts the machine in Incremental mode. On the Emco Compact 5 CNC Lathe the default mode for all moves is *incremental*.

Absolute Value Program- Change Zero G92

On the Emco Compact 5 CNC Lathe the default mode for all moves is *incremental*. You can change this, by using either the G90 or the G92 commands. If you use the G92 command, you must also use the X and Z parameters to tell the machine how far from the current tool position the zero point will be. (This is the *opposite* convention that the DynaMyte G92 command uses.) Recall that in absolute mode, the default is to interpret all X coordinates as diametral values. To change this, one must enter G24 *before* entering the G92 command. If a G92 command is processed *after* a G24 command, all X coordinates will be interpreted as radial values.

Figure 3.5: Sequence of Moves with G86 Command (Note that the machine is in *incremental* and *inch* modes. Additionally, *Positive X* is *Down* and *Positive Z* is *Right*.)

Chapter 4

Notes for the Emco Compact 5 CNC Lathes

4.1 The Emco Compact 5 CNC Lathe Control Panel

4.1.1 Initializing: power, Safety, and Settings

Pictured in Figure 4.1 is a picture of the EMCO Compact 5 CNC Lathe Control Panel. Figures 4.2 and 4.3 show "zoomed views" of the left and right sides (respectively) of the control panel. In the top center (or maybe left of center) of Figure 4.2 (the left hand side of the EMCO Compact 5 CNC Lathe Control Panel), one can see the key which functions as the *Main* (power) *Switch*. When the key is vertical (pointing to the 0), the lathe is *off*; when the key is horizontal (pointing to the 1), the lathe is *on*. Immediately to the right of this switch is the *Control Lamp* which should illuminate when the power switch is turned to 1. Just to the left of the power switch is the *Units Switch*. As one can see, there are two choices: inches or millimeters. Although the purpose of the switch should be somewhat intuitive, it is worth noting that this is the *only* way to switch from one unit system to another. (One can *not* change units via software, or program instruction; it must be done via hardware.) Additionally, you should note that "changing units" is only possible when the controller's memory is *clear*. To the right of the on/off switch is a big red button. This is the *Emergency Stop* button that shuts off everything. (What else would one expect from a big red button?) This is primarily intended as a safety switch. There are ways to stop program from executing *without* shutting off everything (and thereby losing the entire program). However, "better safe than sorry," is probably a pretty reasonable adage in a machine shop, so if you really aren't sure what's happening next, or think that you or the machine or someone next to you (even if it's someone you don't like) *might* get hurt- *use the Emergency Stop button*.

Figure 4.2: The Emco Compact 5 CNC Lathe Control Panel

Figure 4.2: Left Hand Side of the Emco Compact 5 CNC Lathe Control Panel

Figure 4.3: Right Hand Side of the Emco Compact 5 CNC Lathe Control Panel

4.1.2 Spindle Controls

In the *bottom* of Figure 4.2 (the left side of the EMCO Compact 5 CNC Lathe Control Panel), one can see the *Main Spindle* switch, the *Spindle Speed Control* knob, and the *Spindle Speed Indicator* panel. Note that the *Main Spindle* switch is actually a *three* position switch. When the top (labeled HAND) is depressed, the spindle is in "Hand Control" mode. "Hand Control" mode means that as long as the lathe is on, so is the spindle. (In other words, if the *spindle* is in "Hand Control" mode, then the status of the *main controller* and other programming details are irrelevant to the state of the spindle. (Notice here that there is a "Hand" and "CNC" Mode for *both* the spindle *and* the main controller, *and* that the status of one is *independent* of the other. When the spindle is in "Hand Control" mode, the spindle is activated. This is essentially the equivalent of having the spindle on *and* in local mode on the DynaMytes.) If the bottom of the main spindle switch (labeled CNC) is depressed, the spindle is in "CNC Control" mode. When the spindle is in "CNC Control" mode, it will respond to the various M codes (such as M03 and M05) in a program. (Thus, having the spindle in "CNC Control" mode on the EMCO lathes is essentially equivalent to having the spindle on *and* in program mode on the DynaMytes.) Finally, the main spindle switch has a third, "neutral" position. When neither side is depressed, the spindle is *off*, regardless of what else is happening. Just to the left of the main spindle switch, is the *Spindle Speed Control* knob. Notice that the settings indicated on the perimeter of the knob are *percentage* of the maximum possible speed. Actual speed depends on *both* the setting of this knob *and* the positioning of the drive belts (which shall *not* be discussed here). When working with wax (and within the sizes possible), usually somewhere around 900rpm is a "reasonable" setting for turning (slightly less is sometimes better when one is parting). The *Spindle Speed Indicator* panel does give the *actual* spindle speed, *in* rpm. (There is an encoder that actually measures the spindle speed- the indicator *does not* simply feed back data from the control knob.) If you have already looked at the manual for G and M codes on the EMCO Compact 5 CNC Lathes, you will notice that there are *no* codes allowing one to set the spindle speed. Spindle speed is *always* set with the *Spindle Speed Control* knob. Finally, in the bottom center of the control panel (refer to Figure 4.1), one sees the ammeter for the main spindle drive motor. Current to the main spindle drive motor should not exceed 4A. (When machining wax or green plastic this should not be a concern.)

4.1.3 Program Entry, Toolholder and Program Control

Turning to the *right* side of the EMCO Compact 5 CNC Lathe Control Panel (Figure 4.3), one sees the numeric keypad that will be used to enter data for programs. Additionally, just to the left of the numeric keypad are the four *Manual Feed* keys: +X, -X, +Z, and -Z. Refer to Figure 4.5 for an illustration of the directions of the axes. The Z-axis points away from the spindle and the X points away from the machine, as indicated by the respective positions in the feed keypad. (Note that the implied sense of the Y-axis is shown in Figure 4.5, but you can *not* move anything along this axis. The EMCO Compact 5 CNCs are *two* degree of

freedom machines. In other words, in addition to the rotating spindle, we can move two axes- here the X and the Z.) To use the *Manual Feed* keys, one must be sure that the main controller 3 is in "Hand" mode. (Note that the *main* controller and *not* the spindle is what must be in "Hand" mode.) To toggle the "main controller" between "Hand" and "CNC" modes, use the key marked H/C that is the top of the *rightmost* keys. Notice the two icons below this button (shown again in Figure 4.4): one appears to resemble a hand, thus indicating "Hand" mode, while the other depicts several arrows and indicates "CNC" mode. Looking at the top right and left corners of the *right* side of the control panel (Figure 4.3), one sees each of these icons, by itself, atop an indicator light. When you are in "CNC" mode, the appropriate light (far right) illuminates, similarly for "Hand" mode. Additionally, the computer screen atop the control panel (not pictured here) toggles between different "views." In "CNC" mode, the computer screen says "CNC OPERATION" and displays the six columns used for program entry. In "Hand" mode, the computer screen says "HAND OPER." and displays the counter values for the X and Z-axes.

Figure 4.4: "Hand" and "CNC" Icons

Figure 4.5: Directions of Axes on the Emco Compact 5 CNC Lathe

4.2 Operating with "Main Controller" in Manual Mode

4.2.1 Moving the Tool Holder

As previously mentioned, when the main controller is in Hand Mode, the *Manual Feed* keys allow one to manually move the tool holder. The rate at which the tool holder moves (the *feed rate*) can be set using the *Feed Rate Control* knob, shown in the top left of Figure 4.3 (i.e. the top left of the *right* hand side of the control panel). Notice, that the values indicated along the perimeter of the feed rate control knob are *all* in mm/min. *Regardless* of which units one has selected with the units switch, in *Hand* mode, one always sets the feed rate in mm/min. Note that one *must program* feed rates for *CNC* mode. Thus, the *Feed Rate Control* knob is *only* effective when the main controller is in hand mode. Finally, "rapid" moves are achieved in hand mode by *simultaneously* pressing the appropriate manual feed key and the *Rapid Move Key* (~) which is immediately to the right of the *Feed Rate Control* Knob. (Note that "Rapid" moves are *only* intended for moving the tool in air.)

4.2.2 Readouts in Manual Mode

The right hand side display panel indicates the current counter value for either the X or Z-axes, depending upon which is "active." When you move an axis it becomes "active." To toggle "active axes" (*only* one is ever "active" at any time) *without* moving, one can hit the "→" key. Notice the string of indicator lights above the display panel on the *right* side of the control panel. If the X-axis is "active," then the third light from the left will illuminate; the

fourth illuminates when the Z-axis is "active." Additionally, the display monitor on top of the lathe will display the current counter values for both the X and Z-axes. (Thus one can also determine which axis is current by comparing the right hand side display panel with the X and Z values on the display monitor.)

4.2.3 Zeroing/Setting the Axis Counters

To "zero" the counter for the "active" axis, press the DEL key. Pressing the INP key allows one to set the counter for the "active" axis to any arbitrary value. After, the INP key is depressed, the right hand side display panel will go blank and the appropriate LED (for either the X or Z-axis) will flash. One can then key in any numeric value using the numeric keypad. Pressing the INP key again will set the counter to the recently entered value. Note that zeroing or setting the axis counters on the EMCO lathes *only* changes the *local* zero and has *no effect* on the workpiece zero that will be used when running a program in absolute mode. Workpiece zeroes (used in absolute mode) can *only* be set *in a program* and are always at or relative to the current tool position.

Further, note that any time one toggles the main controller to "CNC" mode and then back to "Hand" mode, *both axes are automatically "zeroed" at their present locations*. Again, the axes counter values are really just for one's use in manual mode, and zeroing or setting them does nothing more than make the math a little easier when one is moving from one point to another. However, if one zeroes an axis at an important reference point, then moves from that spot, goes to program mode and back to hand mode, one's reference could well be lost. Because of all this, "good practice" is to be sure one has the tool sitting at the desired reference point *before* toggling the main controller to "CNC" mode.

4.2.4 Manually Machining the Workpiece

To manually machine the workpiece, one must have *both* the main controller and the spindle set in "Hand" mode. The appropriate spindle speed should be selected, using the Spindle Speed Control knob. Additionally, one should set the Feed Rate Control Knob should be set to the proper feed. (Somewhere around 50-100mm=min works well for the machinable wax.) Then the Manual Feed keys can be used to move the tool.

4.3 Various Command Keystrokes

Table 4.1 presents a summary of the various Command Keys and Command Key Combinations. Notice that most of the functions are dependent on the current *mode* of the *main* controller (i.e. "Hand" or "CNC").

Table 4.1: Command Keys on the Emco Compact 5 CNC Lathe

4.4 Working in CNC Mode on the Emco Compact 5 CNC Lathe

4.4.1 Entering and Editing a Program

This section shall present *how* one enters the various G Codes and qualifiers. It will *not* discuss what the commands are or how they work, as this is covered in another section of the lab manual. Finally, it should be noted that the EMCO Compact 5 CNC lathes do *not* retain their programs if one cuts off the main power. As has already been stated, this means that hitting emergency stop will cause all entered programs to be erased.

In order to enter a program in "CNC" mode on the EMCO Compact 5 CNC Lathe, the main controller must be in "CNC" mode. As previously discussed (in Section 4.1), one can determine whether or not the main controller is in "CNC" mode, by checking the LED beneath the CNC icon. Additionally, the display monitor on top of the lathe will say "CNC OPERATION" and displays the 6 columns used to program the lathe. If the main controller is not in "CNC" mode, pressing the H/C button will toggle from "Hand" to "CNC" mode. Once in CNC mode, one uses the →, REV, and FWD keys to move through the program.

The → key functions essentially as a "tab" key. Pressing it once tabs to the next accessible word. When one first switches the main controller to "CNC" mode, the cursor is initially in the N column on line 00. Since the line number is already entered (line numbers *never* need to be entered in the N column), one can press the → key to "tab" the cursor to the G (and M) column. After a program is entered, one can scroll through it a "word" at a time using the → key. To move through the program faster, one can use the FWD key to move to the N column in the following line. Both → and FWD move "forward" through the program. REV allows one to move "backward" through a program. If the cursor is in a column *other* than the N column, pressing the REV moves back to the N column of the current line. If the cursor is already in the N column, REV "jumps" back to the N column of the previous line.

To actually enter or alter data, use the INP and DEL keys. INP can be thought of as the input key. After one has keyed in numeric data, pressing INP enters the data in the register. Pressing INP without first entering any data will copy the data from the preceding line (or most recent line with data in the current column) into the current register. To change an entry that has already been input, one first needs to clear the entry, using the DEL key. Note that pressing DEL does not completely clear the register. One needs to enter new data and press INP to ensure the change is completed. For "larger" changes, one may need to delete or add entire lines. To delete a line, first position the cursor somewhere on the line. Then, simultaneously press ~ and DEL. "Blank" lines may also be inserted, by moving to the line that should *follow* the new line and pressing ~ and INP. (Actually, lines inserted with ~ and INP contain G21, the G code for a "blank" line. One can simply delete the 21 and enter the needed command.) Finally, if one needs to delete the entire program, one can do so by pressing DEL, and (while continuing to hold DEL) then pressing INP. (As has been stated several times, cutting off the main power will also serve to clear the entire program.)

4.4.2 Executing a Program

While the main controller is in "CNC" mode, pressing "Start" causes the program to run from the "current" line (i.e. where the cursor sits) to the end of the program (i.e. the M30 command). Actually, when Start is pressed, the controller first scans the program for errors. If errors are found, an alarm code will be displayed. (Please refer to the Emco Compact 5 CNC Manuals for a complete description of alarm codes.) Alarms must be cleared before *anything* else can be done in "CNC" mode. To clear an alarm, press INP and FWD simultaneously. If no errors are (initially) found, the controller will begin to execute the program.

If one only wishes run a few lines at a time, one can press a numeric key (nn) and then press Start, *while still holding the original numeric key*. The controller will execute the next nn lines. Notice that pressing the Start when the cursor is on the M30 command or on a blank line causes the program to return to the start of the program. Additionally, if the cursor is in the N column, pressing the - (minus) key causes the program to return to the first column of the first line of the program (assuming it was *not* already there). Once the cursor is on the first column (i.e. the N column) of the first line, pressing - checks the first line for errors, and moves to the second line. One can then repeatedly press the - key to debug the program a line at a time.

4.5 Interrupting Program Execution

Program execution can be interrupted in several different ways. Pressing (or slamming) the emergency stop button kills all power to the lathe. If there is ever any kind of a safety concern *hit the emergency stop button*. However, if there is not a safety concern, you may not want to hit the emergency stop button. Hopefully, by now you are tired of reading about how killing the power erases any programs entered into the controller, and can thus understand why emergency stop is *not* always the best method to stop a program from executing. One can use two different keystroke combinations to either stop or pause a program.

Simultaneously pressing INP and REV *immediately* stops the program. If the *spindle* is in CNC mode and an M03 command is active, INP-REV will also turn off the spindle. Note that even though the controller is returned to the *very* start of the program, the tool does not move after one hits INP-REV. Further, there is no way of determining exactly how much of a given command had been executed when INP-REV was hit. Thus, there is no way to retrace the tool motion and reposition the tool at its original starting point.

Simultaneously pressing INP and FWD *immediately* pauses the program. INP-FWD does *not* stop the spindle- even if the *spindle* is in CNC mode, with an M03 active- but it does stop the tool holder at its present position. Additionally, INP-FWD causes controller to insert a "hold" at the break point. When one presses Start, the controller *finishes* the command that was interrupted and then goes on to complete the program. The *controller* does keep track of how much of the given command it had executed before the pause, and will *not* "re-run" any previously executed commands. However, there is no way for the operator to extract any information about how much of the command has been

executed from the computer. Note that the "hold" that gets inserted will cause the controller to begin executing the program at the break point- regardless of where the cursor is positioned. Finally, there is a way to stop the spindle after a pause command, if one is in CNC mode with an M03 command active. Transferring the *main controller* to "Hand" mode and pressing the - (minus) key shuts off the spindle. This method has the following benefit over simply turning the Main Spindle Switch to its neutral position. When one returns the *main controller* to "CNC" mode and presses start, the spindle will automatically start- thus there is no risk of forgetting to restart the spindle.

4.6 Tool Changes

4.6.1 "Typical Tools" Used in the RDPL

Figure 4.6 shows the tools "typically" used in the RDPL. Starting from the left of Figure 4.6, the *right hand turning tool* is so named because it was meant to *approach* the workpiece *from the right*. (Note that this assumes the cutter is working on the "+X" side of the workpiece.) (In other words it was made to cut as it moves *to the left*.) Hopefully, this should be somewhat intuitive from the shape of the cutter. Similarly, the *left hand turning tool* is made to *approach* the workpiece *from the left*. Do *not* misuse the right or left-handed turning tools. Again, just by looking at them, it should be somewhat intuitive that it really does make a difference and cutting "backwards" won't work too well. For cases where one really needs to move to the left and the right, and tool changes are not practical, some clever wag has developed the *neutral turning tool*. Acting as the chunky soup of the turning tool community, the neutral turning tool can cut from the left or the right. Typically, we (here I mean "we in the RDPL," and not some larger brotherhood of man) will use the right handed tool, but the geometry of the workpiece frequently dictates what "approaches" and tools must be used.

Figure 4.6: "Standard Tools" Used with EMCO Compact 5 CNC Lathe

Studying the right hand, left hand, and neutral turning tools, it becomes apparent that using them to machine a narrow groove with straight and square edges could become a serious pain, and may not even be completely successful. What will you do? Where will you turn? Again noting that the name of the tool actually does have some significance, might I humbly suggest the *grooving tool*. Notice that the name is (in fact) intended to signify that this tool is to be used to form grooves (and not that it was developed by Jerry Garcia, Phil Lesh, Bob Weir, and Mickey Hart after an exceptional 30 minute extended jam of "Mindbender"). The grooving tool is also referred to as the *plunging tool*. While a plunging tool by any other name may well smell as sweet, few other names would so succinctly express its intended use. Figure 4.7 shows a close up view of the cutting tip of the plunging tool. (Please note that the exact dimensions of the cutting tip vary from tool to tool- always measure the tool you are using to know the exact dimensions.) Only the tiny tip of the plunging (grooving) tool is intended to cut material, and this small tip is *only* intended to cut material when it is plunging. (My, but those machinists can be literal minded folk, huh?) To cut a groove wider than the plunging tool, one must *plunge* the tool into the workpiece, retract the tool, then move the tool along the workpiece, and again plunge, retract, etc. (Fortunately, there is a canned cycle for this on the EMCO Compact 5 CNC Lathes.)

Lastly, the *parting tool* is designed to part (i.e. cut off some portion of) the workpiece. It is a fairly solid edge that is designed to be plunged deep into the workpiece, as is needed when one wishes to separate a finished part from the rest of the stock. Generally, one will *not* get a very good surface finish when using the parting tool. Thus, if you want a wide groove, refer to the previous discussion of how to do this with the *grooving tool*- do *not* use the parting tool as a grooving tool. (Why would you want to disregard the very clear naming conventions?)

Figure 4.7: "Zoomed View" of Plunging (or Grooving) Tool (with *Approximate* Dimensions)

4.6.2 Tool Changes

Our EMCO Compact 5 CNC lathes do have an optional tool turret, but we will most likely *not* be using it. Instead, you will *manually* change tools. To manually change tools, you need to know two things.

1. How to physically remove one tool and insert another
2. How to get the new tool to "line up" with the old tool

The first item is easier to communicate in a "hands on" fashion so you will be shown how to change tools during your lab session. For now, I will point out the following. There are several socket head cap screws on the tools and toolholders, but *none of them are used when changing tools*. We have two slightly different toolholders on the RDPL lathes: one uses a hex head cap screw to "lock" in the tool, the other a "specialty" piece with a head that looks like an extra thick hex head. Thus, you will use either an open wrench or a socket to change tools on the lathes. If you are using an Allen wrench, please notice that an Allen wrench is *neither an open wrench nor a socket*, read the previous sentence, and *stop what you are doing!*

Lining up the tools is accomplished by moving the tool holder through some compensation move *after* using the first tool and *before* using the second. Listed in Table 4.2 are the *approximate* tool offsets and widths. I will again repeat that all of the values are approximate. To get the exact tool compensation factors, *see the sheet by the specific lathe you are using*. To get the correct tool widths, measure the tool. Tool compensation factors relate other tools to the cutting point on a right-handed cutting tool. For example if you were using a right-handed cutting tool and needed to switch to another tool, the tool compensation factor would indicate how far you would need to move in order to get the cutting tip in the exact same position that the cutting tip of right handed tool had occupied before the tool change. In the case of tools with cutting edges as opposed to cutting tips (like the parting and grooving tools) the tool compensation factor measures from the left corner of the tool. (This is assuming the tool is situated as shown in Figure 4.6.) The existence of a large positive X compensation value indicates that the new tool is much longer than the right-handed cutting tool. In many cases, this might dictate making the compensation move *before* performing the tool change, to ensure there is room for the new tool. (Of course, if you had a large negative move, you would have to make the move *after* changing tools. Basically, you need to *think first* and use a little common sense.)

Table 4.2: Nominal Tool Offsets and Widths

Tool	Compensation		Width
	X	Z	
Right Hand	0	0	N/A
Neutral	0	-275	N/A
Left Hand	0	-550	N/A
Outside Threading	0	-100	
Plunge	400	0	47
Parting	0	0	123

***My Serial Cable was like so...

pins 1,2,3,4 & 7 were used on the DB25 that goes into the lathe.

I connected those to a DB9 connector- pin order matching above sequence=

Shield,3,2,7,5 I thought that I might have to swap the tx/rx lines, but I did not have to.