

Simple introduction to CNC machining

Attached are some handouts that should help you in creating CNC programs to machine your parts. The first one gives a good introduction to milling machines and some basic cut motions. The second one covers how to select the cutting speeds and feeds so that you achieve a good finish on your parts and so that you don't break a tool or damage the machine. When calculating the speed, keep in mind that our machine has a max spindle speed of 5000 RPM (which actually is relatively fast for milling machines), so depending on your tool size it may be necessary to run a little slower than the optimum. When choosing feed rates, use a value of .005 inches per tooth for aluminum, and .010 inches per tooth for plastic or wax. The third handout shows the coordinate systems on a CNC milling machine, and discusses accuracy. Our machine has an accuracy to 4 decimal places, although if you specify more it will round it off for you. The final two handouts are from our machine's user manual, and they describe the instruction codes and also how to verify programs in the software simulation mode.

Here's the basic procedure for creating programs:

1. Choose the shape and size of stock that you will machine your part from. Determine how you will clamp your stock so that it can be held in a way that won't interfere with the cutting tools.
2. Pick an origin for your part. This will be the point that all other dimensions are referenced from. Typically you will set the origin to be the front left-hand corner of the part, on the top surface. This is the point that the machine will use to make all its moves in relation to.
3. Make a detailed drawing of your part, showing the dimensions of each cutout, and the positions of the cutout relative to the part origin. You will need to dimension both the lateral offsets (X-Y positions) and the vertical depth of cuts (Z positions).
4. Choose what size tool you will use for each of the cutouts. If you need to make a .250" slot, then the natural choice is a 1/4" end mill. If that slot has a rounded bottom, then you will want a 1/4" ball-end mill. By using a tool size that matches the cut size you will have to program fewer movements of the tool. Here are some of the tools we have available:

End Mills: 1/16"	Ball-end Mills: 1/8"
3/32"	1/4"
1/8"	3/8"
3/16"	1/2"
1/4"	3/4"
5/16"	
3/8"	
1/2"	
3/4"	

5. Write the G-codes to move each tool to the positions you want. Each G-code movement will be a position for the tip of the tool to move to. You will have to calculate where to position the tip in order to leave behind the edges you want. Don't cut too deep in a single pass: you should limit the cuts to be no more than 1/3 as deep as the diameter of the tool. Deeper slots will require multiple passes. Group the G-codes by tool, so that you do everything you need with one tool before changing to the next tool. That will save you the time of changing tools more than necessary when you run your program. Don't forget to raise the tool when you move from one area of your part to the next area, unless you are making a cut!

Keep in mind that G-codes are simply the same sort of movement instructions as a machinist would perform on a manual milling machine; you are just telling the milling machine where to move for each step, referenced to your part's origin.

I've installed the Benchman software on the right-hand PC in HBH 2202, so you can load your program and verify it using the instructions in the handout. Please verify your programs before you bring them to run on the milling machine.

Mike Vande Weghe
Research Engineer
Institute for Complex Engineered Systems
Carnegie Mellon University, HbH 2207
412-268-6846 / vandeweg@cmu.edu

1.4.4 Vertical-milling Machine (Vertical Miller)

A wide variety of operations involving the machining of horizontal, vertical, and inclined surfaces can be performed on a vertical-milling machine. As the name of the machine implies, the spindle is vertical. In the knee-type machine illustrated in Fig. 1.32 the workpiece can be fed either

1. Along the vertical axis (Z' motion) by raising or lowering the knee
2. Along a horizontal axis (Y' motion) by moving the saddle along the knee
3. Along a horizontal axis (X' motion) by moving the table across the saddle

In larger vertical-milling machines the saddle is mounted directly on the bed, and relative motion between the tool and workpiece along the vertical axis is

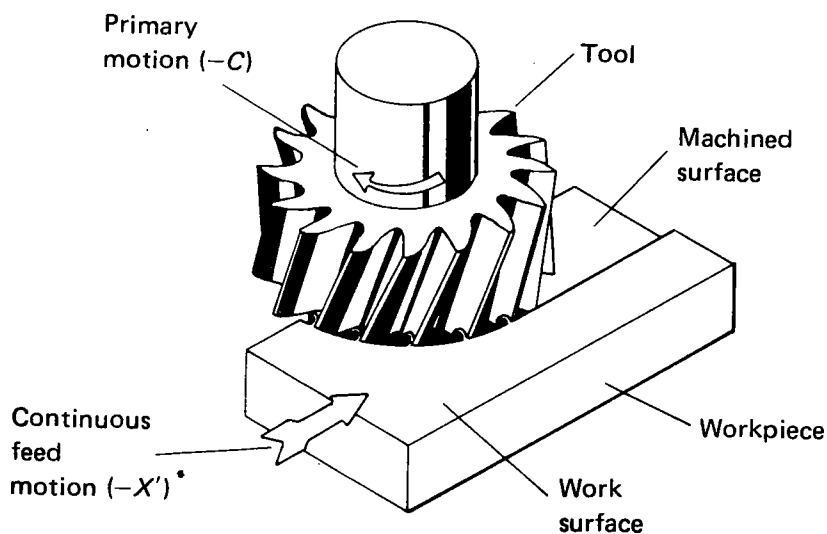
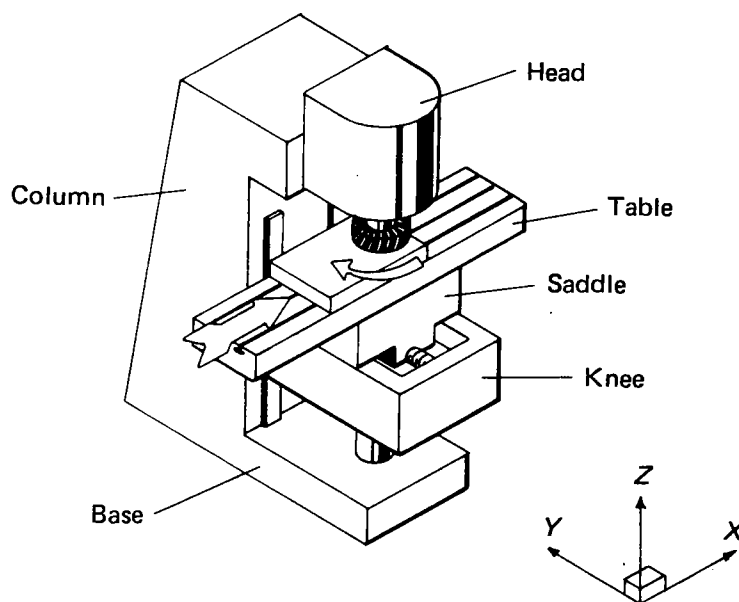


FIG. 1.32 Face milling on a knee-type milling machine.

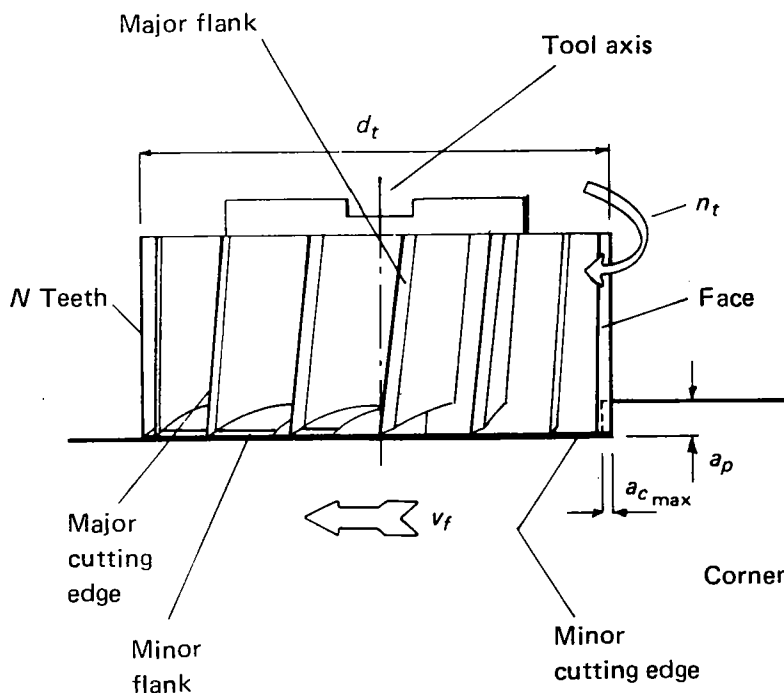


FIG. 1.33 Geometry of face milling, where $a_{c_{\max}} = v_f / Nn_t$.

achieved by motion of the head up or down the column (Z motion); these machines are called bed-type, vertical-milling machines.

A typical face-milling operation, where a horizontal flat surface is being machined, is shown in Fig. 1.32. The cutter employed, known as a *face-milling cutter*, is shown in Fig. 1.33, which also illustrates the geometry of the operation.

The feed f is the distance the cutter advances across the workpiece during one revolution. Thus

$$f = \frac{v_f}{n_t} \quad (1.30)$$

where v_f is the feed speed, and n_t is the rotational speed of the cutter.

If the tool axis passes over the workpiece, the undeformed chip thickness increases to a maximum value and then decreases during the time each tooth is engaged with the workpiece; its maximum value, $a_{c_{\max}}$, is equal to the feed engagement, which is equal to f/N , where N is the number of teeth on the cutter. Thus

$$a_{c_{\max}} = \frac{v_f}{Nn_t} \quad (1.31)$$

In estimating the machining time t_m allowance should again be made for the additional relative motion between the cutting tool and workpiece. As can be

seen in Fig. 1.34, the total motion when the path of the tool axis passes over the workpiece is given by $(l_w + d_t)$ and therefore the machining time is given by

$$t_m = (l_w + d_t)v_f \quad (1.32)$$

where l_w is the length of the workpiece, and d_t is the diameter of the cutter.

When the path of the tool axis does not pass over the workpiece,

$$t_m = \frac{l_w + [2\sqrt{a_e(d_t - a_e)}]}{v_f} \quad (1.33)$$

where a_e is the working engagement. In this case the operation is similar to slab milling with a large working engagement, and the maximum value of the undeformed chip thickness will be given by Eq. (1.26).

The metal-removal rate Z_w in both cases is given by Eq. (1.29).

A variety of vertical-milling machine operations are illustrated in Fig. 1.35. It can be seen that, in one pass of the tool, several combinations of machined surfaces can be produced.

Milling cutters for vertical-milling machines generally have either a bore or a straight shank. Those having a bore are called *shell end-mills* and are secured to an arbor (Fig. 1.36) held in a socket in the machine spindle with a draw bar. Those having a straight shank are either gripped in a chuck or held in the spindle by a screw bearing on a flat surface, machined into the shank.

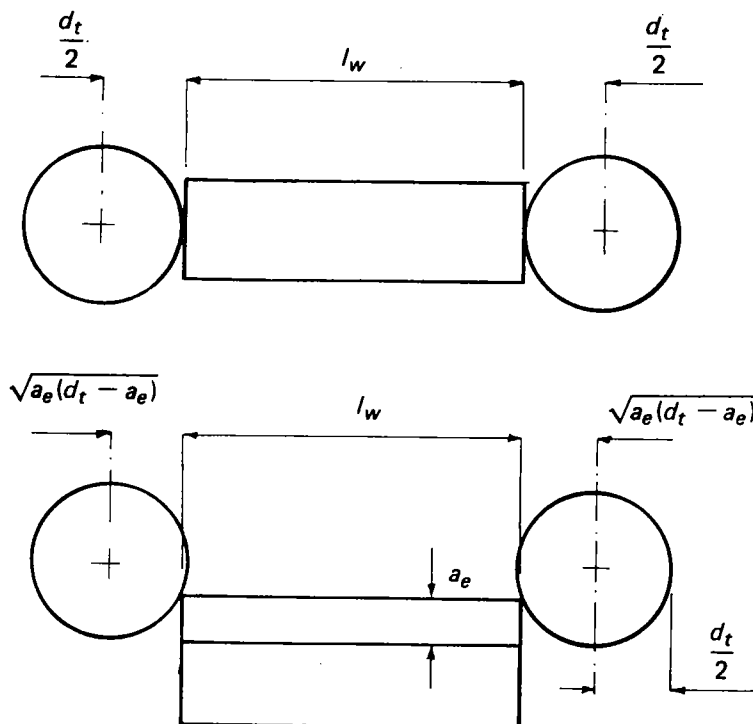


FIG. 1.34 Relative motion between the face-milling cutter and the workpiece during machining time.

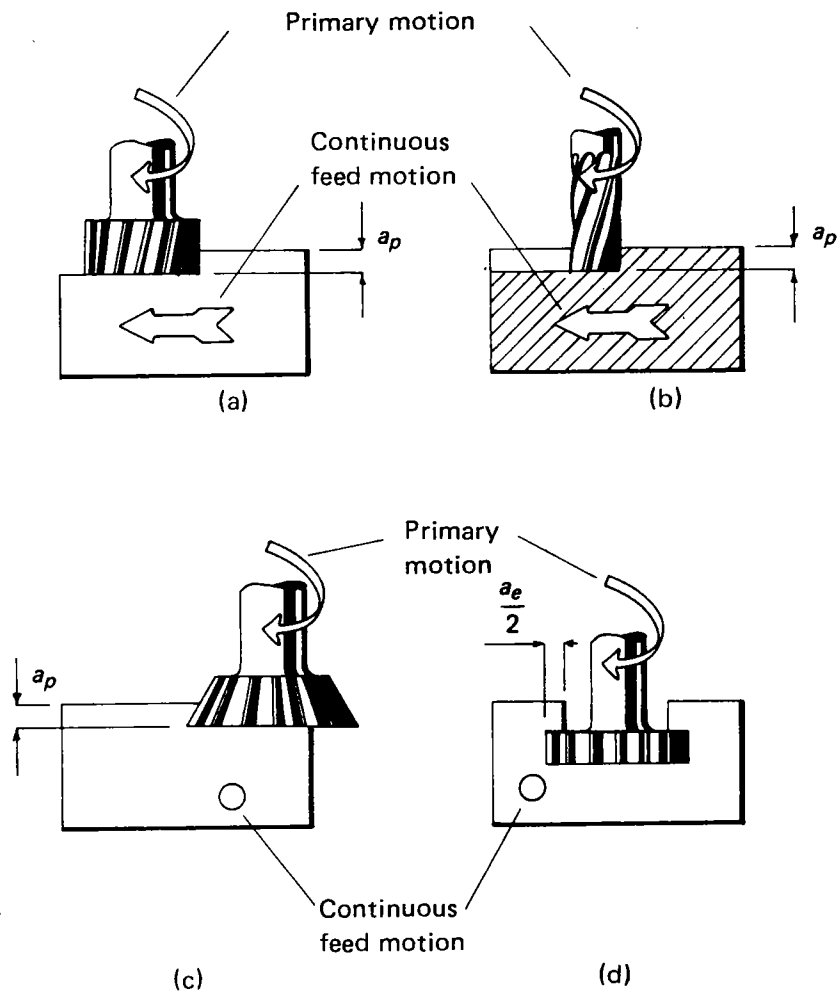


FIG. 1.35 Some vertical-milling-machine operations. (a) Horizontal surface; (b) slot; (c) dovetail; (d) T slot.

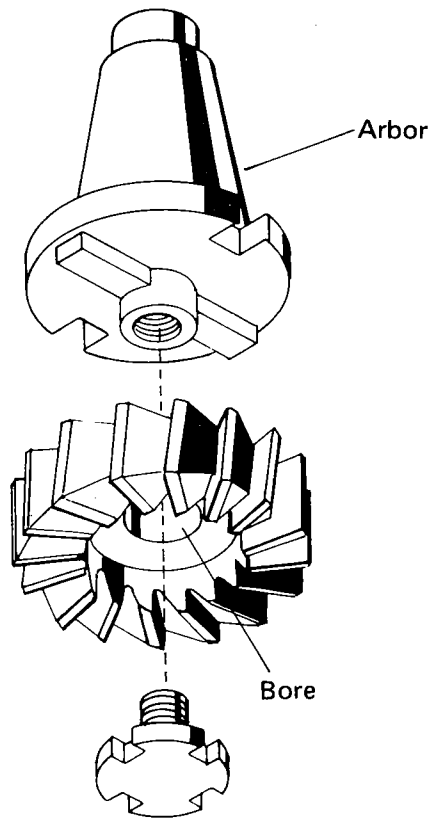


FIG. 1.36 Tool holding for a shell end mill.

Unit 76. Cutting Speeds and Feeds

To operate a milling machine successfully, you should learn how to figure cutting speeds and feeds. While there are formulas to help, you will find that there are many different things that affect cutting speed and feed. Judgment must be used in the final adjustment of the machine. The experienced ma-

chinish merely uses the result of the formula as a guide. You must remember that milling as a production process is primarily concerned with the rapid removal of metal. The amount of metal removed in a given length of time is the important thing in figuring correct cutting speeds and feed. As a beginner, it is

usually best to start with slower speeds and feeds and then to increase them to the maximum limits. Of course, when you use a small hand miller with a hand feed, rapid production is a matter of skill and experience.

1-inch-diameter cutter is employed, it will travel only about 3 inches (1×3.1416). The cutting speed in feet per minute is affected by many things. The two most important are the kind of material being machined and the kind of material in the milling cutter. The cutting speeds shown in Table 76-1 represent averages that will give a fast removal of metal and at the same time assure a long life to the cutter. Too fast a speed will dull or burn the cutter. The cutting speed for finishing cuts can be as much as 40 to 80 per cent higher than for roughing cuts. To find the machine speed to use or the revolutions per minute at which the spindle must rotate, the following should be used:

Cutting Speed

Cutting speed is defined as the distance one tooth of the cutter moves in one minute as measured in feet on the circumference. This is called the surface feet per minute (sfpm). In general, cutting speed is slower for harder materials and faster for softer ones. Cutting speed is not the same as machine speed (rpm). The spindle of the milling machine operates at certain given revolutions per minute. If you place a 4-inch-diameter cutter on the spindle, in one complete revolution a tooth will travel about 12 inches (4×3.1416). If a

$$\text{rpm} = \frac{4 \times \text{sfpm (surface feet per minute)}}{d \text{ (diameter of cutter in inches)}}$$

Table 76-1. CUTTING SPEEDS*

Material to be milled	Material in cutter					
	Carbon tool steel	High-speed steel	Super high-speed steel	Stellite	Tantalum carbide	Tungsten carbide
Cutter speed in feet per minute						
Aluminum	250-500	500-1000	800-1500	1000-2000
Brass, soft	40-80	70-175	150-200	350-600
Bronze:						
Hard	30-60	65-130	100-160	200-425
Very hard		30-50	50-70	125-200
Cast iron:						
Soft	30-40	50-80	60-115	90-130	250-325
Hard		30-50	40-70	60-90	150-200
Chilled			30-50	40-60	100-200
Malleable iron	35-50	70-100	80-125	115-150	250-370
Steel:						
Soft	30-45	60-90	70-100	150-250	
Medium	30-40	50-80	60-90	125-200	
Hard		30-50	40-70	100-150	

* Learn what cutters will stand. Start with slow speeds and step up. For hand millers it is a good idea to use the lowest cutting speed shown in the table.

MACHINE TOOL METALWORKING
 JOHN L. FEIDER AND EARL E. TATRO
 McGRAW-HILL 1961

employed, it will (1 × 3.1416). The minute is affected by important are the hined and the kind cutter. The cutting represent averages al of metal and at g life to the cutter. or burn the cutter. ing cuts can be as at higher than for machine speed to minute at which e following should

feet per minute)
cutter in inches)

Material	Speed Range (sfpm)
Tungsten carbide	1000-2000
	350-600
	200-425
	125-200
	250-325
	150-200
	100-200
	250-370

Best cutting speed shown

Suppose you want to machine a piece of mild steel (SAE 1025) with a high-speed steel cutter that is 6 inches in diameter. The cutting-speed range for a high-speed steel cutter used on mild steel is between 79 and 97. Let's select a cutting speed on the lower side of about 80 sfpm. Then,

$$\text{rpm} = \frac{f \times 80}{n} \quad \text{or} \quad \frac{320}{6} \quad \text{or} \quad 53.333$$

or about 53 or 54. The machine speed, or rpm, is adjusted in one of the following ways:

1. Most small bench and floor hand millers are belt-driven, just like a drill press. To change the simple speed you must change the position of the belt on the pulleys. A plate fastened to the side of the machine will tell you the machine speed when the belt is in different positions. Sometimes there is a back gear that can be engaged to give about eight possible machine speeds.
2. On larger milling machines, several methods can be followed for adjusting for speed:

- a. Some machines have two or three levers on the left side of the machine that can be moved to obtain the desired cutting speed.
- b. Other machines have one lever or knob and a speed-selector dial. When the lever or knob is turned, the machine speed will be indicated on the dial.

Feed

Feed is the rate at which the workpiece advances under the cutter. It is really the most important single factor in determining how fast metal can be removed from the workpiece. Actually, feed, plus depth of cut, plus width of cut, determines how many cubic inches of metal are removed from the workpiece in any given length of time. There are three ways feed is controlled:

1. *Manual feed.* On many small hand millers there is no power feed. Therefore you must depend on your own judgment about how fast to move the workpiece under the cutter. In general a slower feed is best for heavy roughing cuts and a faster feed for light, finishing cuts. Usually the tendency is to go too slow rather than too fast. A slow feed causes excessive wear on the cutter because a slow speed produces more rubbing than cutting action.

2. *Inches per revolution of the spindle or cutter.* On some machines the feed is directly related to the speed. As the speed increases, the feed increases. This feed arrangement is found on cone-driven machines. Some of the smaller hand millers have this feed arrangement.

3. *Inches per minute (ipm).* Most larger milling machines use a feed rate shown in inches per minute. The feed rate is independent of the speed. In other words, the feed is set at the desired amount in inches per minute. A change in the machine speed does not affect this feed.

The actual feed should be in terms of the following:

- a. The kind of material in the workpiece
- b. The kind of cutter
- c. The kind of material in the cutter
- d. The power available at the spindle (horsepower)
- e. The way the workpiece is held (how rigid the setup is)
- f. The shape and kind of workpiece (how rigid it is)

As a general rule, feeds should be as coarse as possible to obtain the desired finish. At the same time they must be fine enough to secure a long cutter life. The feed is generally reduced a little for the finish cut. A method commonly used to find the estimated feed rate in inches per minute is as follows:

$$F (\text{feed rate}) = f (\text{feed per tooth}) \times T (\text{number of teeth}) \times N (\text{rpm of cutter})$$

Table 76-2. RECOMMENDED FEED PER TOOTH FOR HIGH-SPEED STEEL CUTTERS

Material	Face mills	Spiral mills	Slotting and side mills	End mills	Form cutters	Saws
Aluminum. Soft bronze	0.022	0.017	0.013	0.011	0.006	0.005
Medium bronze. Cast iron, soft	0.018	0.014	0.011	0.009	0.005	0.004
Malleable iron. Cast iron, medium	0.015	0.012	0.009	0.008	0.005	0.004
SAE X-1112 steel. Cast iron, hard	0.013	0.010	0.008	0.006	0.004	0.003
SAE 1020 steel. SAE X-1335 steel	0.011	0.009	0.007	0.005	0.004	0.003
SAE 1045 steel. Cast steel	0.009	0.007	0.006	0.005	0.003	0.003
Alloy steel:	0.008	0.006	0.005	0.004	0.003	0.002
Medium	0.007	0.005	0.004	0.004	0.002	0.002
Tough	0.005	0.004	0.003	0.003	0.002	0.0015
Hard	0.006	0.005	0.004	0.003	0.002	0.0015
Alloy tool steel						

Table 76-2 gives the average value for f for high-speed steel cutters of different kinds when used on various materials. You will notice that this formula takes into consideration only the first three factors that should be considered in feed selection. In production milling, the feed is determined in a different way. A chart developed by manufacturers of milling machines shows the maximum rate at which metal can be removed in cubic inches per minute. This rate is determined by the rated horsepower of the machine and the kind of metal being machined. For example, with a 15-horsepower machine, soft steel can be removed at a rate of 7 cubic inches per minute. If the depth of cut and the width of cut are known, the feed (F) in inches per minute can be found as follows:

$$F = \frac{\text{maximum metal removal in cubic inches per minute}}{\text{depth of cut} \times \text{width of cut}}$$

When the feed (F) in inches per minute has been found, then the feed (f) per tooth can be found as follows:

$$f = \frac{F \text{ (feed in inches per minute)}}{\text{cutter rpm} \times \text{number of teeth on cutter}}$$

This feed per tooth (f) in inches can be compared with the suggested feed per tooth for milling different materials, as shown in Table 76-2, to discover if the feed is too fast or too slow. The experienced machinist then uses his judgment for adjusting the correct feed. In production milling, the feed rate is the important factor in getting the maximum amount of milling completed in a minimum length of time.

Unit 77. Work-holding Devices

There are many different devices for holding the workpiece to be machined. Many are similar to those used on the drill press and shaper. The following are important attachments for use on the milling machine:

1. The *plain vise* is the handiest and most common work-holding device. The vise can be fastened to the table with the jaws either parallel or at right angles to the T slots.
2. The *swivel vise* is the same as the plain

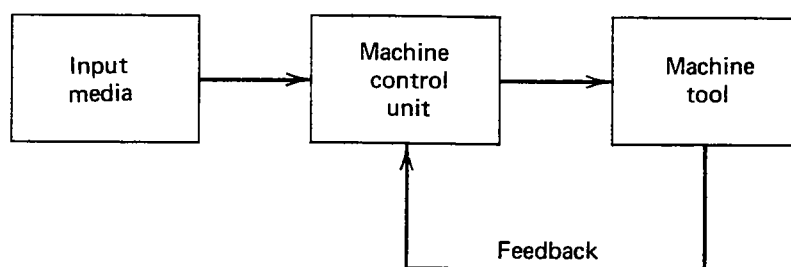


Figure 1.2 A simplified schematic of an NC system.

1.3 The Numerically Controlled Machine

We have defined NC as a method of automatic control that uses symbolically coded instructions to cause the machine to perform a specific series of operations. The most important of these instructions is that of *position*.

1.3.1 Positioning

Although modern NC systems perform many functions, the most important controlled operation is static or dynamic *positioning* of the cutting tool with the use of a system of coordinates that is general enough to define any geometric motion.

The right-handed Cartesian coordinate system, illustrated in Figure 1.3, provides a simple method for the definition of any point in three-dimensional space. While all NC machines make use of a coordinate system, some require only two axis (x and y) motion, and others require three-dimensional linear and angular axes. For the present purposes the 3-D Cartesian system is used.

Referring to Figure 1.3, the point P_1 is defined by the coordinate set $(1, 2, 2)$, where each planar subdivision represents one unit. Likewise, point P_2 is defined by the coordinate set $(3, 3, 4)$. The number of subdivisions within the three-dimensional space may be increased so that any point within the predefined boundaries may be described by a countable number of units.

Unlike a pure Cartesian system in which an infinite number of axial subdivisions is assumed, an NC coordinate system considers only a finite number of subdivisions. A later chapter will show that each *command* which is output by the control unit to the machine corresponds to one unit of motion. An NC machine is only as accurate as this smallest predefined subdivision. To illustrate this concept, consider a one-axis NC device which generates motion along the x -axis bounded by the values $x = -0.1$ m and $x = +0.1$ m. If the axis has 1000 predefined subdivisions, the machine could position to 0.2 mm. If 10,000 subdivisions were used, accuracy would increase tenfold to 0.02 mm. In these illustrations each command would correspond to 0.2 mm or 0.02 mm of motion.

NUMERICAL CONTROL AND COMPUTER-AIDED MANUFACTURING
 ROGER PRESSMAN AND JOHN E. WILLIAMS
 JOHN WILEY & SONS 1977

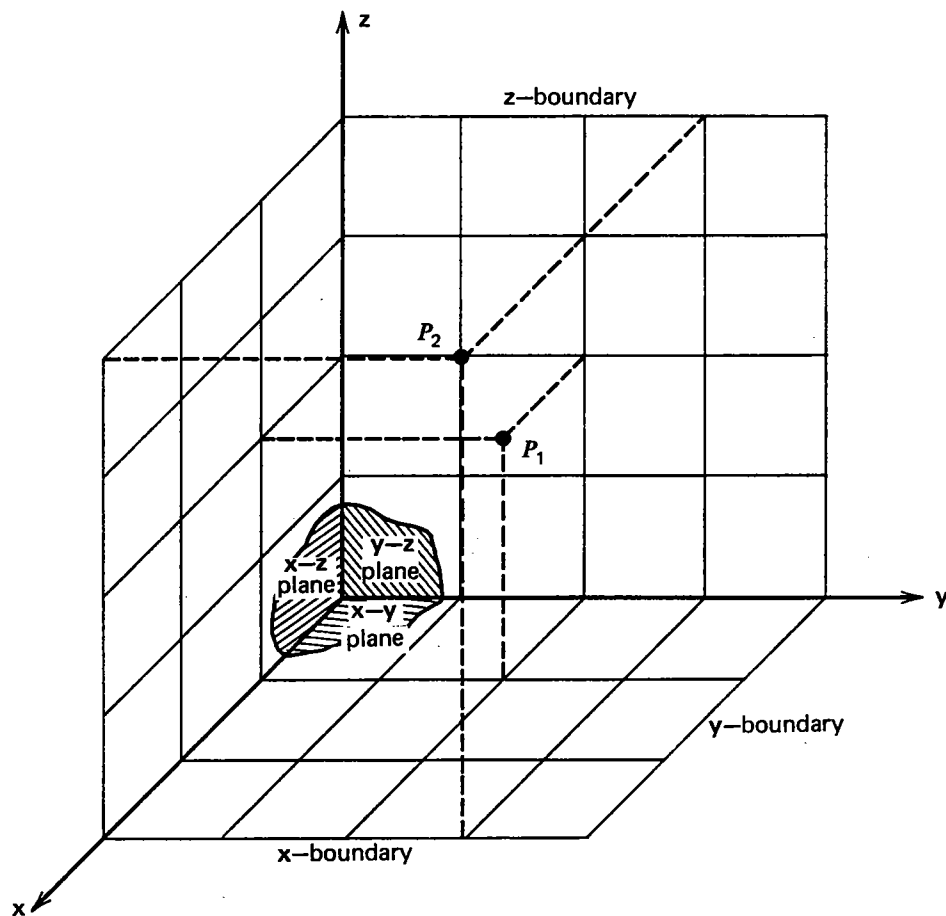


Figure 1.3 A bounded cartesian reference system.

An NC machine tool uses the coordinate system as a framework for positioning the cutting tool with respect to the workpiece. To accomplish this, points along the component profile* are defined by x , y , z coordinates. These coordinates are then fed in sequence to the NC controller which generates the appropriate positioning commands. A typical machine-axis configuration is illustrated in Figure 1.4. Any combination of spindle motion and/or work-holding table motion enables the proper position to be attained.

Positioning can be accomplished using two distinct methods. The first method, called *absolute positioning*, fixes the reference system and enables the actual x , y , z coordinates to be specified with reference to a fixed origin. Using an absolute positioning system, the points P_1 and P_2 would be specified as (x_1, y_1, z_1) followed by (x_2, y_2, z_2) , regardless of the cutter position before the command is issued.

The second method of positioning uses *incremental* movement to obtain the same result. In an incremental device, the reference system is relative to the

* In many cases *offset* points are required (see Chapter Seven).

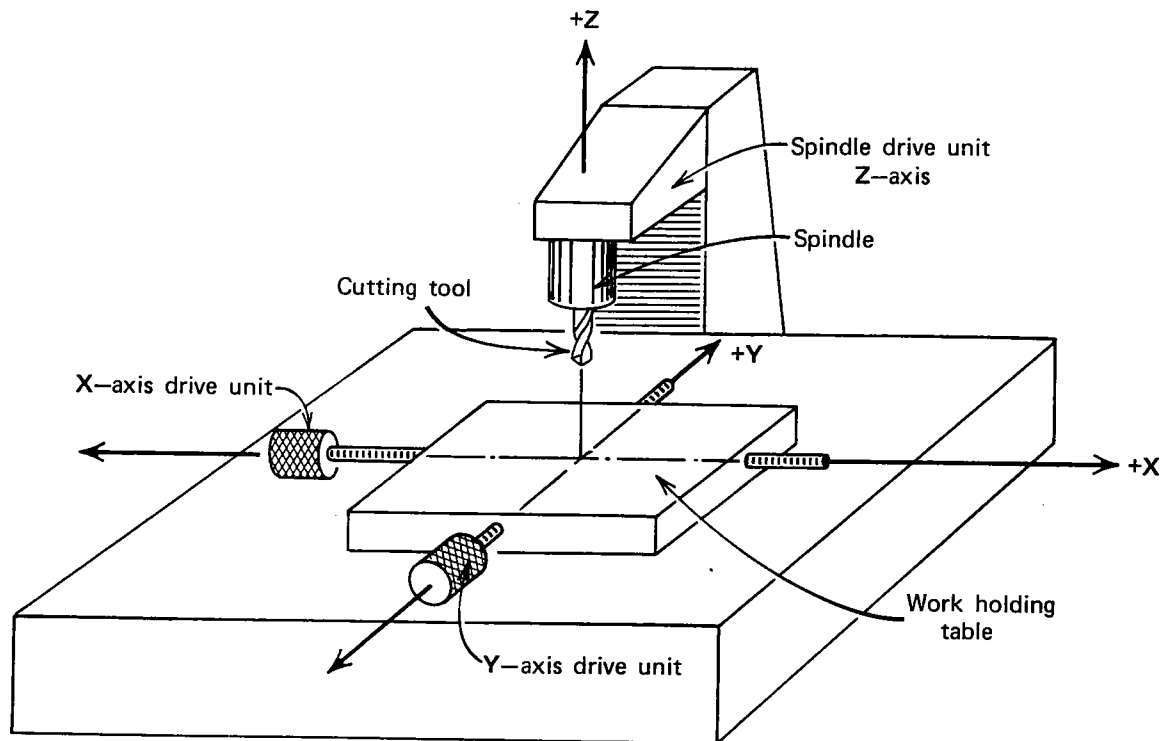


Figure 1.4 A typical NC machine axis system.

last position. For example, assume that the tool's current position is point $P_0(x_0, y_0, z_0)$. To specify a new position at P_1 , the incremental distances, rather than the absolute coordinate values, are given. Let $\Delta x_{10} = (x_1 - x_0)$, $\Delta y_{10} = (y_1 - y_0)$, $\Delta z_{10} = (z_1 - z_0)$. Then, the new position is obtained by $(\Delta x_{10}, \Delta y_{10}, \Delta z_{10})$, where the reference system is assumed to have its origin at P_0 . Similarly, a move from P_1 to P_2 is obtained by $(\Delta x_{21}, \Delta y_{21}, \Delta z_{21})$, where $\Delta x_{21} = (x_2 - x_1)$, etc. The incremental system therefore uses the change in x, y, z dimensions to specify position, whereas the absolute system uses coordinate values.

1.3.2 Control System

In our discussion of positioning, the path taken between points was disregarded. The path which the cutting tool follows as it traverses from point to point depends upon the type of control system used. Three basic path control systems are found in general usage.

The *point-to-point* system (also called a *positioning system*) effectively disregards the path between points. Each axis of motion is controlled independently so that the path steps from the start position to the next position as shown in Figure 1.5. The path shown is not unique as some point-to-point systems first satisfy the x command and then the y , whereas others reverse the

The Elements of an NC Part Program

Part programs generally incorporate two types of instructions: those which define the tool path (such as X, Y and Z axis coordinates), and those which specify machine operations (such as turning the spindle on or off). Each instruction is coded in a form the computer can understand.

An NC program is composed of *blocks* (lines) of code. The maximum number of blocks per program is limited by the memory (RAM) on your computer. You can, if necessary, chain programs together to form very large part programs.

Each block contains a string of *words*. An NC word is a code made up of an alphabetic character (called an *address character*) and a number (called a *parameter*). There are many categories of address characters used in NC part programs for the machining center (see Categories of NC Code).

Each block of NC code specifies the movement of the cutting tool on the machining center and a variety of conditions that support it. For example, a block of NC code might read:

N0G90G01X.5Y1.5Z0F1

If the machine is currently set for inch units, the individual words in this block translate as:

- N0** This is the block sequence number for the program. Block 0 is the first block in the program.
- G90** This indicates absolute coordinates are used to define tool position.
- G01** This specifies linear interpolation.
- X.5** This specifies the X axis destination position as 0.5".
- Y1.5** This specifies the Y axis destination position as 1.5".
- Z0** This specifies the Z axis destination position as 0". The cutting tool will move to the absolute coordinate position (0.5,1.5,0).
- F1** This specifies a feed rate of 1 inch per minute, the speed at which the tool will advance to the specified coordinate points.

Categories of NC Code

There are many categories of NC code used for programming. The following is a list of the NC codes (designated by the address character) supported by the BENCHMAN 4000.

Code:	Function:
%	Incremental Arc Centers (Fanuc).
\$	Absolute Arc Centers (LMC).
\	Skip.
/	Optional skip.
D	Compensation offset value.
F	Feed rate in inches per minute; with G04, the number of seconds to dwell.
G	Preparatory codes.
H	Input selection number; Tool length offset.
I	Arc center, X axis dimension (circular interpolation).
J	Arc center, Y axis dimension (circular interpolation).
K	Arc center, Z axis dimension (circular interpolation).
L	Loop counter; Program cycle (repeat) counter for blocks and subprograms; Specify homing tolerance.
M	Miscellaneous codes.
N	Block number (user reference only).
O	Subprogram starting block number.
P	Subprogram reference number (with M98); Uniform scale multiplier (with G51).
Q	Peck depth for pecking canned cycle.
R	Arc radius for circular interpolation (with G02 or G03); Starting reference point for peck drilling (with canned cycle codes).
S	Spindle speed.
T	Tool specification.
U	Incremental X motion dimension for absolute dimensioning.
V	Incremental Y motion dimension for absolute dimensioning.
W	Incremental Z motion dimension for absolute dimensioning.
X	X axis motion coordinate.
Y	Y axis motion coordinate.
Z	Z axis motion coordinate.
;	Comments.

Incremental Arc Center (%)

The incremental arc center code selects the Fanuc mode for programming arc coordinates. This mode is selected for the entire NC program as well as for any chained programs.

In the Fanuc mode, arc centers are always incremental, regardless of whether the system is in G90 (absolute) or G91 (incremental) mode. In contrast, arc center specifications in EIA-274 mode follow the selected programming mode, absolute or incremental.

You can specify the default arc center mode in the Run Settings dialog box.

This character must stand alone on the first line of the NC program in which it appears.

Absolute Arc Centers (\$)

The absolute arc center code selects the EIA-274 mode of programming arc coordinates. This mode is selected for the entire NC program as well as for any chained programs.

In the EIA-274 mode, the mode of programming arc centers follows the selected programming mode; absolute (G90) or incremental (G91). In contrast, arc center specifications in Fanuc mode are always incremental, regardless of whether the system is in absolute or incremental mode.

You can specify the default arc center mode in the Run Settings dialog box.

This character must stand alone on the first line of the NC program in which it appears.

Skip (\) and Optional Skip (/)

The Skip and Optional Skip codes allow you to skip particular lines of code in your program.

To use the Skip code (\):

Place the code at the beginning of the line you wish to skip. When you run the NC program, the specified line will be skipped.

To use the Skip code (\) with a parameter:

Use the Skip code with a parameter to instruct the Control Program to execute the line of code every nth pass. Place the code at the beginning of the line you wish to skip.

Note: The optional skip (/) code works only when the Optional Skip parameter from the Run Settings dialog box is on.

The syntax is: \n, where n is the number of passes between executions.

For example, if you want to execute a block of code every 5 passes, place \5 as the first code at the beginning of the block.

To use the Optional Skip code (/):

1. Place the code at the beginning of the line you wish to skip.
2. Select the Optional Skip option from the Run Settings dialog box or the Operator Panel.

When you run the NC program, the specified line will be skipped. If you do not select the Optional Skip option in the Run Settings dialog box, the skip code is ignored and the line is executed normally.

To use the Optional Skip code (/) with a parameter:

Use the Optional Skip code with a parameter to instruct the Control Program to execute the line of code every nth pass. Place the code at the beginning of the line you wish to optionally skip.

The syntax is: /n, where n is the number of passes between executions.

For example, if you want to execute a block of code every 5 passes, place /5 as the first code at the beginning of the block.

Compensation Offset Value (D Code)

The D code is used to select a value from the Control Program Offset Table. For example, D1 selects entry number 1 from the Offset Table.

Use the D code with:

- ◆ Cutter compensation codes to specify the tool radius.
- ◆ Tool offset adjust codes to specify a consistent increase or decrease in the commanded movement.

Use the Offsets command under the Setup Menu to view and manage the Offsets Table.

Feed Rate (F Code)

Use the F code to:

- ◆ Specify the rate of speed at which the tool moves (feed rate) in inches per minute (ipm). For example, F3 equals 3 ipm.

The feed rate should be set to a low value (up to 50 ipm) for cutting operations. Feed rate values are in millimeters per minute (mpm) when using metric units. The Control Program limits the programmed feed rate so it doesn't exceed the maximum allowed by the machining center.

- ◆ Specify the number of seconds to dwell when used with the G04 code.

Preparatory Codes (G Codes)

G codes take effect before a motion is specified. They contain information such as the type of cut to be made, whether absolute or incremental dimensioning is being used, whether to pause for operator intervention, and so on.

More than one G code from different groups can appear in each NC block. However, you may not place more than one G code from the same group in the same block.

The G codes supported by the Control Program fall into the following groups:

- ◆ The Interpolation Group
- ◆ The Units Group
- ◆ The Plane Selection Group
- ◆ The Wait Group
- ◆ The Canned Cycle Group
- ◆ The Programming Mode Group
- ◆ The Preset Position Group
- ◆ The Compensation Functions Group
- ◆ The Coordinate System Group
- ◆ The Polar Programming Group

Note: More than one G code from different groups can appear in each NC block. However, you may not place more than one G code from the same group in the same block.

The Interpolation Group

The interpolation group allows you to specify the type of motion for interpolation. These G codes are retained until superseded in the NC program by another code from the interpolation group.

The supported interpolation G codes are:

- G00 Rapid traverse
- G01 Linear interpolation (default)
- G02 Circular interpolation (clockwise)
- G03 Circular interpolation (counterclockwise)

The Units Group

By default, an NC program is interpreted using the units of measure (inch or metric) specified using the Units command on the Setup Menu.

The codes in the Units group, G70 (inch) and G71 (metric), are used to override the Units command, for the entire program, or for a single line.

If the code is placed at the beginning of the program before any tool motions are made, that unit of measure is assumed for the entire program. If the code appears in a block of code, the unit of measure is in effect for that block and all following blocks. You can use these codes to switch between inch and metric modes throughout your program at your convenience.

The Fanuc equivalents, G20 (inch) and G21 (metric), can also be used.

The Plane Selection Group

This group of codes allows you to select different planes for circular interpolation. G17 is the Control Program default.

The supported Plane Selection Group codes are:

- G17 Select the X, Y plane for circular interpolation. The arc center coordinates are given by I for the X axis and J for the Y axis.
- G18 Select the X, Z plane for circular interpolation. The arc center coordinates are given by I for the X axis and K for the Z axis.
- G19 Select the Y, Z plane for circular interpolation. The arc center coordinates are given by J for the Y axis and K for the Z axis.

The Wait Group

Wait Group codes apply only to the block in which they appear. The program does not continue until the wait conditions are satisfied.

The supported Wait Group codes are:

- G04** Dwell (wait): Stop motion on all axes for the number of seconds specified by the F code, then continue the program. Because the F code is used to specify the number of seconds, you cannot also specify a new feed rate in the same block.
Example: G04F10; Wait for 10 seconds
- G05** Pause: Used for operator intervention. Stop motion on all axes until the operator manually resumes program execution.
- G25** Wait until TTL input #1 goes low before executing the operations on this block. Used for external device synchronization.
- G26** Wait until TTL input #1 goes high before executing the operations on this block. Used for external device synchronization.
- G31** Linear move to specified coordinate; used with H code to specify both the input number and the High or Low condition for stop (designated by the input operator, + or -). The move occurs until an input is triggered or until a coordinate is reached. The move stops short if specified input goes High (if H is positive) or Low (if H is negative). The default is input 1 High.
You can have the control program go to a specified block (N Code number) if the input meets the required condition. Use a P code to specify the destination, as with the M98 code.
For example, G31X5Y5H-2P50000 instructs:
Move (using the current programming mode) to the X and Y given.
If input 2 goes low during the move, jump to block number 50000.
If input 2 doesn't go low, continue with the next block in the program.
- G35** Wait until TTL input #2 goes low before executing the operations on this block. Used for external device synchronization.
- G36** Wait until TTL input #2 goes high before executing the operations on this block. Used for external device synchronization.
- G131** Specifically for digitizing with the probe. The user specifies a Z position and a feedrate. The probe moves from its current position to the specified Z position at the specified feedrate (or current feedrate if not specified on the same block). If the probe input is tripped before reaching the specified Z position, a valid point is captured. In either event (point or no point), when the probe stops moving down, it rapids back to the initial Z position.

The Canned Cycle Group

Canned cycle codes allow you to perform a number of tool motions by specifying just one code. Canned Cycle codes are typically used for repetitive operations to reduce the amount of data required in an NC program. Canned cycle codes are retained until superseded in the program by another canned cycle code.

The supported Canned Cycle codes are:

- G80 Canned cycle cancel
- G81 Canned cycle drilling
- G82 Canned cycle straight drilling with dwell
- G83 Canned cycle peck drilling
- G85 Canned cycle boring
- G86 Canned cycle boring with spindle off (dwell optional)
- G89 Canned cycle boring with dwell

Refer to Section G for more information on these functions.

The Programming Mode Group

Programming mode G codes select the programming mode, absolute (G90) or incremental (G91). These codes remain in effect until superseded by each other. The default code on program start up is G90.

With absolute programming, all X, Y and Z coordinates are relative to origin of the current coordinate system. With incremental programming, each motion to a new coordinate is relative to the previous coordinate.

The supported Programming Mode codes are:

- G90 Absolute programming mode
- G91 Incremental programming mode

The Preset Position Group

The preset position G codes move the tool to a predetermined position, or affect how future motions will be interpreted.

The supported Preset Position codes are:

- G27 Check reference point: This code moves the machine to its home position and compares the reported position against zero to see if any position has been lost. The difference between the reported position and zero is compared to a tolerance value specified using the Setup Program. Use the L code in this block to override the tolerance value in the Setup Program.
- G28 Set reference point: This code moves the machine to its home position and sets the machine position to 0,0,0. The G28 code performs an automatic calibration of the axes.

- G92 Set position: This code works like the Set Position command under the Setup Menu. The X, Y and Z coordinates following a G92 code define the new current position of the tool.
- G98 Rapid move to initial tool position after canned cycle complete.
- G99 Rapid move to point R (surface of material or other reference point) after canned cycle complete.

The Compensation Functions Group

Use the cutter compensation NC codes to automatically compensate for the variations in a cutting tool's radius and length. Refer to Section H for more information on using cutter compensation.

The supported Compensation codes are:

- G39 Corner offset circular interpolation.
- G40 Cancel cutter compensation.
- G41 Left cutter compensation: Enables cutter compensation to the left of programmed tool path.
- G42 Right cutter compensation: Enables cutter compensation to the right of programmed tool path.
- G43 Tool length offset: Shifts Z axis in a positive direction by a value in the Offset Table, specified by an H code.
- G44 Tool length offset: Shifts Z axis in a negative direction by a value in the Offset Table, specified by an H code.
- G45 Tool offset adjust: Increases the movement amount by the value stored in the offset value memory.
- G46 Tool offset adjust: Decreases the movement amount by the value stored in the offset value memory.
- G47 Tool offset adjust: Increases the movement amount by twice the value stored in the offset value memory.
- G40 Tool offset adjust: Decreases the movement amount by twice the value stored in the offset value memory.
- G49 Cancels tool length offset.
- G50 Cancels scaling.
- G51 Invokes scaling.
- G68 Invokes rotation.
- G69 Cancels rotation.

