

OBJECTIVE

Understand Okuma programming terminology and recognize basic programming codes that will be used during class.

INTRODUCTION

For the next 2 days you will be covering information necessary to properly operate an Okuma with the OSP5000 series control.

The primary objective of this course is to give you the information you need to translate your understanding of lathes, whether manual or CNC (Computer Numerically Controlled), for use on the Okuma.

It is important for you to realize that all programs are written for the Okuma as if only the A (or upper) turret were being used even if you are programming a 4-axis (two turret) machine.

A program controls the Okuma spindle and two (2) ballscrews. The first ballscrew is the X-axis which will position the tool to part diameters by moving the turret in an up and down direction. Second is the Z-axis ballscrew which will position the tool left and right to determine the length of parts.

A 4-axis machine has special codes in the program to recognize an additional X and Z-axis ballscrew.

Figure 1-1 shows how X and Z-axis movements are actually referenced to a number line that has both positive and negative numbers.

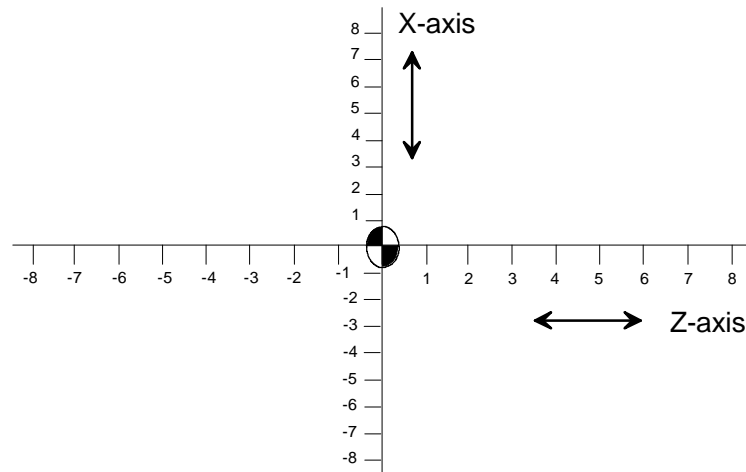


FIGURE 1-1

Notice that there is one point where both number lines cross. Because this point is zero on both number lines it is sometime called **absolute zero**. When writing programs for your Okuma, this point is called **program zero**.

Each graduation on the number lines in Figure 1-1 represents one (1) inch. The X-axis markings are half the size of those for the Z-axis. This is because the Okuma Lathe "thinks" **diametric**. Figure 1-2 shows how the Okuma lathe realizes that a 1 inch diameter is really .5 on both sides of the number line.

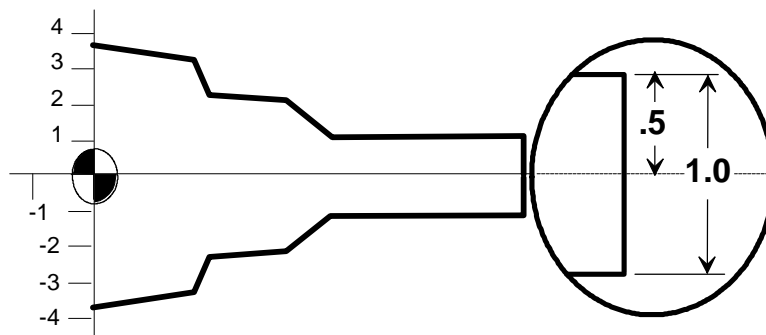


FIGURE 1-2

Because the Okuma uses diametric dimensions you will *only* enter *positive* numbers for X-axis locations during programming.

All programs written for an Okuma will use a method of programming called **absolute**. In absolute programming the lines for each graduation on the number line can be carried out to give you a grid of real points that represent actual dimensions. Figure 1-3 shows an example.

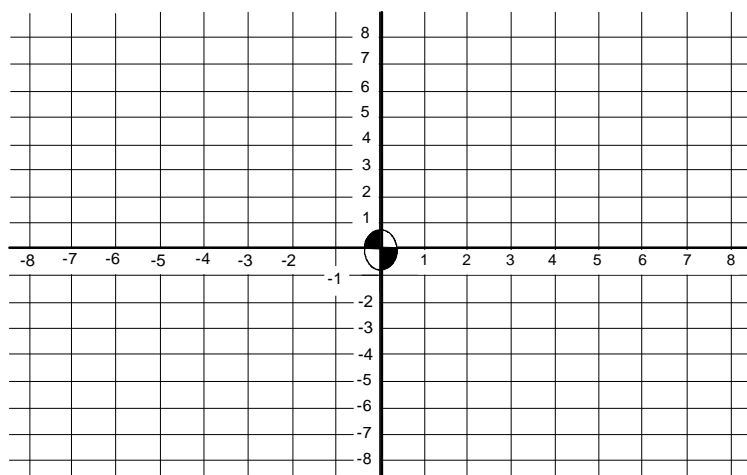


FIGURE 1-3

The diagram in Figure 1-3 shows many one (1) inch squares. If you were to zoom in on one of these squares you would see that each is divided into smaller squares of .0001. Using Absolute programming gives the ability to "tell" the tool to go to a specific location (or address).

Because only positive numbers are entered for X-axis dimensions figure 1-4 shows the grid area used.

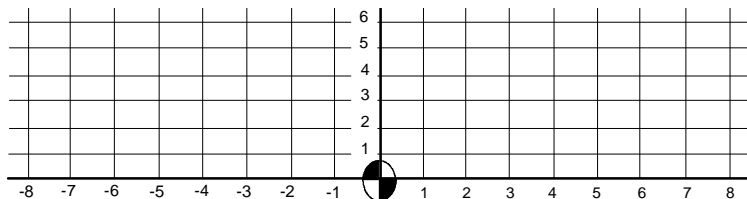


FIGURE 1-4

DEFINITIONS

As with any new undertaking, programming has many terms that may be unfamiliar to you. The following list will explain some of the more frequently used terms.

Character -- Any individual letter, number, punctuation mark, arithmetic operator (+, -, *, /, =) or even a space can be considered a character.

Alphanumeric Word -- A combination of any letter of the alphabet followed by any number is referred to as an alphanumeric word.

Equated Word -- A combination of a word and an equal sign "=" to define the value of a word. The equal sign may be followed by a simple numeric value or an equation to calculate the value.

Block -- Any number of words placed together to form a line of related information is known as a block

In some cases a block may be longer than can be displayed in a line. When this is true a \$ or an & at the beginning of a line tells you that line is tied to the previous to form a block

Execution Priority -- Every word entered into the Okuma has a priority. Regardless of which words are on the line first, those with the highest priority will be acted on first. Words that have the highest priority are S (speed) T (tool change) and M (machine functions).

Default -- When you power up your machine or press the RESET button certain choices (like the absolute programming mode) will be pre-selected.

Modal -- Any word which remains active beyond the block in which it occurs. An example of a modal command would be a feed rate.

Non-Modal -- Any word which is active only in the block in which it occurs is non-modal.

On the Okuma lathe there are many "fixed cycles" which perform complex operations with one line of program. Any value that is set by the fixed cycle will not stay active into the next block and is therefore non-modal.

Format -- The order in which elements of a program are assembled is the format.

On older NC (Numerical Control) machines the format was critical because the machine performed the operations *in the order programmed*. Because the Okuma gives each word an execution priority format is not of major concern.

Absolute Programming -- A coordinate system which has a fixed reference or program zero from which all other points are derived.

Incremental Programming -- A coordinate system which uses distance and direction commands from the current position with no fixed program zero.

File (or Program) -- As in most computer systems, a file is a specific group of data that is stored and named for future use. In the OSP (Okuma Sampling Path) computer a file generally contains *a single program*. A file name can be up to 16 characters long and **must** start with an alpha character (letter).

Some machine controls have a limited number of files available which forces the user to store many programs together.

Bubble Memory -- The OSP uses a device called a "magnetic bubble memory" to store the programs (or files) for future use. This is a physical device that does not need voltage to maintain and "keep" data.

RAM (Random Access Memory) -- This is a device that stores and maintains programs (or files) as long as the OSP is in a "running" condition. This device can contain only one file at a time and is emptied of its file when the control is turned OFF. The OSP "reads" this file from RAM and acts upon its data.

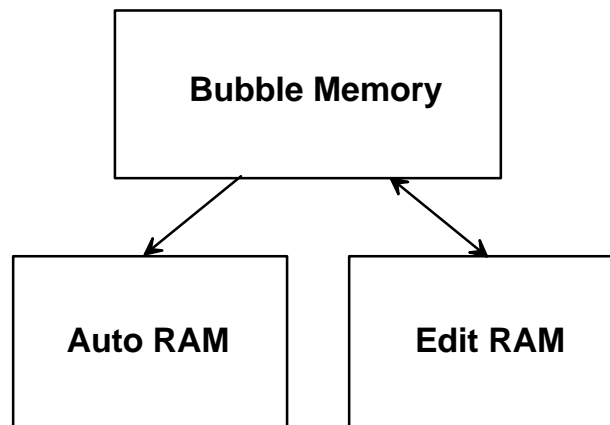


FIGURE 1-5

Figure 1-5 demonstrates that information can be passed from the bubble memory to and from the edit RAM as well as to the Auto RAM but ***not between the edit and auto Ram.***

This design allows you to go to the edit RAM and write a program while the auto RAM is currently running a *different* program. Because these are separate you need to remember that every time you edit a program that is being run in the auto RAM you **must** do another P-SELECT to have the newest program in the auto RAM.

Directory -- A directory is basically a list of programs (or files) that are stored by the OSP in the bubble memory. The directory contains only the names of the programs and not the program information.

Tool Nose Radius Compensation -- This is a function of the OSP software that adjusts the programmed path of the tool tip so that actual programmed dimensions are achieved regardless of the actual radius on the tool tip.

Zero Set -- This term refers to establishing the program zero point for each axis before running a program. This is generally done by positioning a *master tool* to a known value and setting that value in the ZERO SET mode of operation.

Offset -- An offset is the term given to a value assigned to a specific tool and axis designation. The offset value can be automatically calculated for each new tool and will represent *how much different it is than the master zero tool*.

Setting the Okuma lathe using a master zero tool gives you the ability to change from one set-up to the next without touching off all the tools again. Because the offset is how much *physical* difference there is between a tool and the master tool, simply redoing the zero set brings all tools left over from the last job to the correct setting.

ALPHA CODES

Following are definitions for various commonly used alpha codes. The alpha code will be the first character of an alphanumeric word and will instruct the OSP to perform a task; the number following the alpha code tells the OSP how much or what.

A -- Used with a single axis command to execute an angular move.

B -- Used in threading cycle as a tool tip included angle. Infeed angle will be half of the B value.

D -- Depth of cut in fixed and LAP cycles.

E -- Dwell time for fixed cycles, in seconds.

E -- Feed rate for roughing passes in a LAP cycle.

F -- Feed rate.

F -- Dwell time when a G4 is also programmed in the same block.

G -- Preparatory function (see G code list).

H -- Diametric thread height will be used to determine the target dimension and a relationship to the first pass of the threading operation.

I -- When used with a G2 or a G3, designates the incremental distance from the start point of a radius to the center of that radius along the X-axis.

I -- When used with a G71, designates the difference between the start point and end point of the taper in a tapered OD or ID thread.

I -- When used with a G73 or a G74, designates the shift amount in the X-axis for the Grooving fixed cycle.

J -- Thread lead divisor. If F is equal to 1 (one inch) then J is equal to the number of threads per inch (TPI).

K -- When used with a G2 or a G3, designates the incremental distance from the start point of a radius to the center of that radius along the Z-axis.

K -- When used with a G72, designates the difference between the start point and the end point of the taper in a tapered face thread.

K -- When used with a G73 or a G74, designates the shift amount in the Z-axis for the grooving cycle.

L -- Chamfer or radius amount when used with a G75 or a G76.

L -- When used with a G2 or a G3 L designates the size of the radius.

L -- Thread pull out distance. The tool will begin to pull out of the threading pass L distance before the target value.

M -- Machine or miscellaneous functions (see the M code list).

N -- Sequence number or name.

O -- Program name.

T -- The T word is actually divided into 3 pair as shown



in the following example.

The first pair identifies the TNR register that will be read. The second pair identifies the turret position of the tool. The third pair identifies the offset register that will be read.

U -- Finish stock allowance in the X-axis during a LAP cycle.

V -- A common variable (1-32)

W -- Finish stock allowance in the Z-axis during a LAP cycle.

X -- The diametric axis.

G CODE LIST

Z -- The longitudinal axis.

Following is a listing of the more common G codes that you will see displayed by the Okuma during automatic operation.

G0 Rapid Feed -- Used to feed the axes at a rapid feed rate to the commanded coordinate position. The tool path during positioning is not always straight.

G1 Linear Interpolation -- Used to cut a straight line parallel to the X or Z-axis. A taper can also be cut in G01. Feed rate to be used in this mode is commanded by an F word.

G2 Circular Interpolation, CW -- Used to cut a radius in the clockwise direction. Feed rate to be used is commanded by an F word.

G3 Circular Interpolation, CCW -- Used to cut a radius in the counterclockwise direction. Feed rate to be used is commanded by an F word.

G4 Dwell -- Used to activate the dwell function which stops axis motion for any required duration of time during an operation. The duration of dwell movement is programmed by entering the seconds as an F word.

G40 TNR Compensation, Cancel -- Used to cancel the tool nose radius compensation function.

G41 TNR Compensation, ID Ordinary -- used to call out the tool nose radius compensation mode for ordinary ID cutting.

G42 TNR Compensation, OD Ordinary -- Used to call out the tool nose radius compensation mode for ordinary OD cutting.

G50 Maximum Spindle Speed Designation -- Used with an S word to set the maximum allowable spindle speed.

G94 Feed per Minute -- Used to establish the feed mode where distance moved is timed in inches per minute.

G95 Feed per Revolution -- Used to establish the distance moved in a relationship to spindle revolutions.

Any codes that do *not* appear in this manual can be identified in brief by using the OSP5000 Operator's Pocketbook.

M CODE LIST

Following is a listing of the more common M codes that you will see displayed by the Okuma during automatic operation.

M00 Program Stop -- When the M00 is executed, machine operation goes into a cycle stop state; spindle rotation and coolant supply are also stopped. To continue execution of the part program, press the CYCLE START button. This program stop function is effectively used for measuring finished dimensions and for removing chips during a cycle.

M01 Optional Stop -- M01 performs the same function as the M00 Program Stop, except that the control ignores programmed M01 codes unless the OPTIONAL STOP key switch is also turned "ON".

M2 End of Program -- M02 at the end of a part program marks the end of the program as well as initiating a reset of the control back to default conditions.

M3 Spindle CW -- M03 assigns the normal spindle rotational direction (CCW when viewing the chuck).

M4 Spindle CCW -- M04 assigns the reverse spindle rotational direction (CW when viewing the chuck).

M5 Spindle Stop -- M05 stops spindle rotation.

M8 Coolant ON -- This code turns coolant ON if the front panel controls are set correctly.

M9 Coolant OFF -- This code turns coolant OFF if the front panel controls are set correctly.

M23 Chamfering ON -- During the thread cutting fixed cycle this code allows for the "feathering out" of the last thread. The site of this pull out thread will be determined by an L word.

M32 Straight Infeed Along Left Face -- During the thread cutting fixed cycle this code allows the tool to feed down the left face of the thread.

M33 Zig-Zag Infeed -- During the thread cutting fixed cycle this code allows the tool to feed in alternating passes of left face, right face, left face, etc...

M34 Straight Infeed Along Right Face -- During the thread cutting fixed cycle this code allows the tool to feed down the right face of the thread.

M73 Thread Cutting Pattern 1 -- During the thread cutting fixed cycle this code allows the tool to feed toward the destination until it reaches a single depth plus a finish pass. With a single depth of cut remaining the final passes 1/2 a depth, 1/4 of a depth, 1/8 of a depth, 1/8 of a depth followed by the finish pass.

M74 Infeed Pattern 2 -- During the thread cutting fixed cycle this code allows the tool to feed toward the destination in the amount of depths until the thread destination (less the finish pass) is reached. Finally the finish pass is made.

M75 Infeed Pattern 3 -- During the thread cutting fixed cycle this code allows the tool to feed toward the destination in continually decreasing depths.

Any codes that do *not* appear in this manual can be identified in brief by using the OSP5000 Operator's Pocketbook.