TEKTRONIX®

1704

MACHINE CONTROL UNIT

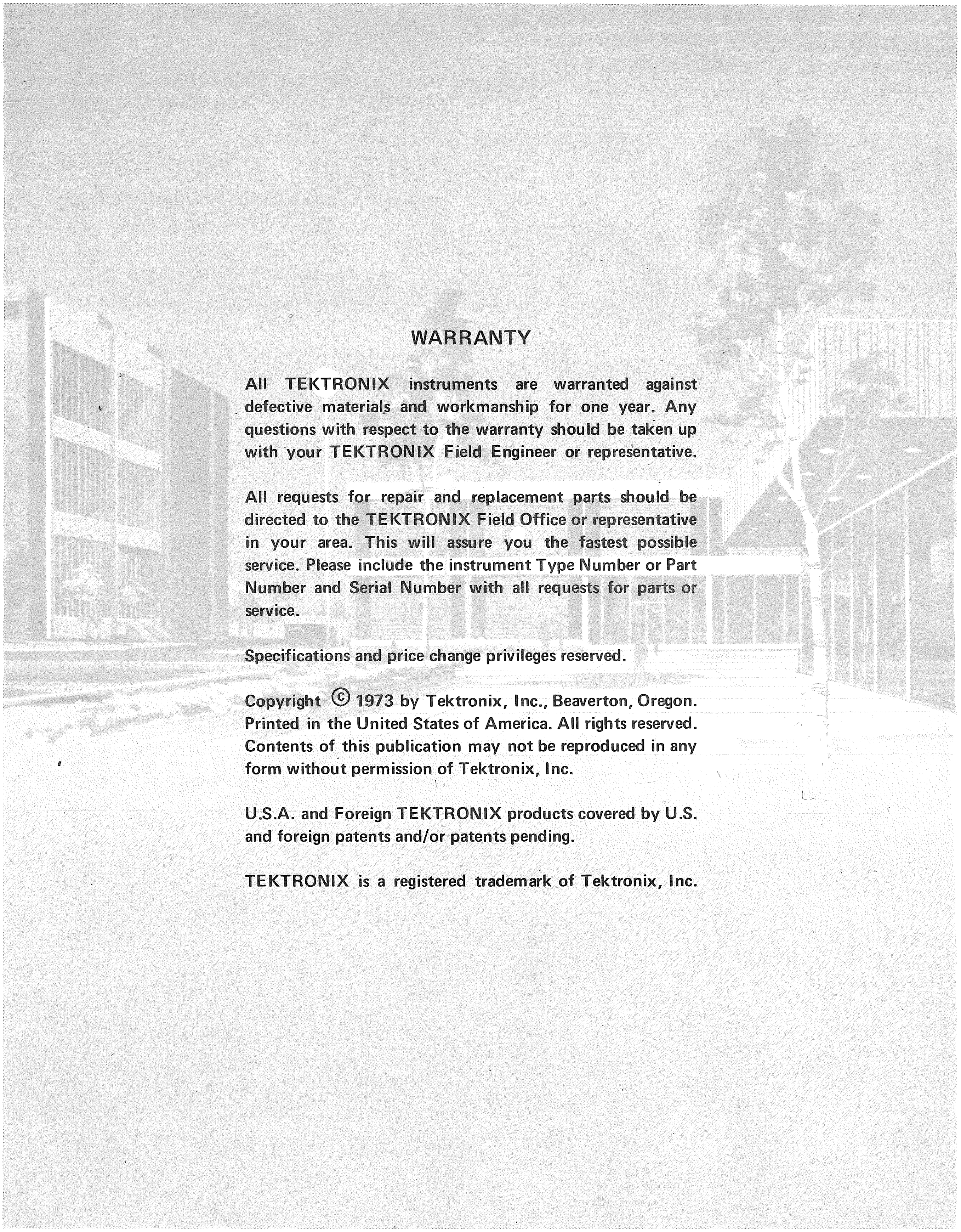
PROGRAMMER'S MANUAL

Tektronix, Inc.
P.O. Box 500
Beaverton, Oregon 97005

Serial Number ____________________

070-4107-00

1273



WARRANTY

All TEKTRONIX instruments are warranted against defective materials and workmanship for one year. Any questions with respect to the warranty should be taken up with your TEKTRONIX Field Engineer or representative.

All requests for repair and replacement parts should be directed to the TEKTRONIX Field Office or representative in your area. This will assure you the fastest possible service. Please include the instrument Type Number or Part Number and Serial Number with all requests for parts or service.

Specifications and price change privileges reserved.

TABLE OF CONTENTS

Fig. 1-1. Tektronix 1704 Machine Control Unit.

INTRODUCTION

General

This manual contains information necessary for programming the Tektronix Model 1704 two and three axis contouring controls. It contains the necessary rules pertaining to the full floating zero and setup points, plus and minus programming, and the coordinate system. A detailed explanation of the tape format is included, as well as information pertaining to EIA and ASCII character coding. The information and instructions in this manual are for three axis controls. For two axis applications, the same techniques are applicable by eliminating all reference to Z axis motion.

Due to the versatility of the Tektronix Model 1704 contouring controls, there are many approaches to each machining operation. This manual describes a few basic techniques. As the programmer becomes more familiar with the capability of the control and machine, he will devise new and more economical methods suited to each individual application.

Control Features

Integrated Circuit Logic System for Maximum Reliability. All Tektronix controls described in this manual are buffered, complete contouring systems. With the extensive use of integrated circuits, a uniquely compact package has been used (see Fig. 1-1). In all standard applications the control, tape reader and operator's station are in the one compact unit.

Four-Digit Inch-per-minute Feedrate Programming. The feedrate number is programmed directly in inches per minute for either tape input or manual data entry. The feedrate is a 4-digit F word (Fxxx.x) allowing the programming from feedrates of .1 inch per minute up to 200.0 inches per minute, or the maximum rate of the machine tool, whichever is less. Optional feedrate cpabilities are found in Section 4.

Automatic Acceleration and Deceleration. This feature permits the programmer to change the feedrate between two blocks of tape without special consideration of G codes. To decrease machine cycle time, automatic deceleration can be inhibited by a G code in cases where potential machine slide overshoot would not affect part tolerance.

Mirror Image Controls. A selector switch for each axis allows the operator to choose mirror image operation so that right- and left-hand parts can be machined from the same tape.

Feedrate Override. Feedrate override operates with programmed or manual data input feedrates. This allows override from 125% of programmed feedrate down to 12.5% in 12.5% steps, with an OFF position below the 12.5% setting. Feedrates programmed at 200 IPM (maximum) cannot be overridden above 100%.

Absolute/Incremental Dimensioning. The programmer and operator can use either absolute or incremental numbers eliminating unnecessary addition and subtraction of numbers, according to part print dimensions, to enter correct input values into the control. The standard dimensioning format of the control is in absolute values. Entry of a G91 code allows the input of incremental dimensions. A G90 code re-establishes absolute format.

Full Range Floating Zero. A full range floating zero shift is provided for each axis by means of jog or manual data input.

Manual Data Input (MDI). With MDI, all programmed data can be entered into the control. In addition, slide position data can be entered.

For additional control features and operation, see Machine Control Unit 1704 User's Manual, Tektronix Part Number 070-1184-00.

Coordinate System

Commands that direct machine slide movements to a precise position are planned in accordance with a three-axis coordinate system (see Fig. 1-2). Coordinates are expressed as absolute dimensions having either a positive or negative value relative to the zero point. The longitudinal travel of the carriage normally defines the X axis. The transverse travel of the carriage or spindle head assembly defines the Y axis. The spindle axis travel defines the Z axis.

Generally, the zero is selected at a line on the part print from which surface dimensions originate. This method minimizes coordinate calculations for points symmetrical about one line or plane. The zero may occur outside the slide travel capacity. For correct coordinate planning the planner should think in terms of moving cutter position as related from the zero position on a fixed workpiece. In essence, the coordinate system is superimposed on the part print.

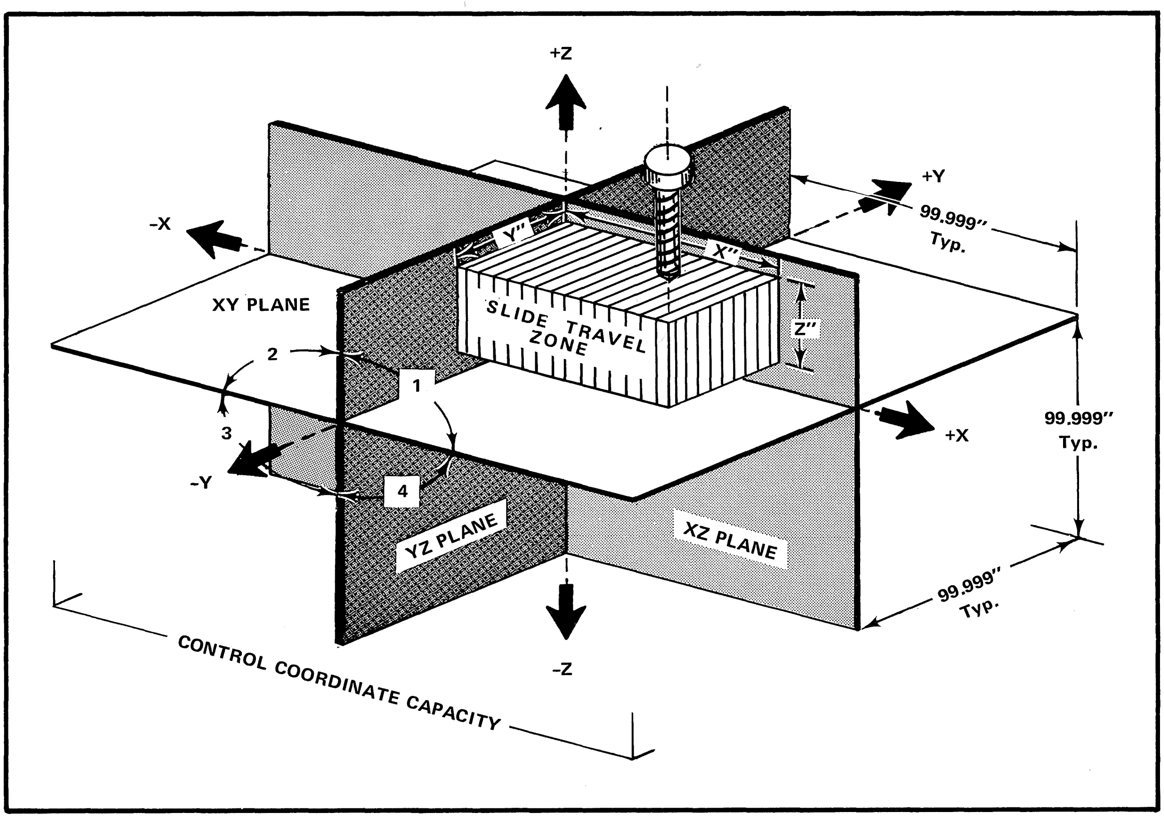


Fig. 1-2. Part Coordinate System, showing the proportional capacities of the coordinate axes and slide travel for a typical three-axis machine. The control capacity for the zero point, arcs and slopes is ± 99.9999".

Figure 1-2 illustrates that the slide travel zone may be synchronized with the part coordinate system, in any combination of octants, as desired by the planner. Generally, the zero for part applications will fall within the slide travel capacity.

Coordinate Synchronization

The full floating coordinate system is synchronized with the slide travel by a setup location for each axis. Three setup coordinates are defined by the planner, each having an end point within the respective X, Y, and Z slide travels. These coordinates are the initial entries on the process planning sheet. In order to synchronize the coordinate system, a good practice is to setup from the part fixture locating surfaces. Initial setup is made by manual adjustment after the operator starts the machine.

Axis Inversion

Inversion of the X, Y, and Z axes is an optional feature of the Model 1704 controls. The use of axis inversion is beneficial when "Mirror Image" work is to be done. The inversion is accomplished by use of a selector switch associated with the axis. The machine operator must be directed as to the use of these selector switches.

Programming Manuscript

Instructions that define, in sequence, the desired machine functions to accomplish workpiece operations are recorded on the Programming Manuscript (see Fig. 1-3). The order of columns, from left to right, beginning with the Sequence Number (N), is the recommended sequence for the information to be entered to ensure that no data is omitted for each operation, although the word address format on the tape does not have a sequential restriction. Information included in each column must be punched on tape. The arc center, clearance plane, spindle and tool number columns are provided for controls with circular interpolation, drill cycles, spindle speed and tool selection options respectively. For controls without one or more of these options, these columns may be ignored.

BCD Tape Coding

The control uses 8-channel, 1 inch wide tape whose dimensions and coding are in accordance with EIA or ASCII standards. All numerical tape data is in the form of Binary Coded Decimal (BCD). For proper operation of the photo-electric tape reader, it is recommeded that only opaque mylar or black paper tape be used. Translucent paper tape is not recommended.

Tape Structure

The following explanations pertain to the general tape structure.

Character. A character is a set of punched holes that represent a digit, letter, sign, punctuation mark, etc.

Row. A row is a path perpendicular to the edge of tape where the holes for one character are located.

Channel. A channel is a path parallel to the edge of the tape that can contain either a hole or no hole depending on the characters. There are eight channels. The first three channels are separated from the last five by a series of smaller holes known as sprocket holes. Sprocket holes contain no information; they are used as a timing track for reading data from tape.

TAPE AND PART PROGRAM STRUCTURE
MODEL 1704

SEQ NO. N	PREP FUNC G	AXIS X	AXIS Y	AXIS Z	ARC CENTER (X) I	ARC CENTER (Y) J	ARC CENTER (Z) K	CLEARANCE PLANE	SPNDL S	TOOL NO T	MISC FUNC M	FEED RATE F

Fig. 1-3. 1704 Programming Manuscript Worksheet.

Word. A word is a combination of Characters that represents an axis coordinate command, a sequence number, a feedrate, a spindle speed, or a G or M function. The first character of each word is the letter address code; some examples follow:

G00	N06
X–200000	Y13471
F1241	S04

Block. A block is a combination of words, arranged in accordance with the specified format, that represents one complete sequence of commands. Each block of data must end with the End Of Block (EOB) code. The EOB character is punched by pressing the carriage return. The first block in the tape must be preceded by a Rewind Stop Code (RSC, % sign in ASCII) and an EOB.

EIA, ASCII and ISO Character Coding

The control is designed to accept control tapes whose characters are coded in accordance with either EIA or ASCII standards (see Fig. 1-4). The choice of the EIA or ASCII coding is provided by a two position selector switch. The machine operator must be instructed as to which coding is being used for a particular control tape.

The American Standard Code of Information Interchange (ASCII) is designed to achieve coding uniformity for information interchange between data processing and communication systems. The increased use of computers and related equipment for control tape preparation has demonstrated a need for such a code.

The ISO coding, commonly used in Europe, can be read in the ASCII position of the selector switch.

Tape Errors

The control monitors the input tape for four possible tape errors. Detection of one of these errors lights the read error light on the Operator's Console and interrupts the automatic cycle. These errors are as follows:

1. Parity Error—Each character of a control tape is monitored for the correct parity. In the case of EIA character coding, a character with correct parity has an odd number of holes.

   A correct ASCII character has an even number of holes. (The sprocket hole is not considered in parity.)

2. Incorrect sprocket hole spacing.

3. Missing sprocket hole.

4. Incomplete chad removal.

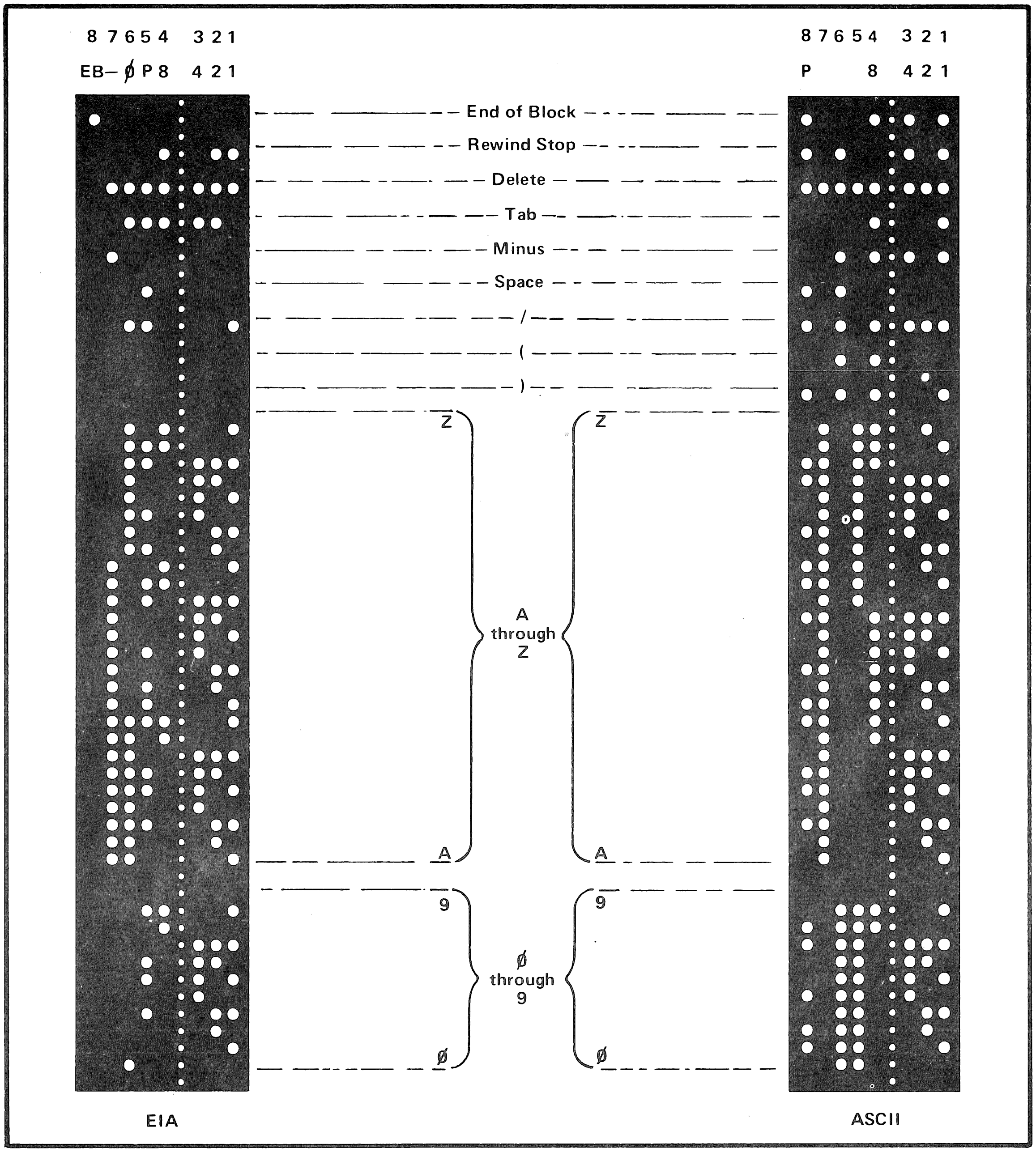


Fig. 1-4. EIA (left) and ASCII (right) coded tapes as used in the 1704.

PROGRAMMING DATA

General Input Format

Tape Format Definition. The Tektronix Model 1704 controls use a word address, tab ignore, variable block length format, which minimizes the amount of data in a tape. The use of an alpha character to denote each function or coordinate results in a more flexible block format. Once a word is programmed, it need not be re-entered and stays in force until changed by another word of the same class.

Words are entered on the process planning sheet in accordance with the following detail:

N3	:G2	:X± 2.4	:Y± 2.4	:F3.1	:M2	*

ALL LETTERS	—The letter address code of that particular word.
N3	—3 digit sequence number.
:	—Denotes where tab appears between words (if used).
G2	—2 digit preparatory function.
X±2.4	—X axis coordinate in inches (requiring the use of a minus sign) 2 digitis before and 4 digits after the implied decimal point.
Y±2.4	—Y axis coordinate, refer to X.
Z±2.4	—Z axis coordinate, refer to X.
F3.1	—Feedrate in IPM, 3 digits before and 1 digit after the implied decimal point.
M2	—2 digit miscellaneous function.
*	—EOB character.

NOTE

A detailed explanation of the format words is contained in this section.

Sequence Number (N3). This three digit number may indicate the block of tape being performed or may, at the programmers option, be used to indicate operation number, tool number, hole number or may be omitted entirely. The sequence number is used as a program reference and does not affect the function of the control.

The three digit number is programmed with the address N. Leading zeros may be omitted.

Preparatory Functions (G2)

The Tektronix contouring controls incorporate four standard preparatory or G Functions which determine the mode of operation of the control. Additional G Functions used with optional control features, which greatly reduce the amount of required input data, are also listed in this section. A summary of preparatory functions, for quick reference, can be found at the rear of this section. A detailed explanation of each standard G Function is found in the following paragraphs.

NOTE

Two different G words must not be programmed in the same block.

G00 Linear Interpolation With Full Deceleration. All linear motion command by the control is linearly interpolated; that is, the cutter moves along a line that can be drawn from the starting point to the programmed coordinates of the end point. The starting point of the slope is defined by the X, Y, and Z (if applicable) positions of the previous block. The coordinates of the slope end point are entered along with G00. The G00 causes the slide to decelerate and come to a full stop before the next block of tape is read.

The full stop mode caused by G00 remains in effect until cancelled by the G01 preparatory functions.

Requirements:

a. The start and end points of a slope must be within a 199.9998 inch cube.

b. The 199.9998 inch cube must be within the coordinate capacity of ± 99.9999 inches.

c. G00 should be used whenever the change in direction of the next block exceeds 30°.

d. G00 must be in effect for the last move in the tape (to the final programmed position).

e. Linear interpolation provides complete control of the tool path and feeds up to 200.0 IPM.

f. Requirements (a) Through (d) are control requirements. The practical requirements of these plus (e) are the machine tool travel and feedrate capacity.

G01 Linear Interpolation with Deceleration Override. This preparatory function causes the same machine tool responses as G00 above, except that the full stop mode is overridden. The G01 Function allows the slides to continue motion during reading of the next block of tape. This preparatory function provides a time saving feature, as the control will input the next block of tape and begin motion to the coordinates while the axes are approaching the commanded position of the previous block. For changes in direction, from block to block, this will result in a slight arc motion of the tool during this tape-read period. This feature is particularly useful when using linear interpolation to cut curved surfaces, as the transition point from one line segment to the next will be smoothed by the slight arc motion.

The mode caused by this preparatory function will remain in effect until cancelled by the G00 preparatory function.

Ⓐ

Requirements:

Requirements (a), (b) and (e) under G00 also apply to G01.

a. G01 should be used whenever curved paths are cut with linear interpolation.

b. G01 should not be used whenever the change in direction of the next block exceeds 30°.

c. G01 must not be in effect in a block commanding terminal motion; that is, whenever slide motion is to come to a full stop. This would cause overshoot of the machine slides and possible loss of control synchronization.

G02 (G20)—Arcs, Clockwise, and G03 (G30)—Arcs, Counterclockwise. (Optional). The cutter follows the path of an arc in the direction defined by G02 (G20) and G03 (G30) as viewed from the cutter onto the plane selected for arc. The arc to be followed is defined by the starting point, the end point, and the arc center (I, J, and K coordinates).

NOTE

G01, G02 and G03 must not be programmed in the block preceding a Head Down or a Head Up command.

Requirements:

a. The arc and its center is confined to a 199.9998 inch square in the XY, YZ, or ZX planes.

b. The 199.9998 inch square must be within the control coordinate capacity of ± 199.9998 inches.

c. Maximum radius is 99.9999 inches.

d. Requirements (a), (b) and (c) are control requirements. In order to machine the arc, the entire arc must be within the X, Y, and Z travel range of the machine. The arc center may be outside the machine travel if all other requirements are met.

e. The G02 and G03 Preparatory Functions maintain the deceleration override mode during circular interpolation. If deceleration is desired at the end of the circular path, G20 or G30 must be used.

f. For a circular tool path that terminates with no subsequent motion or with a change of direction greater than 30°, G20 or G30 must be in effect.

G94 Preset to Zero. This function is used to establish a revised coordinate position on the machine from tape. A block with G94 will not move the machine slides but re-set the position value of the machine at zero. The most common use for G94 is for multiple part setups on the same table. G94 is effective only in the block programmed and does not have to be cancelled.

G CODE	PREPARATORY FUNCTIONS SUMMARY
00	Linear Interpolation, X, Y, and Z with Full Stop Mode.
01	Linear Interpolation, X, Y, znc Z Axes with Full Stop Mode Override.
*02	Circular Interpolation—Arcs, Clockwise—No Deceleration.
*03	Circular Interpolation—Arcs, Counterclockwise—No Deceleration.
*20	Circular Interpolation—Arcs, Clockwise—Deceleration at end point.
*30	Circular Interpolation—Arcs, Counterclockwise—Deceleration at end point.
50	Rapid Traverse.
91	Incremental Mode.
94	Preset to Zero.

*Indicates optionally available Preparatory Functions.

Coordinate Information

All dimension words consist of an address and the command number. The address designates the numeric information to a specific axis. The numbers specify an absolute position accurate to four places to the right of the decimal. The coordinate dimension can be up to six digits, depending on the machine range. The commands of X012345 or X12345, both specify the dimension of 1.2345 inches in the X axis and both are acceptable by the control system.

The use of absolute input eliminates the addition and subtraction of numbers to obtain the correct input data to the control. Usually, fewer errors are made when information can be put into the control or on tape in actual coordinate values. Incremental values can be used by first setting the control to incremental mode by programming a G91. The coding of the coordinate information must contain trailing zeros. However, leading zeros may be omitted.

Feedrate

The feedrate number is programmed directly in inches per minute. The feedrate is a four digit F word (Fxxx.x) allowing the programmer to express feedrates in steps of .1 inch per minute up to 200.0 IPM or the maximum rate obtainable by the machine tool. The coding of this number must contain trailing zeros. However, leading zeros may be omitted. Refer to Feedrate, Section 3, for description and coding of the feedrate word.

Miscellaneous Functions (M2)

The control incorporates Miscellaneous or M Functions to activate and deactivate certain system devices. The sequence letter (A), designated with the corresponding M Function, indicates the function called will be active (A) or executed as soon as the block of tape is read. The sequence letter (D), designated with the corresponding M Function, indicates the function will be delayed (D) until all other commands in the block have been completed. A summary of the Miscellaneous Functions, for quick reference, can be found at the rear of this section. A detailed explanation of each M Function is found in the following paragraphs.

NOTE

Two different M words must not be programmed in the same block. No M code, other than a program stop can be programmed in or direclty after a block that is executed in G01, G02, or G03 modes.

M00 Programmed Stop (D). The programmed stop occurs after any motions programmed in the same block have been completed. All motions stop after completion of all commands in the block. The automatic cycle is resumed when the operator presses the CYCLE START button.

M01 Optional Stop (D). The optional stop function is identical in performance to the programmed stop. However, the optional stop command will not function unless the operator has pushed the optional stop switch on the console. The optional stop is used whenever the programmer requires the automatic cycle to stop at the option of the operator. The automatic cycle is resumed when the operator presses the CYCLE START button.

M02 End of Program (D). This function stops motion after completion of all commands in the block. This function signifies completion of the workpiece. It causes the forward tape reading to stop but will not rewind the tape to the rewind stop code (EOR). M02 is suggested for use when short "looped" tapes are used, thus eliminating the need for spoolers.

M03 Spindle on Clockwise (A). This function starts the spindle rotation in a clockwise direction as viewed from the machine spindle onto the workpiece, so as to advance a right-handed screw into the workpiece. The spindle rotates at the speed stored in the spindle speed register if the S Function option is available: otherwise spindle speed is selected manually. Rotation commences as soon as the tape is read.

M04 Spindle on Counterclockwise (A). This function will start the spindle rotation in a counterclockwise direction as viewed from the machine spindle onto the workpiece, so as to retract a right-handed screw from the workpiece. The spindle will rotate at the speed stored in the spindle register (or at the manually selected speed). Rotation will commence as soon as the block is executed.

M05 Spindle Stop (D). This function stops the spindle rotation after completion of all commands in the block.

M06 Tool Change (A). This function will be acted upon as soon as the tape is read. The following is a typical sequence for a machine tool that has automatic tool change cycle.

a. Spindle rotation and coolant flow will be interrupted.

b. The Z axis will rapid traverse to the tool change position.

c. The X and/or Y axis will rapid traverse to the transfer position (depending on the machine tool).

d. The tool transfer arm will exchange the tool in the machine spindle with a new tool from the tool magazine. The arm will return to home position and the cycle control is ready for a new command.

For machines without automatic tool changing, M06 can be used to alert the machine operator for a manual tool change. In this case, M06 operates identically to M00 except the "TOOL CHANGE" indicator on the operator's console will light.

M07 Option-Mist Coolant ON (A). This function selects and turns the mist coolant on if the machine is equipped with this option. The coolant will be turned on as soon as the tape is read.

M08 Flood Coolant ON (A). This function selects and turns the flood coolant on. The coolant will be turned on as soon as the tape is read.

M09 Coolant OFF (D). This function turns either mist or flood coolant off after completion of all commands in the block.

M30 Tape Rewind (D). This function rewinds the tape during completion of all commands in the block. Otherwise, M30 operates identically to M02.

NOTE

M10 through M29 and M31 through M99 are accessible for use at user's option.

TABLE 2-1

Miscellaneous Functions Summary

M Code	Function	Sequence
M00	Programmed Stop	D
M01	Optional Stop	D
M02	End of Program	D
M03	Start Spindle Clockwise	A
M04	Start Spindle Counterclockwise	A
M05	Spindle Stop—Coolant Off	A
M06	Tool Change	A
M07	Option—Mist Coolant On	A
M08	Flood Coolant On	A
M09	Coolant Off	A
M30	Tape Rewind	D

Plus and Minus Programming

By adding + or — signs to the dimension for motion words it is possible to program motion in all eight octants. If no sign is given it is assumed as +. The sign must be placed after the letter address and before the numeric value (X—123456).

Tape Coding

Tab Codes. Although tab codes are not required for operation of Tektronix Controls, they may be used to make the printout of tape data more readable when making a listing on a tape-reading typewriter. If tab codes are used, they must occur prior to the letter address for each word.

Code Delete. The Tektronix Controls will ignore deleted codes for both EIA and ASCII coding. With EIA coding all characters except End Of Block (Track 8) may be deleted. All characters in ASCII may be deleted.

Rewind Stop. The Rewind Stop code (% in ASCII) is used to terminate tape rewinding activated by M30 (tape rewind word). The Rewind Stop code should appear prior to the first block of data on the tape.

End-Of-Block. The End-Of-Block character (carriage return) is used to indicate the end of each information block. This code must precede the first block of information on the tape.

Decimal Point. Placement of the decimal point is fixed by format for motion words and is not required. See General Input Format, earlier in this section for assigned decimal point location. Decimal points may be used in the tape for printout clarity; they will be ignored by the control.

Parantheses (ASCII Only). The left [(] and right [)] parantheses may be used to enclose informational statements within the tape. The information enclosed in parentheses can be composed of any valid ASCII characters. (Note: this feature does not function in EIA code.) All data within parentheses will be ignored by the control.

EXAMPLE:

N037G00X13700Y14002M06 (CHANGE TO .250 DRILL)

Tape Leader. The leading portion of the tape (prior to the first block) may contain sprocket holes only or any combination of punches except END-OF-BLOCK. This allows for unformatted readable image titles punched in the leader.

DETAILED PROGRAMMING DESCRIPTION

Linear Interpolation

Description and Programming Example. To select linear interpolation the standard EIA preparatory functions, G00, and G01 are used. In this mode of operation the control system moves the cutter in a straight line between the input points at the specified feedrate. A step of information is defined by two end points. Any dimension between ± 99.9999 inches can be traversed in a single step when in linear interpolation. Under G00 the machine slides automatically decelerate to a full stop at each end point. G01 overrides this deceleration to save time as noted in Section 1. Both slides move simultaneously if programmed in the same block. (See sequence number N100 in the programming example shown in Fig. 3-1.)

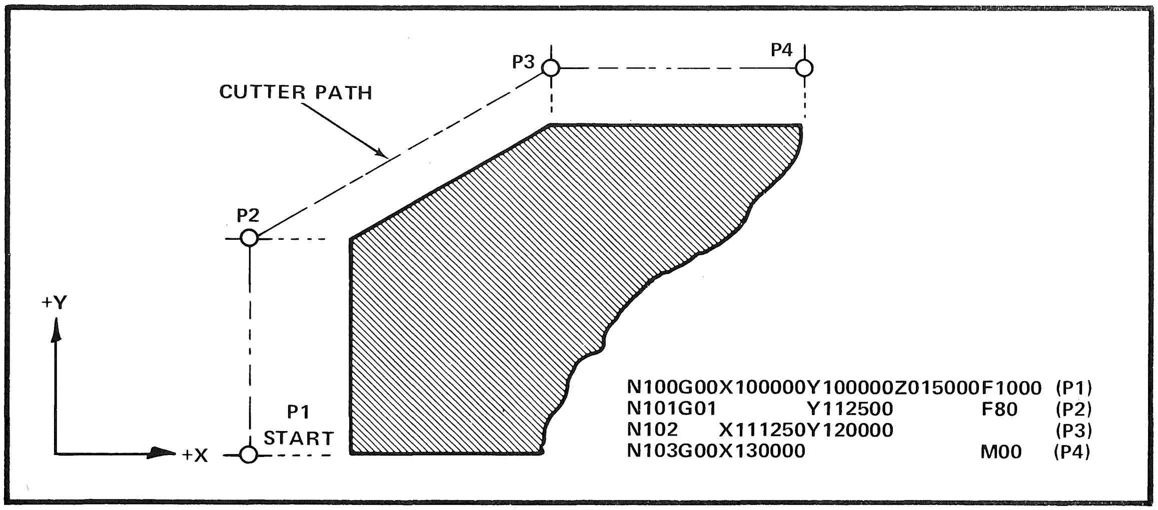


Fig. 3-1. Typical commands and moves in linear interpolation.

Sequence number N100 with G00 linear interpolation codes, positions the cutter at 100 IPM to the starting position in P1 example above. G00 is used since the change of direction in the next move exceeds 30°.

Sequence number N101 generates the straight line movement from P1 to P2 at 8 IPM. The G01 code inhibits deceleration at P2.

Sequence number N102 contains the absolute coordinates of P3. This command generates the angular movement from P2 to P3.

Sequence number N103 initiates the linear move from P3 to P4. The M00 code causes the slides to stop at P4. G00 code allows full deceleration to this point without overshoot.

Programming for Circles and Arcs

In linear interpolation straight line segments are used. The accuracy of the arc depends on the size of the line segments.

Chord Method (see Fig. 3-2). This is one means of determining the amount of programmed points necessary for the linear approximation of the arc, within a specified tolerance.

Linear interpolation with the chord method will be used to program the sample part shown on page XX. As indicated on the drawing, the radius of the part plus the cutter radius is 2.0 inches. The tolerance specified is .002 inches. To program this part it is first necessary to establish the number of steps required to machine this part within the specified tolerance. Secondly, the coordinate value of these end points will be calculated using trigonometric functions. Use the following steps to establish these factors.

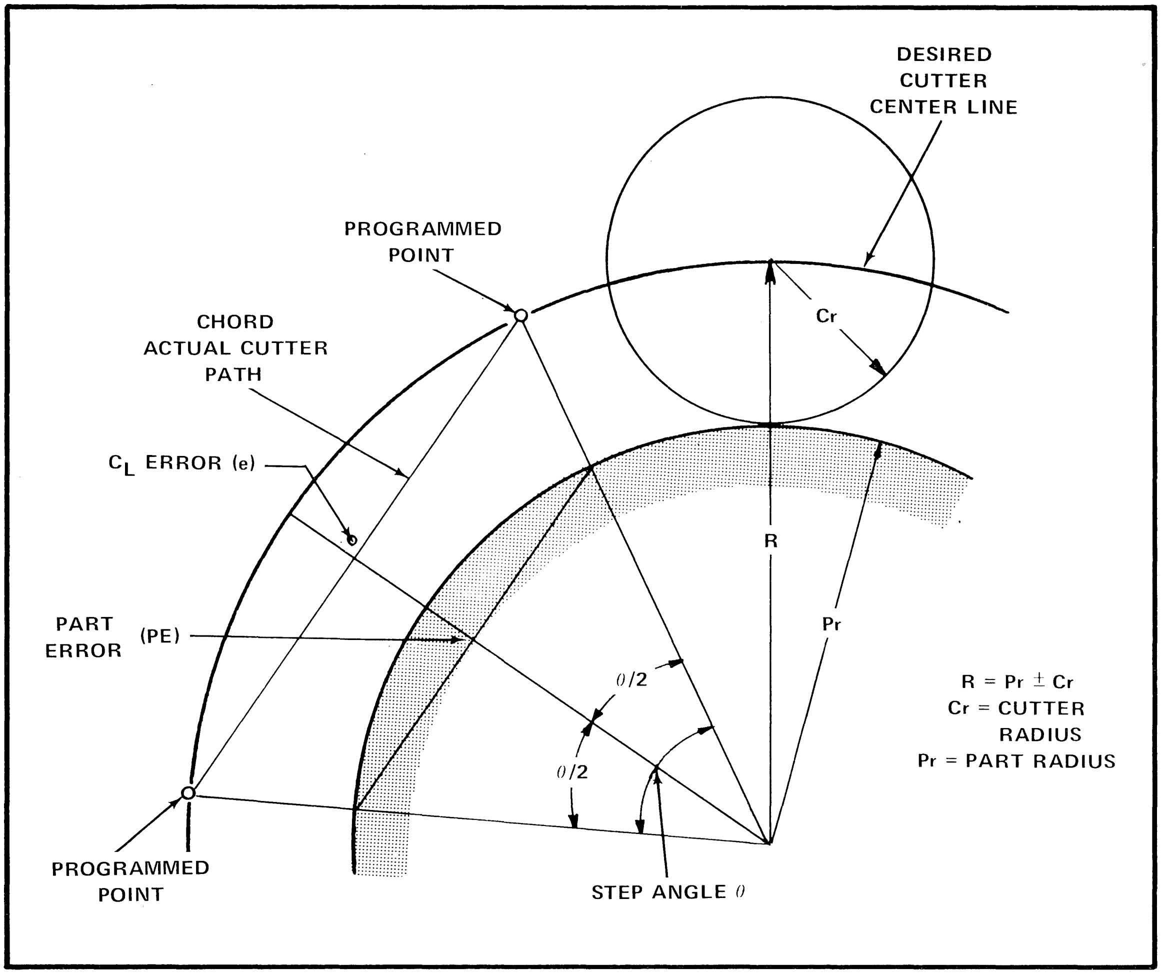


Fig. 3-2. The relationship between the desired cutter path and the actual cutter path when using the chord method to cut circles.

Step 1 —Establish the value (R), the part radius plus or minus the cutter radius. In our example, the 2-inch radius would be selected.

Step 2 —Select the desired cutter centerline error (e). Our specified tolerance is .002 inches. (See note below).

Ⓐ

Step 3 —Compute the step angle by $\theta = 2 \times \text{Arccos}\left(\frac{R-e}{R}\right)$. In our case, $\theta = 2 \times \text{Arccos}\left(\frac{1.998}{2}\right) = 5.2°$.

Step 4 —Step angles may be lowered for convenience in calculating step end points. In this example, a step angle of 5° will be used.

Step 5 —The number of steps can be calculated using the following formula:

$$N = \frac{a}{\theta}$$

where:

a = the total included angle of the arc

θ = step angle

N = number of steps

(If N is not a whole number, round to the next higher integer.)

$N = \frac{a}{\theta} = \frac{180}{5} = 36$ steps

NOTE

The actual part error (PE) is less than cutter error (e) for outside contouring and greater than (e) for inside contouring.

Step 6 —To find the coordinate value of step S_1 shown in Fig. 3-3.

The absolute coordinates assigned to P_3 and P_4 in the preceding example are:

P_3X10.0000Y12.0000

P_4X10.0000Y10.0000

Coordinates of Point S_1 are:

X = 10.0000 + .1743 = 10.1743

Y = 10.0000 + 1.9924 = 11.9924

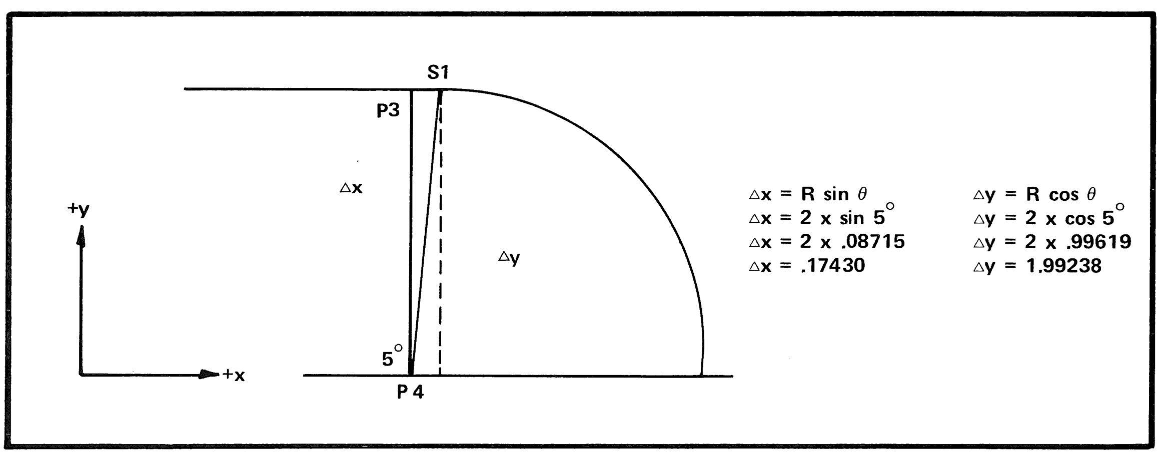
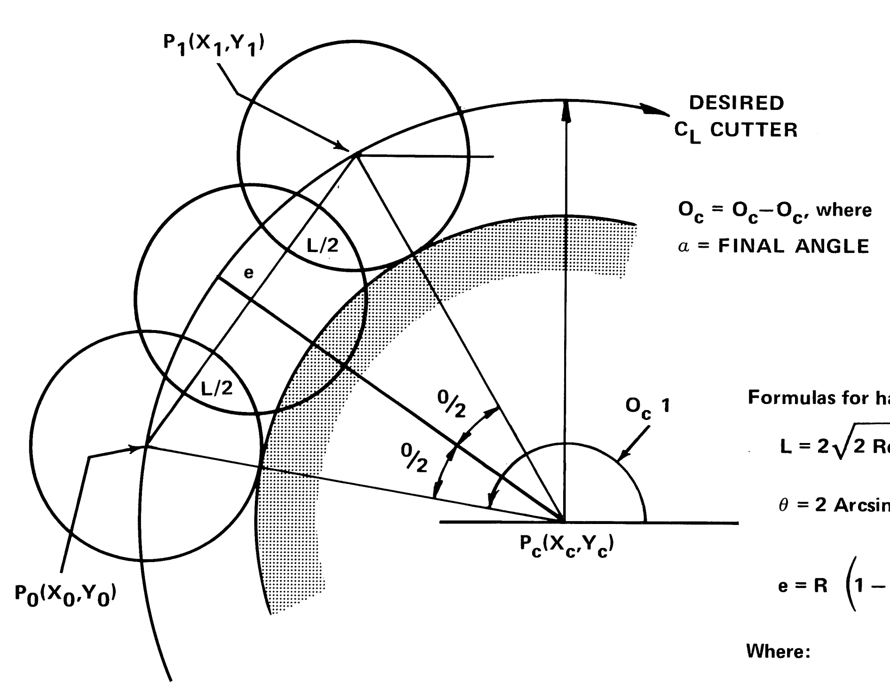


Fig. 3-3. Calculating end-point coordinates when using the chord method to cut circles.

Step 7 —The remaining 35 points are calculated in the same manner.

The formulas given (with Fig. 3-4) show all relations of step angle, step length, radius and error for the chord method.



Formulas for hand calculations are:

$$L = 2\sqrt{2Re - e^2}$$

$$\theta = 2\ \text{Arcsin}\left(\frac{L}{2R}\right) = 2\ \text{Arccos}\left(\frac{R-e}{R}\right)$$

$$e = R\left(1 - \cos\frac{\theta}{2}\right)$$

Where:

L = STEP LENGTH

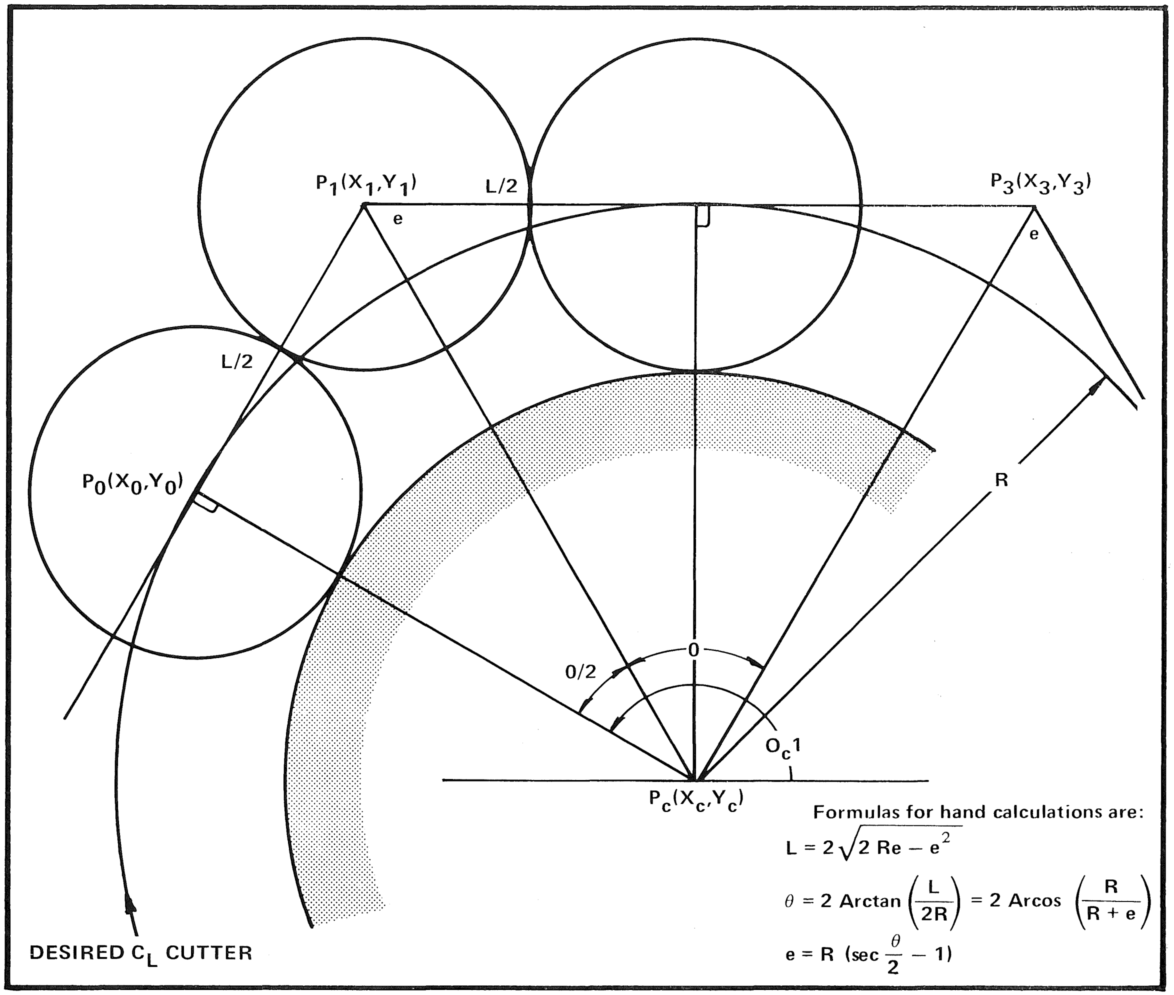
Fig. 3-4. Relationship of step angle, step length, radius, and error when using the chord method of cutting circles.

In the chord method all the end points are on the circle and all the error is inside the circle. Of the two other methods for approximating circles with small radii, the TANGENT method is useful since the error is entirely outside the circle. Longer spans with the same tolerance are permitted by use of the SECANT method, which has part of the error inside and part outside the circle.

Tangent Method. In this method, tangents to the circle are drawn at the midpoints of each step. Half steps are used for the first and last steps so that these points will be on the exact offset point of the curve intersections (see Fig. 3-5).

The number of steps are the same as with the chord method.

$N = \frac{a}{\theta}$. The formulas are shown below for the tangent method.



Formulas for hand calculations are:

$$L = 2\sqrt{2\,Re - e^2}$$

$$\theta = 2 \text{ Arctan} \left(\frac{L}{2R}\right) = 2 \text{ Arcos} \left(\frac{R}{R + e}\right)$$

$$e = R\ (\sec \frac{\theta}{2} - 1)$$

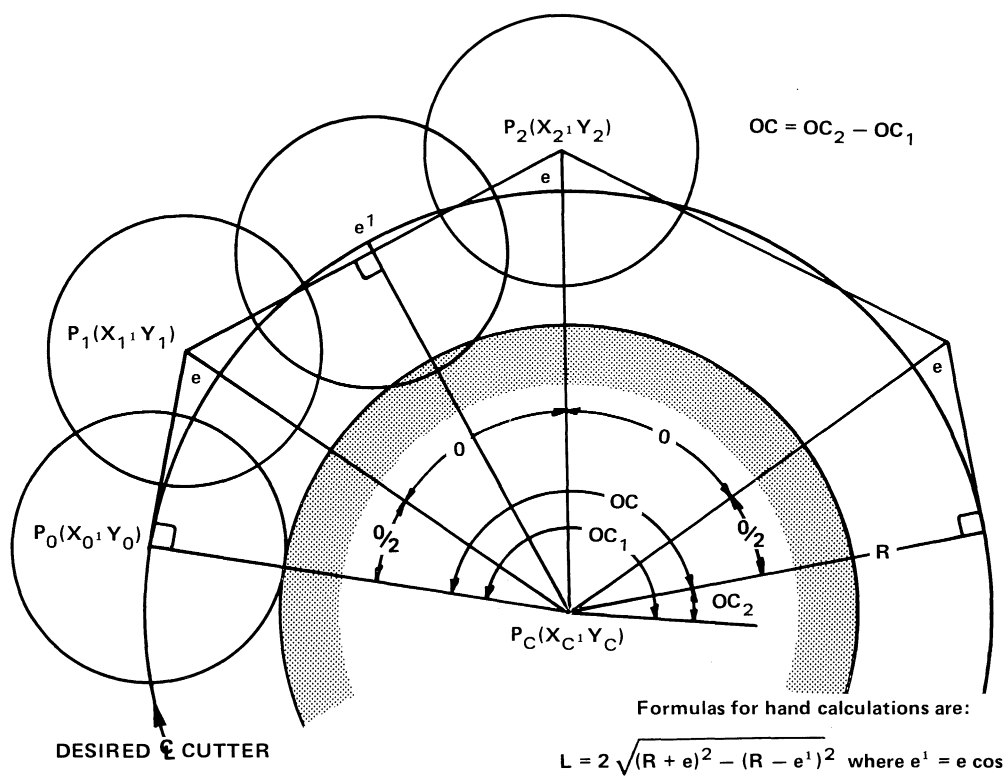
Fig. 3-5. Desired cutter path vs actual cutter path when cutting circles using the tangent method.

Secant Method. This method allows longer steps by permitting the error to be bilaterial (see Fig. 3-6). For easier programming, a combination of tangents and secants is used.

First the step angle (θ) for the first and last step is found from the formulas for the tangent method, using the radius of the cutter centerline and the tolerance.

Next, the step angle (ϕ) for the secant is calculated for the formulas below, and the number of steps (N) is:

$$N = \frac{a - \theta}{\phi} \quad \text{(round to the next higher integer.)}$$



Formulas for hand calculations are:

$$L = 2\sqrt{(R + e)^2 - (R - e^1)^2} \quad \text{where } e^1 = e \cos O/2$$

$$L = 2\sqrt{e\,(1 + \cos O/2)\,(2R + e - e \cos O/2)}$$

$$O = 2 \arcsin\left[\frac{L}{2\,(R + e)}\right] = 2 \arccos\left(\frac{R - e^1}{R + e}\right)$$

$$c = R/2\,(\sec O/2 - 1)$$

Fig. 3-6. The relationship of the desired cutter path and the actual cutter path when using the secant method of cutting circles.

Sample Program-Absolute Dimensioning

The part program listed in Table 3-1 generates the tool path for a one-inch diameter cutter to cut the sample part in Fig. 3-7. In this program, all dimensions are given in absolute coordinate values from the zero point X0, Y0. The set up point, PO, is located one-inch to the left and below the lower left hand border of the part. The control is synchronized at this point with the values X0, Y0.

N001:	The tool is poisitioned to start the cut at P1 (X=.5", Y=.5"). F2000 indicates that this motion is at 200 IPM and M03 activates spindle rotation. (A rewind stop code and end of block is required prior to N001.)
N002:	G00 establishes linear interpolation with deceleration at the end of the move and M09 activates the coolant supply.
N003:	The tool moves to P2 at 8 IPM (F80). The cutting depth is maintained.
N004:	The tool moves to P3 to start entry into the circular cutout. G01 inhibits end point deceleration to maintain maximum tool velocity throughout this cutout.
N005 to N030:	These blocks describe the tool path in the circular cutout through P4 to P5.
N031:	The tool moves to P6. G00 causes deceleration at P6 since the ensuing change of direction exceeds 30°.
N032 to N035:	The tool moves to P7, P8, P9, and P10.
N036:	The tool moves to P11. G01 again is used to inhibit deceleration.
N037 to N051:	The tool moves to P12, P13, and P14 with no deceleration.
N052:	The tool moves to P15 and decelerates with G00.
N053 to N054:	The tool moves to P16 and P17 to complete the part path.
N055:	The tool moves to P0 at rapid traverse (G50). M30 rewinds the tape to the rewind stop code prior to N001 and shuts off the spindle and coolant.

TABLE 3-1

Part Program for Sample Part — Absolute Dimensions

```
N001X5000Y5000M03F2000M03;
N002G00M00;
N003&25000F80;
N004G01X10000;
N005X10603Y25036;
N006X11197X25145;
N007X11773/22573;
N009X12840Y25885;
N010X13316Y26257;
N011X13743Y26684;
N012X14115Y27160;
N013X14426Y27676;
N014X14675Y28227;
N015X14855Y28803;
N016X14964Y29397;
N017X15000Y30000;
N018X14964Y30603;
N019X14855Y31197;
N020X14675Y31773;
N021X14427Y32324;
N022X14115Y32840;
N023X13743Y33316;
N024X13316Y33643;
N025X12840Y34115;
N026X12324Y34427;
N027X11773Y34675;
N028X11197Y34885;
N029X10603Y34964;
N030X10000Y35000;
N031G00X5000;
N032Y53604;
N033X40384Y65398;
N034X65000Y53090;
N035Y336615;
N036G01X55786Y32929;
N036Y32930;
N038X55566Y32815;
N039X55363Y32600;
N040X55215Y32470;
N041X55097Y32252;
N042X55025Y32015;
N043X44000Y31769;
N044Y27124;
N045X55024Y26880;
N046X55095Y26646;
N047X55210Y26430;
N048X55366Y26241;
N049X55555Y26085;
N050X55771Y24970;
N051X56005Y25898;
N052G00X65000Y24099;
N053Y5000;
N054X5000;
N055G50X0Y0M30.
```

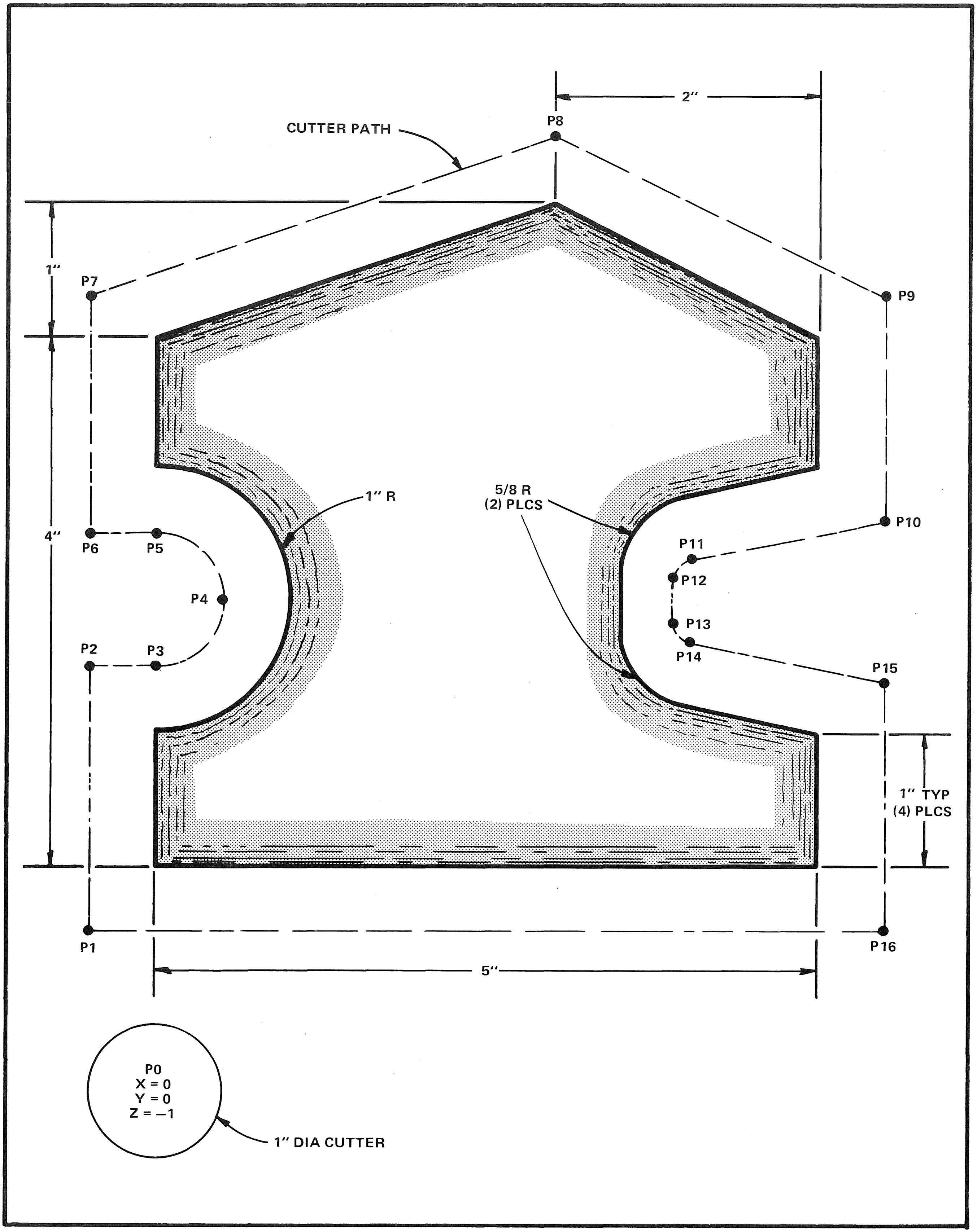


Fig. 3-7. Tool paths for cutting a typical part.

Axis Inversion

This feature permits both left- and right-hand parts to be cut from the same tape.

Under the normal operating conditions, Mirror Image OFF, the machine slide moves in the same direction as the control is commanding. At times, however, it is desirable for right- and left-hand parts to be machined from the same part program tape. The Tektronix 1704 Control is supplied with Mirror Image ON/OFF switches for each axis (optional). To reverse the direction of any or all axes with respect to the program, it is only necessary to set the respective MIRROR IMAGE switch or switches to the ON position.

In Mirror Image, the axis inverts about the programmed zero location.

Figure 3-8 illustrates the relationship to zero of the part to be programmed with the X-axis under normal operating conditions and with X-axis inverted.

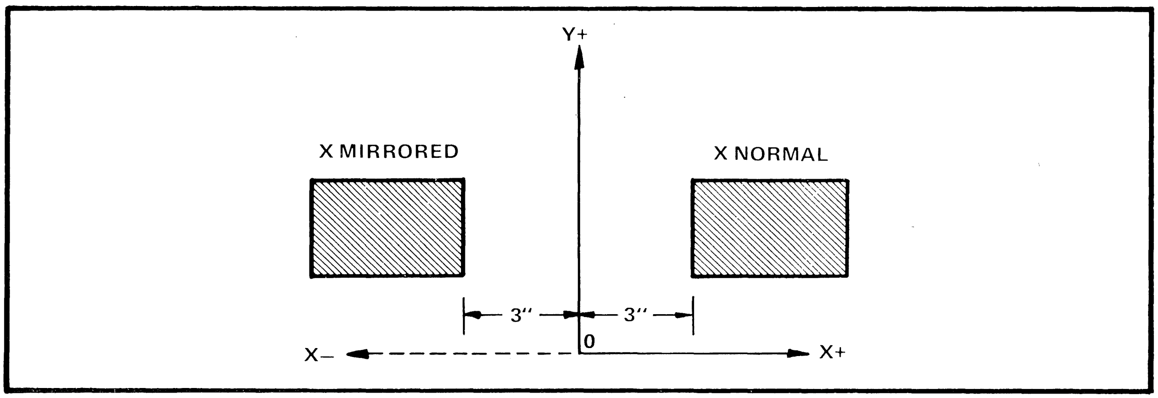


Fig. 3-8. X-axis inversion.

If the parts or fixtures are not initially set up at an equal distance from the programmed zero location, and Mirror Image ON is selected, it will be necessary to realign using the zero shift controls before cutting the inverted part.

In Fig. 3-9, the X, Y plane has been chosen for the part and all four combinations of Axis Inversion are illustrated. Note that two parts are right-hand parts and two are left-hand parts.

Feedrate Coding

General Description. The feedrate (F) word is coded directly in inches per minute. It is a four digit number with the decimal point assumed to be between the third and fourth digit (XXX.X).

Example:

.1 IPM = F1 (minimum)

1.6 IPM = F16

30 IPM = F300

200 IPM = F2000 (maximum)

 Ⓐ

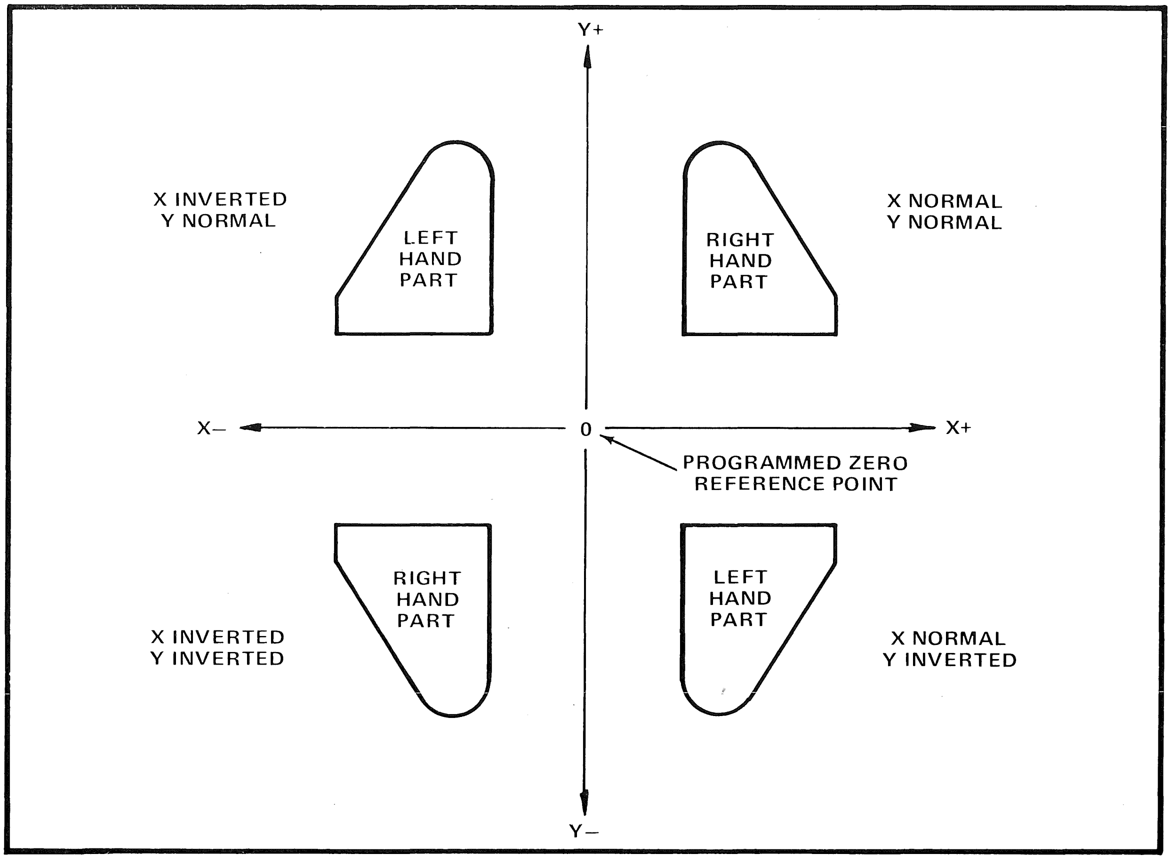


Fig. 3-9. The effect of axis inversions on workpieces.

Leading zeros may be omitted but trailing zeros must be coded.

The feedrates acceptable by the control systems are as follows:

Minimum Feedrate = 0.1 inch per minute

Maximum Feedrate = 200.0 inches per minute

NOTE

The maximum and minimum feedrates per axis of the machine tool will dictate the feed range possible to use.

The following example illustrates the feedrate word and when it occurs with respect to the motion command.

Example:

N014	X(P1)	Y(P1)	
N015		Y(P2)	F100

N014 is the end point of the previous span and N015 is the movement to P2 in the Y-axis and it will be interpolated at 10 inches per minute (F100).

All traverse and feedrate motions are fully interpolated.

Traverse rates to 400 inches per minute are available from the 1704 control. The particular machine tool application determines the actual maximum feedrate.

Reader Limitation. For continuous non-decelerated machine tool motion, under preparatory functions of G01, G02, and G03, the tape reader must read blocks of data faster than they are being executed by the machine tool. Usually, this is the situation; however, it is possible that with a high feedrate or with very short movements, the machine can reach the programmed point before the next block of tape has been completely read. This would result in a stop of machine motion until the reader has completed reading.

This situation does not occur with decelerated machine tool motion. If a stop of the machine motion or dwelling of the cutter in the material can be tolerated, then the programmer should use G00, G20, or G30 preparatory functions. In this case no reader limitation corrections need to be made.

To check for possible reader limitation, two time factors must be calculated:

1. The machine execution time of the tool motion.
2. The read time of the succeeding block of data.

The formulas for calculating each time factor are given below:

MACHINE EXECUTION TIME (T1)

$$T1 = F_1 \quad \begin{matrix} L \times 60 \\ \times F_2 \\ 2 \end{matrix}$$

Where: L = length of move

F_1 = initial feedrate in IPM

F_2 = final feedrate in IPM

TAPE READ TIME (T2)

$$T2 = .010 \times \frac{N-1}{300}$$

Where: N = number of characters in the next block

.010 = approximate acceleration time of reader

300 = reader speed (300 characters per second)

Both times T1 and T2 are in seconds.

The system will be reader limited whenever T2 is larger than T1.

If reader limitation does occur, one of three corrective measures can be used:

1. Decrease the Feedrate.

2. Decrease the number of characters in a block, if possible by:
   a. Eliminating leading zeros in motion and feedrate words,
   b. Eliminating sequence numbers, or
   c. Placing auxiliary function words (G, M, S, T) in a previous block if this does not affect machine responses.
3. Use preparatory functions G00, G20, or G30, which allow for deceleration of machine motion.

Feedrate Calculation for Short Moves in G01 Mode

The graph in Fig. 3-10 provides an easy way of calculting the maximum possible feedrate when making short moves in G01, G02, or G03. Use the graph as follows:

1. Total the characters in the command block.

2. Locate this total on the NUMBER OF CHARACTERS IN BLOCK scale.

3. Move vertically from the number of characters in the block to the appropriate MOVEMENT DISTANCE (IN) line. Interpolate as necessary.

4. From the intersection of a vertical from the NUMBER OF CHARACTERS IN BLOCK scale and the appropriate slope from the MOVEMENT DISTANCE (IN) scale, go right horizontally to the MAX FEEDRATE (IN PER MIN) scale appropriate for the tape reader used in the system.

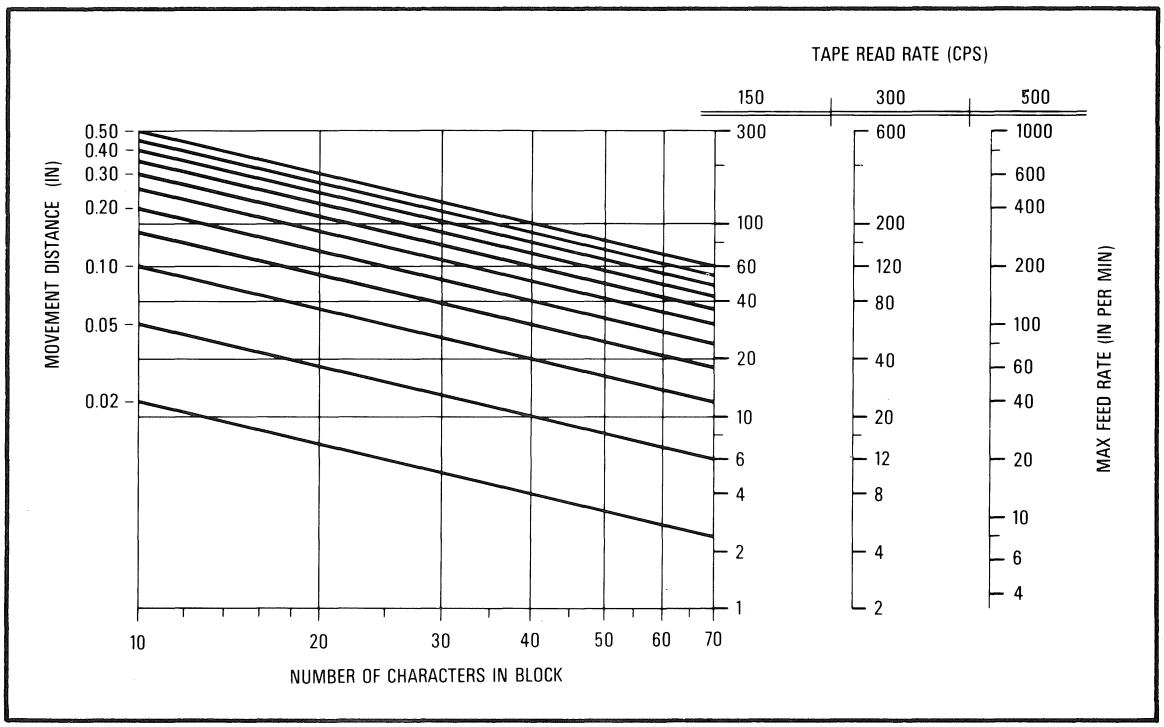


Fig. 3-10. Feedrate calculator for short moves.

OPTIONAL CONTROL FEATURES

Circular Interpolation

Description and Programming Examples. Circular interpolation gives the programmer the ability to move the tool in circular arcs up to 90 degrees within one circle quadrant. The cutter path along the arc is generated by the control system.

To select circular interpolation the following EIA preparatory functions G02 and G03 are used.

G02	CIRCULAR INTERPOLATION CLOCKWISE (CW)—INHIBIT DECELERATION
G03	CIRCULAR INTERPOLATION COUNTERCLOCKWISE (CCW)—INHIBIT DECELERATION
G20	CIRCULAR INTERPOLATION CLOCKWISE (CW)—WITH END POINT DECELERATION
G30	CIRCULAR INTERPOLATION COUNTERCLOCKWISE (CCW)—WITH END POINT DECELERATION

The circular arc must be parallel to the plane, a canted arc or helical motion cannot be generated. The plane selection for circular interpolation is accomplished by addressing the desired axes, with the center point location.

The direction of the arc is determined by looking at the arc in its plane from the positive direction of the perpendicular axis. Figure 4-1, following, shows a composite of the three possible configurations.

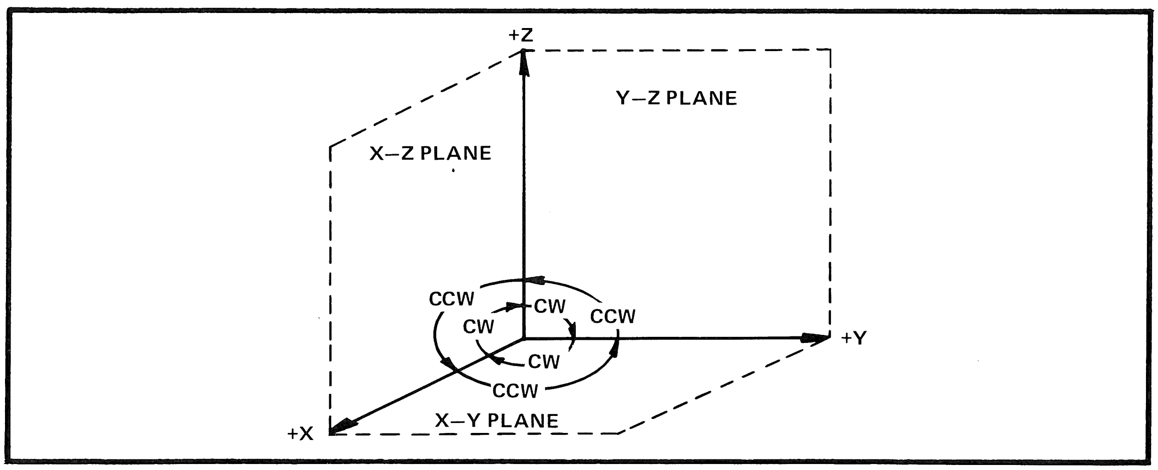


Fig. 4-1. Plane selection for circular interpolation.

Two blocks of tape are required to input circular information to the control system. For the absolute system the input would be:

N001 Absolute location of the beginning point of the arc.

(Normally not required to be programmed as this is usually the present slide position as the result of a previous programmed step.)

N002 Circular G code (direction or rotation), absolute location of the center of the arc, and absolute location of the ending point of the arc.

Circular interpolation is a higher order of interpolation than linear interpolation. In using circular interpolation, the programmer has the means to reduce tape preparation time and also the overall tape length. It greatly simplifies the programming of the two dimensional circular segments.

In linear interpolation only straight line segments are used. The accuracy of the arc depends on the length of the line segments and the radius of the curvature of the part. The sample part in Fig. 3-7 was programmed using linear interpolation. It required 27 steps to program the 180° circular arc. A printout of the programmed points for the sample part using linear interpolation is shown in Table 3-1.

To program the 180° circular arc using circular interpolation requires only two circular steps. The control requires only two blocks of input information.

Circular "G" code, absolute location of the center of the arc (PC) and an absolute location of first quadrant point (P4).

Absolute location of ending point (P5).

A printout of the programmed points for the sample part (Fig. 4-2) using both linear and circular interpolation is shown in Table 4-1.

TABLE 4-1

Sample Part Programmed with Linear and Circular Interpolation

Absolute Dimensions Only	Position No.
N001X5000Y5000M03F2000	P0
N002G00F20M09	P1
N003Y25000F80	P2
N004G01X10000	P3
N005G03X15000Y30000110000J30000	P4
N006X10000Y35000	P5
N007G00X5000	P6
N008Y53604	P7
N009X40384Y65398	P8
N010X65000Y53090	P9
N011Y36615	P10
N012G01X55786Y32929	P11
N013G03X55000Y31769156250J31769	P12
N014G01Y27124	P13
N015G03X56005Y25898J27124	P14
N016G00X65000Y24099	P15
No17Y5000	P16
N018X5000	P1
N019X0Y0M30F2000	P0

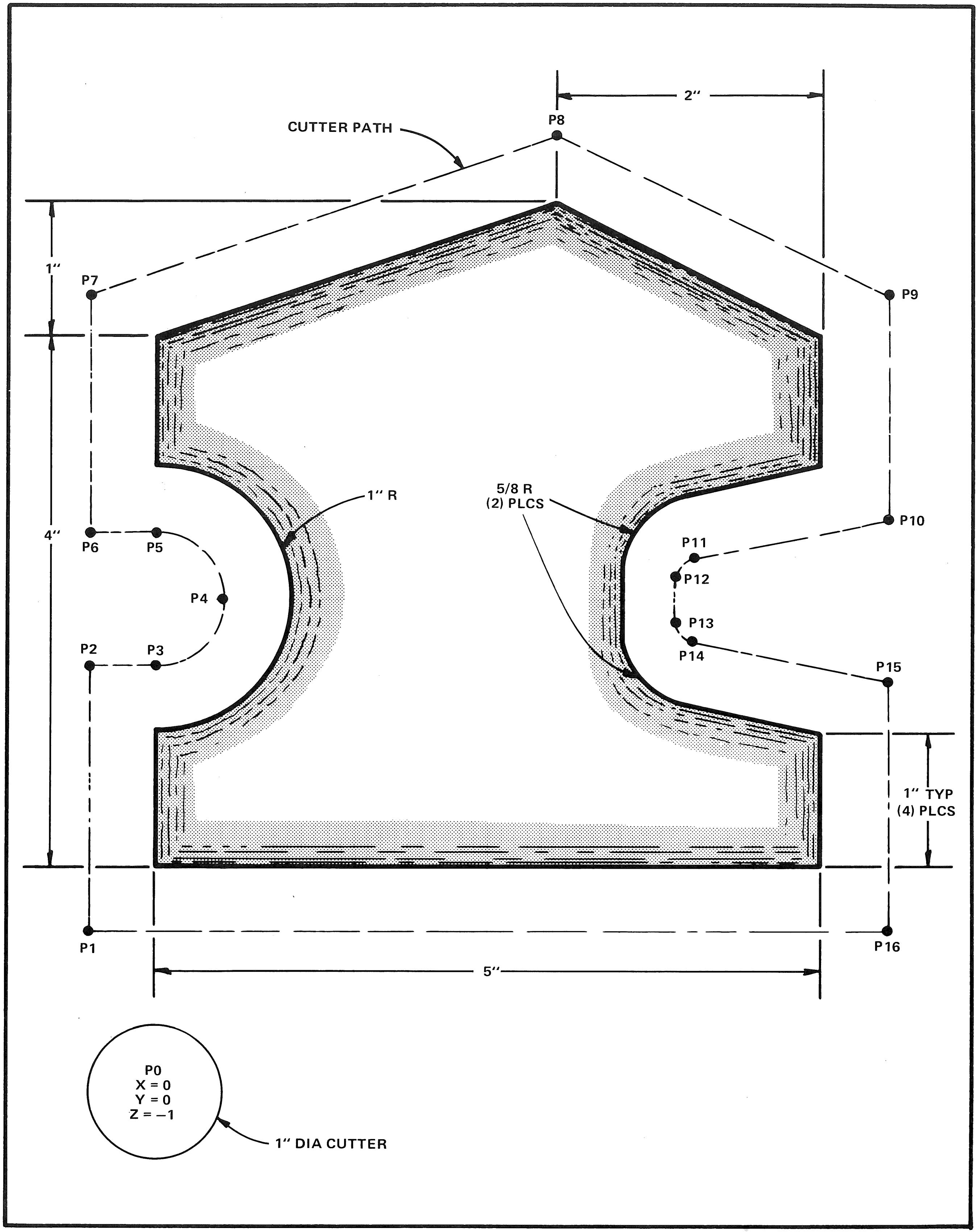


Fig. 4-2. Typical part cut through the use of both linear and circular interpolation.

The tape length for the above program using circular interpolation would be approximately 3.8 feet as compared to a tape length of 6.3 feet for the linear program shown in SECTION 3.

In the following example, succeeding arcs have been programmed. The end point of the first arc becomes the starting point of the next arc. The G function (direction of rotation code) does not have to be repeated if the direction of the rotation does not change. The center location does not have to be repeated if it does not change.

Example:

TABLE 4-2

N001	G00	X100000	Y100000			
N002		X110000				(Beginning Point arc 1)
N003	G02	X120000	Y110000	I125000	J100000	(Ending Point at quadrant of Arc 1)
N004		X130000	Y100000			(Ending Point Arc 1 - Beginning Point Arc 2)
N005		X125000	Y095000	I125000		(Ending Point at quadrant of Arc 2)
N006	G20	X120000	Y100000			(Ending Point Arc 2 - (Beginning Point Arc 3)
N007	G30	X125000	Y095000	I120000	J95000	(Ending Point Arc 3 - with deceleration)

The cutter motion for this example program is shown in Fig. 4-3.

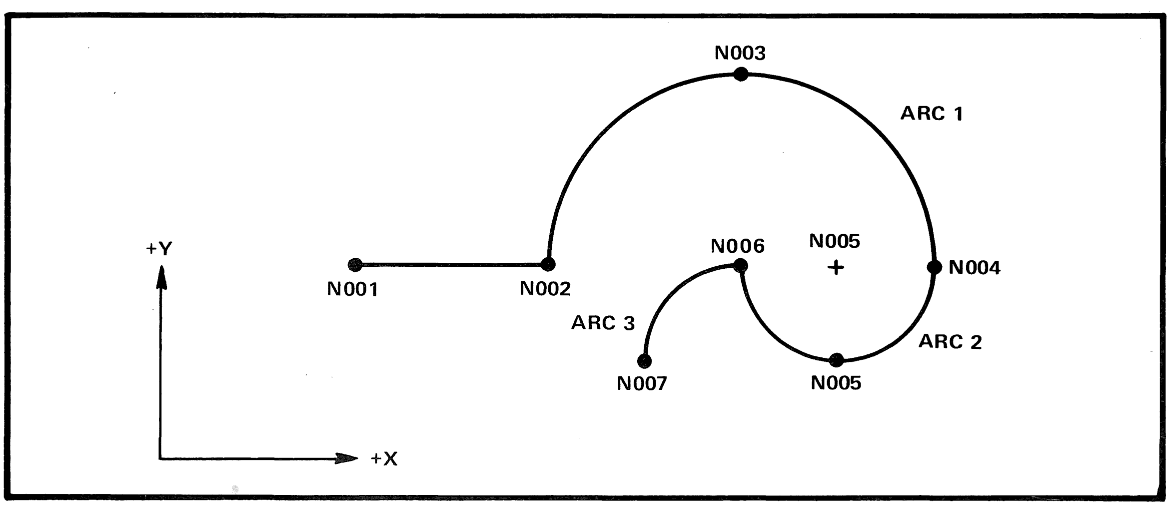


Fig. 4-3. Cutter motion for the commands given in Table 4-2.

The two axes given in the block defining the end point of the arc determine which plane the arc is in. Only two axes may be given the end point location. Blocks N003, N005, and N007 in example (page X) place the arcs in the X-Y plane. The two axes location of the ending point must be given for each ending point to enable plane selection. *Even though the axis location does not change, the axis must be given.* There are separate stores for the center point location and the end point location. If the end point store is not re-addressed, a value of zero will be set into the stores. The control will use this value for the end point location and attempt to generate an arc accordingly.

The radius of the arc is determined by the distance from the beginning point to the center. The control starts to generate the arc, in the indicated direction, from the beginning point around the center location. The arc will be generated until the end point is encountered.

When programming with the absolute system, the center of a circular arc may have negative coordinate values. (The center location is not restricted to the first quadrant.)

Circular Interpolation Ranges

Maximum radius is 99.9999 inches.

Minimum radius is 0.0001 inch.

Maximum feedrate acceptable by the control system is 200.0 inches per minute when in circular interpolation.

Alternative System Resolution and Feedrates

The standard resolution of the 1704 controls is 0.0001 inch with a maximum feedrate of 200.0 IPM.

Two optional resolutions/feedrates are available:

1. 0.0002 inch with maximum feedrate of 400.0 IPM.

2. 0.001 inch with maximum feedrate of 2000 IPM.

For option 1, minimum feedrate is 0.2 IPM and is programmable in .2 IPM steps to 400 IPM.

Example:

F1	0.2 IPM (minimum)	F1
F100	10.0 IPM	
F400	400.0 IPM (maximum)	

No other modifications to the programming information in Sections 1 through 4 are necessary.

For option 2, minimum feedrate is 1 IPM and is programmable in 1.0 IPM steps to 2000 IPM.

Example:

F1	1.0 IPM (minimum)
F100	100.0 IPM
F2000	2000.0 IPM (maximum)

In addition, with option 2, all motion words (X, Y, Z, R, I, J, K) use three places preceding and three places after the decimal point.

Example:

X1	.001 inches
X1000	1.000 inches
Y12345	12.345 inches
Z678901	678.901 inches

All other information in Sections 1 through 4 is not affected.

Fixed Cycle Option (G80 Series)

The Fixed Cycle Option is available in Tektronix 1704 controls to provide for ease of programming in drilling, tapping and boring operations. As the name implies, a fixed operation of positioning, feeding, dwelling and returning is accomplished in one block of programming by using the appropriate preparatory functions (G codes). Although a standard sequence is defined for each code by EIA Standard RS-326-A, the Tektronix G80 series is programmable at the time of manufacture and any sequence of operations may take place with any desirable G80 code. Dwells, spindle stops, spindle reverse, rapid or feed out, and/or program stops may be programmed for any of the Fixed Cycles.

FIXED CYCLE SEQUENCE

CODE	MOVEMENT IN	AT BOTTOM DWELL	AT BOTTOM SPINDLE	MOVEMENT OUT TO "R" PLANE	
80	-----	-----	-----	-----	XYR Positioning
81	Feed	-----	-----	Rapid	Drill, Spot Drill
82	Feed	Yes	-----	Rapid	Counterbore
83	Intermittent	-----	-----	Rapid	Deep Hole Drill
84	Feed	-----	Stop/Rev	Feed	Tap
85	Feed	-----	-----	Feed	Bore
86	Feed	-----	Stop	Rapid	Bore
87	Feed	-----	Stop	Program Stop Manual	Bore
88	Feed	Yes	Stop	Program Stop Manual	Bore
89	Feed	Yes	-----	Feed	Bore

With the programming of a series 80 code, the X, Y, and Z axes will rapid traverse to the X, Y, and R coordinates programmed. The Z axis can optionally position concident with the XY motion or be inhibited until XY positions are obtained. When a dwell occurs, a signal is generated to start either an internal or external timer which in turn generates a signal at the completion of the dwell cycle. The spindle stop and spindle reverse likewise generate a signal to perform the appropriate function and must receive a signal indicating completion of the spindle function.

G80 dwell and peck times are selected by front panel switches when the internal timers are used. Range is 1.3—12 seconds in 15 positions.

The deephole drill cycle is cycled by an internal peck signal, causing the spindle to retract to the "R" plane, then rapid back into the previous depth where it continues to drill.

R Clearance Coordinate

An R Clearance coordinate must be entered when fixed machining cycles are programmed by one of the series G80 preparatory commands. This coordinate establishes an X, Y clearance plane between the tool tip and part surface, which allows the tool to traverse across the part without contact. The amount of clearance is determined by the programmer based on possible variations of the part surface. Generally, .050 to .100 inch is allowed.

The R Clearance plane is established and the fixed cycle preparatory functions are executed in one of the two following sequences.

1. If the tool tip is located at a dimension that is closer to the workpiece than desired by the R coordinate, the sequence is as follows:

   a. The Z axis will rapid traverse to the R position.

   b. The X and Y axes will simultaneously rapid traverse to the command position.

   c. The Z axis will feed, at the command feedrate, to the Z position.

   d. The Z axis will return the tool tip to the R Clearance plane in accordance with the specified G Function.

2. If the tool tip is located at a dimension that is farther from the workpiece than desired by the R coordinate, the sequence is as follows:

   a. The X and Y axes (and Z, if applicable) will simultaneously rapid traverse to the command position.

   b. The Z axis will rapid traverse to the R position.

   c. The Z axis will feed, at the command feedrate, to the Z position.

   d. The Z axis will return the tool tip to the R Clearance plane in accordance with the specified G Function.

Dwell

The Dwell option causes the machine slides to stop at a point and dwell there for a specified period of time. The dwelling of a tool motion is caused by a G04 code and the F word, which determines dwell time from .1 second to 99.9 seconds.

To accomplish a dwell, the G04 Fxxx must be programmed in a separate block. The value of the F word determines the dwell time in increments of .1 seconds. The F word used with G04 for dwell does not cancel the existing feedrate, which is automatically reinstated in the succeeding block.

Example:

G04	F1	(dwell .1 seconds—minimum)
G04	F10	(dwell 1 second)
G04	F154	(dwell 15.4 seconds)
G04	F999	(dwell 99.9 seconds—maximum)

Quill Actuator Interface

This option allows for the use of quill actuators on two axis machines. These actuators are usually pneumatic or hydraulic devices that raise and lower the machine tool quill and are activated by a signal from the control. Many such devices also have a rotary turret with mechanically set depth stops for operations at varying depths on the part to be machined.

The auxiliary codes used for this option are:

M50	Quill Up
M51	Quill Down
M52	Rotate Turret One Position
G98	Cancel Drill Cycle
G99	Drill Cycle Active

The M codes for this option may be programmed in a block with motion words. If either M50 or M51 is programmed in a block with motion words, the quill cycle will be activated before motion occurs. If turret rotation of more than one position is desired, successive blocks of M52 must be programmed.

DRILL CYCLE (G99) will cause a complete down-up cycle at each programmed dimension. This function is modal and is active in the block programmed and in following blocks until cancelled by G98.

Spindle Control

This option allows the programmer to control the speed of the spindle from tape commands. The spindle speed code is two (2) digits. Spindle codes are programmed with an S word. The spindle speeds corresponding to the S words are the responsibility of the machine tool builder.

Example:

S02	15 RPM
S40	100 RPM

With the spindle control option, axis motion will not occur until the spindle has reached program RPM.

Tool Control

The tool selection option allows the programmer to select appropriate tools from the machines tool changing carrier, if so equipped. The tool selection code is two (2) digits. Tool selection codes are programmed with a T word. The tool codes corresponding to the T words are the responsibility of the machine tool builder.

Tape Search

This option allows the operator to search for a selected sequence (N) number in the tape. If this feature is used, the programmer must:

1. Program an N word in the block to be searched.

2. Include all data in the searched block pertinent to machine operation such as feedrate, spindle rotation, and coolant.

If the operator searches to a block without all data necessary to machine operation, he must enter the missing values through the MDI switches.

Metric Option

For machine tools using metric dimensioning; the following changes to program entry of motion and feedrate words are necessary.

All motion words have three places leading and three places following the decimal point (X3.3).

All programming is in millimeters (mm).

Maximum motion dimension word is 999.999 mm (e.g. X999999) maximum dimension for I, J, K words is also 999.999 mm.

Feedrates are programmed in mm per minute.

F word range is 1 mm per minute to 2000 mm per minute in 1 mm steps. The F word has four digits leading and none trailing the decimal point (F4).

Example:

F1	1 mm per minute (min.)
F100	100 mm per minute
F2000	2000 mm per minute (max.)

Rapid Traverse (G50) is 4000 mm per minute maximum.

MANUAL CHANGE INFORMATION

At Tektronix, we continually strive to keep up with latest electronic developments by adding circuit and component improvements to our instruments as soon as they are developed and tested.

Sometimes, due to printing and shipping requirements, we can't get these changes immediately into printed manuals. Hence, your manual may contain new change information on following pages.

A single change may affect several sections. Sections of the manual are often printed at different times, so some of the information on the change pages may already be in your manual. Since the change information sheets are carried in the manual until ALL changes are permanently entered, some duplication may occur. If no such change pages appear in this section, your manual is correct as printed.