SECTION IV

MANUAL PROGRAMMING

4.1   POINT-TO-POINT PROGRAMMING.

Most positioning programs can be prepared by the programmer manually writing a manuscript followed by a clerk transferring the data onto a program tape. The following steps are usually involved:

1. Workpiece dimensioning. The programmer studies the engineering drawing and adds any dimensions required that are omitted or not in the proper format for the control. For example, absolute dimensions must be converted to incremental dimensions, holes on bolt circles must be broken down into X-Y moves using trigonometric tables.

2. Tooling and Fixturing. The means for holding the workpiece down must be determined, and, if necessary, sketches provided to the operator. The tools required for machining must be specified, if possible, through use of a tool drawing classification system that includes tool setting information. Figure 4-1 shows a typical tool classification list which includes the following:
    a. Cutting tool name, ID number, diameter, length, material composition, number of teeth and flute length
    b. Toolholder information
    c. Adapter or extension diameter and length
    d. Special information such as tool configuration parameters
    e. Tool part number

When tool changes are made, some method of adjusting for random length tools is required. This may be done using either of the following methods:

Method 1: Tool length compensation using a presetting height gage with micrometer adjustment. This gage has a 1" micrometer adjustment that allows setting tool lengths within .001" (see Figure 4-2). The arm is removable, permitting it to be changed from one to another of the 1" step spaces.

    a. Using the height gage, measure the length of each tool. This value is entered below as TOOL LG.

NAME	CODE	TOOL DIAMETER	SETTING DISTANCE	TYPE	NO. OF TEETH	FLUTE LENGTH	EXTENSION DIA.	EXTENSION LENGTH	HOLDER NUM	TOOL PART NO.
CBORE	2704	0.4375	6.5000	HSS	3	.0	.0	.0		532106
DRILL	2705	0.7500	7.5000	HSS	2	5.87	.0	.0		0
DRILL	2709	0.1719	6.0000	HSS	2	2.12	.625	3.82		563017
REAM	2718	0.4385	7.5000	HSS	5	1.75	.0	.0		563522
DRILL	2727	0.1495	5.5000	HSS	2	2.18	.625	3.82		563016
CSNK	2741	0.5000	1.1250	HSS	3	.0	.0	.0	37	563019
MILL	2793	0.2500	6.5000	CAR	2	.50	.625	5.0		564103

Figure 4-1.   Typical Tool Classification List

PRE-SETTING HEIGHT GAGE
WITH MICROMETER ADJUSTMENT

THIS GAGE HAS A I" MICROMETER ADJUSTMENT THAT ALLOWS
SETTING TOOL LENGTHS WITHIN ±.OOI. THE ARM IS RE-
MOVABLE PERMITTING IT TO BE CHANGED FROM ONE TO
ANOTHER OF THE I" STEP SPACES.

±.OOI
MAXIMUM
GAGING DIMENSION

GAGE LINE
LOCATION

A
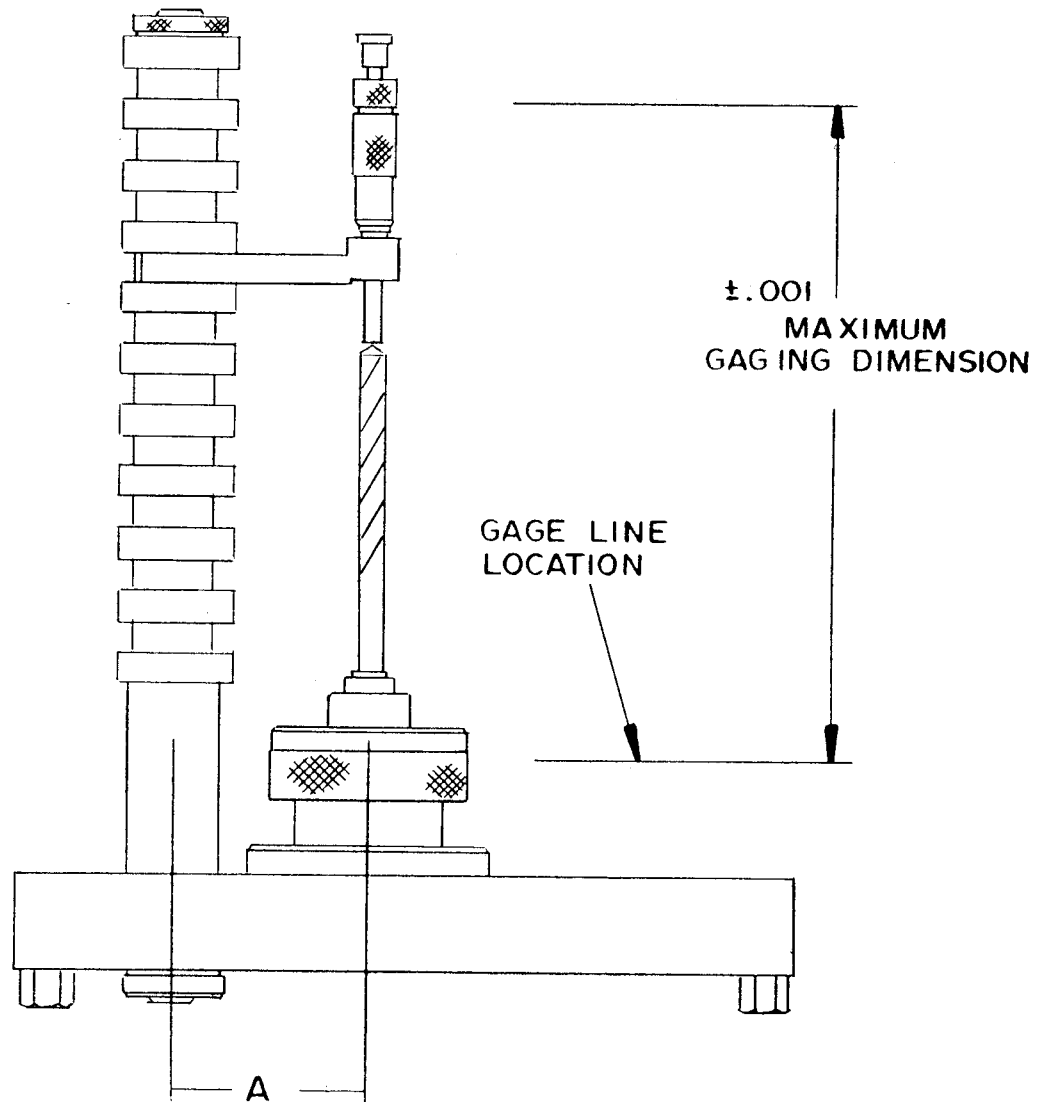
Figure 4-2.   Tool Length Compensation

b. Subtract each tool length from the length of the longest tool. Enter this value as ZV.

c. The Z axis tool length compensation move (ZTLC) can be calculated as follows:

$$ZTLC = ZV \text{ (TOOL NO. n-1)} - ZV \text{ (TOOL NO. n)}$$

For example:

$$ZTLC \text{ (3)} = ZV \text{ (2)} - ZV \text{ (3)}$$
$$= .500 - .250$$
$$= .250$$

d. Each time the operator changes tools, he also manually moves the Z axis the required tool length compensation distance. (Literally, the tool will be set at the Z axis startpoint).

NO.	1.	2.	3.	4.	1.
TOOL	13/32	19/32	19/64	.201	13/32
TOOL MAX.	4.625	4.625	4.625	4.625	4.625
TOOL LG.	4.625	4.125	4.375	3.625	4.625
ZV	0	.500	.250	1.000	0
ZTLC	Z	-.500	.250	-.750	1.000

A modification of this method involves the use of a tool classification system together with the use of the 12-position Automatic Turret Stop. Fixed pre-set stops are used as depth posts in the turret with the following assigned values:

Turret Stop	1	2	3	4	5	6	7	8	9	10	11	12
Quill Travel	.5	1.0	1.5	2.0	2.25	2.5	2.75	3.0	3.25	3.5	3.75	4.0

In the above example, the initial turret stop position would be position 8 (3.0"). For tool 2, the turret stop would be set to position 10 (3.5"); for tool number 3, position 9 (3.25"); for tool number 4, position 12 (4.0") and, at the end of the program, the turret stop would be reset to position 8. Using this method, no Z axis tool length compensation move is required when changing tools.

Method 2: This method enables adjustment for random length tools at run time by using the Z axis mechanical read out counters.

a. With the quill down, establish a Z axis reference plane in air such that the longest tool clears the top of a reference surface. Enter the reading of the Z axis mechanical counter in the table below as ZI.

b. Use a feeler gage and -Z axis moves to bring the top of the reference surface up to within .010" of the tool. Enter the reading of the Z axis mechanical counter in the table below as Z2.

c. Repeat this for each tool using the same Z axis reference plane and reference surface.

d. Tabulate this data in the order that the tools will be used, including the change from the last tool used back to the first tool.

e. Calculate ZV by subtracting Z1 from Z2. The Z axis tool length compensation move (ZTLC) can be calculated as above in Method 1.

f. The operator manually moves the Z axis the required tool length compensation distance.

3. Startpoint Identification. The workpiece can be mounted at any position on the table. In writing up the part program, it is necessary to define the program startpoint. This is the point at which the tool rests before beginning the machining operations. When the workpiece is clamped to a fixture on the table, the distance from the center of the spindle to this zero reference point must be determined. The table must be moved using the manual console controls so that the spindle center is directly above this zero reference point. The X, Y, Z table coordinates of the workpiece zero reference point can be read directly from the mechanical digital counters located on the drive housings. Each time a set-up for a new part is made, this zeroing procedure has to be repeated.

It is suggested that the starting point be selected to allow for loading and unloading the part with minimum interference from the tool and holding fixtures. In general, the startpoint should be clear of the workpiece and at a convenient location for tool load/unload.

4. Operation Sequence. The programmer plans the type and sequence of operations. For example, drill point wander may occur if the surface of the part is not square with the drill or has a rough finish, or if the point of the drill is not rotating at or has been sharpened off its true axis. The best practice is to first spot the hole location with a spot drill, then drill to size with the shortest drill available. Drill bushings are not required.

5. <u>List Necessary Control Data</u>. Using the Programming Sheet (Figure 3-4) all the appropriate machine operations and functions are written in the language of the N/C tape format.

6. <u>List Machining Instructions</u>. Any function not a part of the program will have to be incorporated as part of the machine operator's set of instructions. For example, the operator will be given explicit instructions when to change tools, what tools are to be loaded into the spindle, and what spindle speed to use.

7. <u>Prepare Tape</u>. The part program manuscript is released to a clerk who will then prepare the punched tape.

8. <u>Check Tape</u>. Any programming or typing errors will lead to scrap being produced instead of the desired part. Some errors could be catastrophic. One method of checking the tape is to cycle the control without a tool in the spindle. Other methods include cutting foamed plastic material, three dimensional plots drawn by a high speed drafting machine, CRT displays of the tool path, etc.

PROGRAMMING EXAMPLES

Example No. 1. Point-to-Point Drilling (2 Axis)

A simplified positioning program is shown in Figure 4-3. The spindle is started (m04) and coolant turned on (m07). The tool is then directed from the startpoint to move sequentially to points 1, 2 and 3. At the end of each move, the quill actuator is directed to cycle down and up (m51 code). The tool then returns to the startpoint and the tape is rewound (m02) back to the rewind stop code (RWS).
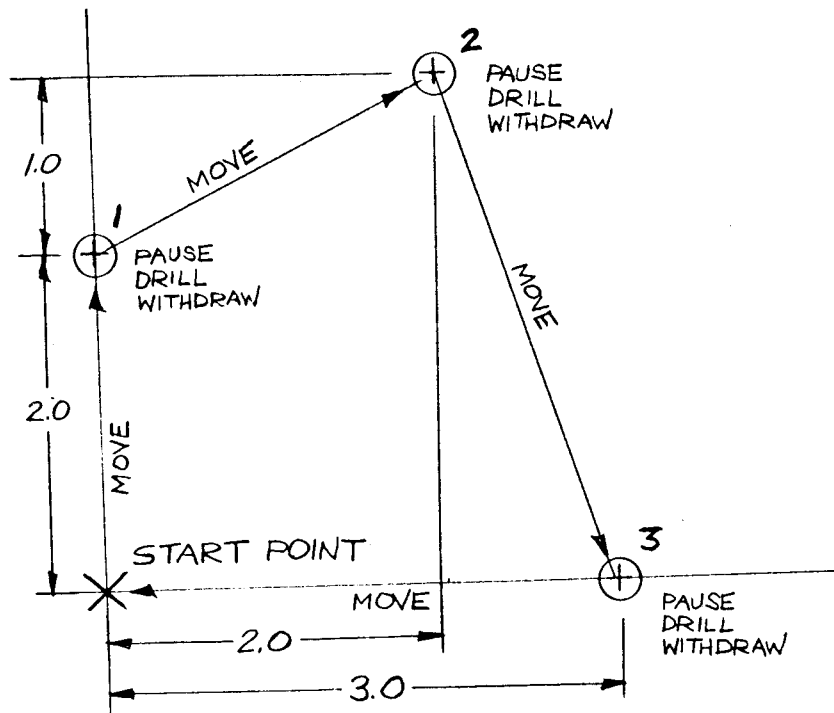
Bridgeport machines

a textron company 500 LINDLEY STREET, BRIDGEPORT, CONN. 06606

CONTINUOUS PATH CONTOURING CONTROL PROGRAM

EXAMPLE # 1

PREPARED BY MK	DATE 8/18/72	REMARKS		JOB NO. 713	PART NO.
CHECKED BY	DATE				PART NAME
SHEET 1 OF 1					COMPANY NAME
TAPE NO.	DEPT.				ADDRESS

N SEQ. NO.	T OR E	X	T OR E	Y	T OR E	Z/I	T OR E	J	T OR E	F FEED RATE	T OR E	M MISC FUN	E	REMARKS
RWS	E										T	4	E	Rewind stop code
	T										T	7	E	Spindle on
	T													Coolant on
1	T	.0	T	2.0	T						O T	51	E	drill hole 1
2	T	2.0	T	1.0	E									drill hole 2
3	T	1.0	T	-3.0	E									drill hole 3
4	T	-3.0	T								T	2	E	returns to startpoint

Figure 4-3. Simplified Point-to-Point Drilling Example

Example No. 2: Point - to - Point Drilling (3 - Axis)

A positioning program using the Z-axis multi-hole machining routine is shown in Figure 4-4.

1. Data blocks 1 & 2 turn the spindle and coolant on.

2. SEQ. NO. 1 - The quill moves down. The knee moves up 1.49" in rapid traverse toward the cutter to within .01" of the surface. At the same time the X axis places the tip of the tool over hole #1.

3. SEQ. NO. 2 - The tool is directly over point 1; the Z-axis drill cycle is used (m51) to drill a hole .5" deep at 6 ipm. The depth motion is the sum of the clearance, full diameter depth and drill point (.01+.5+.075).

4. SEQ. NO. 3 - The tool moves 1.0" +Y in rapid traverse (F=0) to clear the non-stepped portion of the workpiece. Although the 51 knee cycle is set, it is inhibited in this block by the rapid traverse.

5. SEQ. NO. 4 - The tool moves 2.0" +X to point 3; the Z-axis drill cycle is used (m51) to drill a hole .5" deep at 6 ipm. This again requires the full Z motion (.01+.5+.075) to be placed in the block of data.
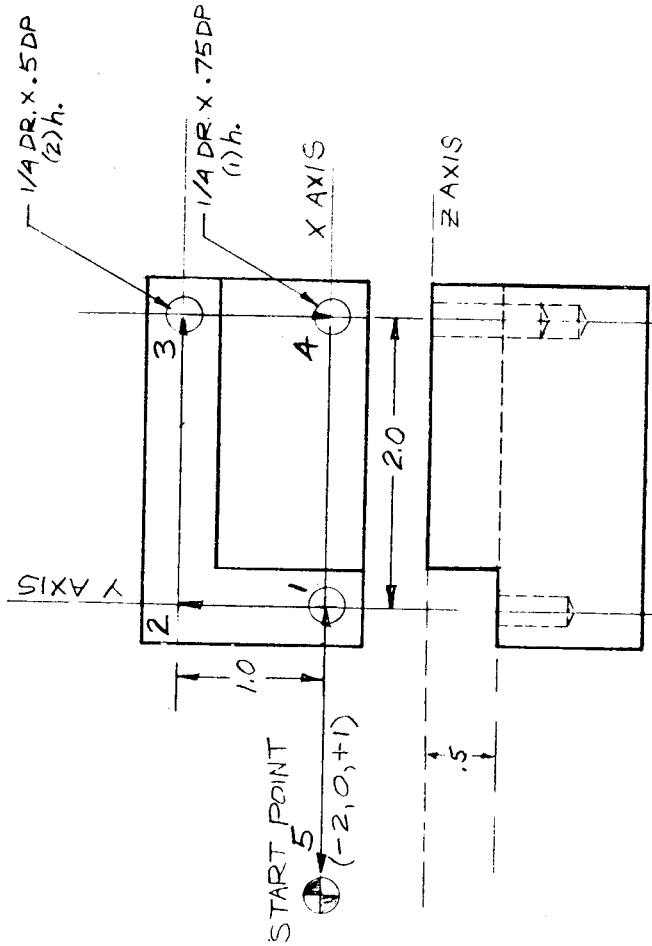
6. SEQ. NO. 5 - The tool moves +Z .5" to a clearance plane .01" above the non-stepped surface.

7. SEQ. NO. 6 - The tool moves to point 4; the Z-axis will cycle .835" to drill a hole .75" deep at 6 ipm. Motion in Z equals (.01+.75+.075).

8. SEQ. NO. 7 - The quill goes up; the tool returns to the startpoint; the tape is rewound (m02) back to the rewind stop code (RWS).

NOTE: Since the Z axis knee cycle has been used, the sum of the Z axis moves will not be zero. The considerations are as follows:

Disregard the Z cyclic moves (.585, .585, .835)
Reminder = -1.49+.5+-.99 Return to Z zero=+.99

Note also that the knee cycle Z axis move must be entered as a positive value.

Figure 4-4. Point-to-Point Drilling (3-Axis)

Bridgeport machines
a **Textron** company · 500 LINDLEY STREET, BRIDGEPORT, CONN. 06606

CONTINUOUS PATH CONTOURING CONTROL PROGRAM

PREPARED BY MK DATE 5/18/72
CHECKED BY DATE
SHEET 1 OF 1
TAPE NO. DEPT.

EXAMPLE #2

JOB NO. 713

PART NO.
PART NAME
COMPANY NAME
ADDRESS

N SEQ. NO.	T OR E	X	T OR E	Y	T OR E	Z/I	T OR E	J	T OR E	F FEED RATE	T OR E	M MISC FUN	E	REMARKS
RMSE											T	4	E	Spindle on
											T	7	E	Coolant on
1	T	2.0	T	.0	T	-1.49	T			0	T	57	E	Quill down more to hole 1
2	T	.0	T	.0	T	.585	T			6	T	51	E	Z drill hole 1
3	T	.0	T	1.0	T	.0	T			0	E			more, dr not drill
4	T	2.0	T	.0	T	.585	T			6	E			Z drill hole 2
5	T	.0	T	.0	T	.5	T			0	E			more, don't drill
6	T	.0	T	-1.0	T	.835	T			6	E			Z drill hole 3
7	T	-4.0	T	.0	T	.99	T			0	T	2	E	return to start point

Example No. 3: End Cap. PARTNO 050115

Purpose: To demonstrate point-to-point programming.

PROGRAM NOTES:

1. Program statements are preceded by an RWS code. The Rewind Stop code (RWS) is an instruction to the tape reader to stop at this point during tape rewind. The RWS code is ignored when the tape is reading normally.

2. Data blocks 1 & 2 turn the spindle and coolant on.

3. SEQNO 1 begins the drill cycle (m51 code). The quill will cycle at the end of this move.

4. SEQNO 2 through 6 - X and Y axis moves to spot drill the remaining 5 holes on this level.

5. SEQNO 7 - Z axis move to next level. The quill will not cycle at the end of this move (m50 code - Cycle/Off).

6. SEQNO 8 through 10 - spot drill 3 holes.

7. SEQNO 11 - return to startpoint. The m06 code will turn the spindle and coolant off, end the drill cycle and stop the control system so that the tool can be changed. When the operator has loaded a new tool, he restarts the unit by pressing the START button. The next three data blocks are necessary to turn the spindle and coolant on and index the turret.

8. SEQNO 13 through 16 - drill four 13/32" holes.

9. SEQNO 17 - return to startpoint, spindle off, coolant off, cycle off, control stop. Load tool 3. Restart control, spindle on, coolant on and index the turret.
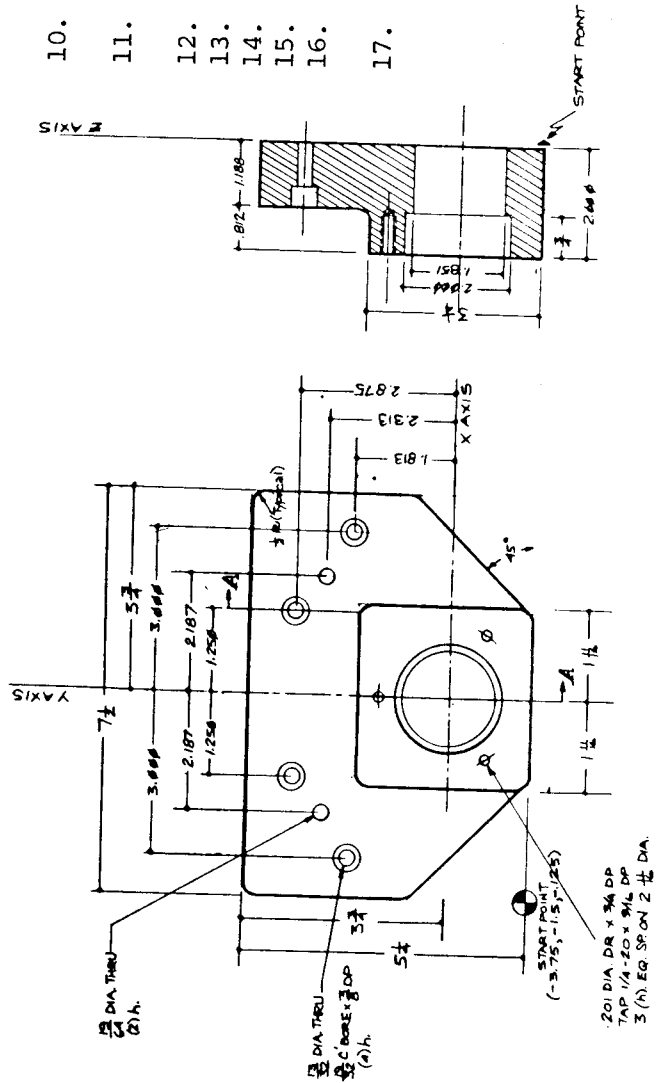
10. SEQNO 19 through 22 - counterbore four 19/32" holes.

11. SEQNO 23 - return to startpoint. Load Tool No. 4, etc.

12. SEQNO 25 and 26 - drill two 19/64" holes.

13. SEQNO 27 - load tool No. 5, etc.

14. SEQNO 29 through 31 - drill three .201" holes.

15. SEQNO 32 - cycle off and return to startpoint.

16. SEQNO 33 through 40 - index turret stop back to position 1.

17. SEQNO 41 - the MO2 code will turn the spindle off, coolant off, and cause the tape to rewind.

Figure 4-5. End Cap - PARTNO 050115

Bridgeport machines

a **Textron** company   500 LINDLEY STREET, BRIDGEPORT, CONN 06606

CONTINUOUS PATH CONTOURING CONTROL PROGRAM

EXAMPLE #3

PREPARED BY	RP	DATE	12/19/73
CHECKED BY		DATE	
SHEET	2	OF	3
TAPE NO.		DEPT.	

JOB NO. 713

PART NO. 050115-1
PART NAME   END CAP
COMPANY NAME
ADDRESS

N SEQ NO	T OR E	X	T OR E	Y	T OR E	Z/I	T OR E	J	T OR E	F FEED RATE	T OR E	M MISC FUN	E	REMARKS
18												4	$	SPINDLE /ON
19											→	7	$	COOLANT /ON
20		.75	→	3.313	→						→	54	$	TURRET STOP 3
21		1.75	→	1.062	$						→	51	$	CYCLE / DRILL CBORE
21		2.75	$											(4) 19/32 H X 3/8 DP.
21		1.75	→	-1.062	$									SET DWELL
23		-6.75	→	-3.313	→						→	6	$	STOP, SPINDLE OFF COOLANT /OFF, LOAD TOOL 4
24											→	4	$	SPINDLE /ON
											→	7	$	COOLANT /ON
25		1.563	→	3.813							→	54	$	TURRET STOP 4
26		4.374	$								→	51	$	CYLCE / DRILL (Z) 19/64"
27		-5.937	→	-3.813										THRU
											→	6	$	STOP, SPINDLE OFF COOLANT /OFF, LOAD TOOL 5
28											→	4	$	SPINDLE /ON
29		2.586	→	.828	$						→	7	$	COOLANT /ON
											→	54	$	TURRET STOP 5
30		1.164	→	2.016	$						→	51	$	
31		1.164	→	-2.016	$									
32		-4.194	→	-.828							→	50	$	CYCLE OFF
														RETURN TO START POINT
33											→	54	$	RETURN TURRET

Form No. F-101

Bridgeport machines
a Textron company   500 LINDLEY STREET, BRIDGEPORT, CONN. 06606

CONTINUOUS PATH CONTOURING CONTROL PROGRAM

EXAMPLE #3

PREPARED BY	RP DATE 12/19/73
CHECKED BY	DATE
SHEET 1 OF 3	
TAPE NO.	DEPT.

REMARKS: S/U START POINT -3.75, -1.5, -.125 FROM ORIGIN.
① SPOT DR. ② DR. (4) 13/32 H THRU ③ C'BORE (4) 19/32 X 3/8 DP. ④ DR. (2) 19/64 H THRU ⑤ DR. (3) .201 H X 3/4 DP.

PART NO. 050115-1
PART NAME: END CAP
COMPANY NAME:
ADDRESS:
JOB NO. 713

N SEQ NO.	T OR E	X	T OR E	Y	T OR E	Z/I	T OR E	J	T OR E	F FEED RATE	T OR E	M MISC FUN	E	REMARKS
RWS														
↑									↑	0	↑	4	$	SPINDLE / ON
↑									↑		↑	7	$	COOLANT / ON
1	↑	.75	↑	3.313					↑		↑	51	$	CYCLE / DRILL (6) H
2	↑	.813	↑	.5	$									.050 DP. ON
3	↑	.937	$	.562	$									LOWER LEVEL
4	↑	2.5			$									
5	↑	.937	↑	-.562	$									
6	↑	.813	↑	-1.5	$									
7	↑	.0	↑	.0	↑									
8	↑	-3.0	↑	-.469	↑	.812			↑		↑	50	$	Z MOVE (UPPER LEV.)
9	↑	1.164	↑	-2.016	$				↑		↑	51	$	CYCLE / DRILL (3) H
10	$	-2.328			$									.050 DP. ON
11	↑	-2.586	↑	-.828	↑	-.812			↑		↑	6	$	UPPER LEVEL / STOP, SPINDLE/OFF, COOLANT/OFF, LOAD TOOL 2
12	↑								↑		↑	4	$	SPINDLE / ON
13	↑	.75	↑	3.313					↑		↑	7	$	COOLANT / ON
14	↑	1.75	↑	1.062	$				↑		↑	54	$	TURRET STOP 2
15	$	2.5			$						↑	51	$	CYCLE / DRILL (4)
16	↑	1.75	↑	1.062										13/32 H THRU
17	↑	-6.75	↑	-3.313					↑		↑	6	$	STOP, SPINDLE/OFF, COOLANT/OFF, LOAD TOOL 3

Form No. F-101

Bridgeport machines

a Textron company   500 LINDLEY STREET, BRIDGEPORT, CONN. 06608

CONTINUOUS PATH CONTOURING CONTROL PROGRAM

EXAMPLE #3

PREPARED BY: RP   DATE 12/19/73
CHECKED BY: DATE
SHEET 3 OF 3
TAPE NO.
DEPT.

PART NO. 050115-1
PART NAME: END CAP
COMPANY NAME
ADDRESS
JOB NO. 713
REMARKS

N SEQ NO	T OR E	X	T OR E	Y	T OR E	Z/I	T OR E	J	T OR E	F FEED RATE	T OR E	M MISC FUN	E	REMARKS
34											1	54	$	STOP TO 1
35											1	54	$	
36											1	54	$	
37											1	54	$	
38											1	54	$	
39											1	54	$	
40											1	54	$	STOP, SPINDLE OFF,
41											1	2	$	COOLANT OFF, REWIND

4-13

Form No. F-101

Example No. 4: End Cap, PARTNO 050115

Purpose: To demonstrate the Z axis multi-hole routine.

PROGRAM NOTES:

1. This program demonstrates machining the previous part using the "canned" knee routine. The quill actuator is not used. If it is available, the program could be somewhat simplified by using the quill actuator to move the tool to and from the work surface at each tool change.

2. SEQ. NO. 1 - moves the knee to the proper depth for spot drilling 6 holes at level 1, allowing .05" clearance for the tool (.125+1.188+.05).

3. SEQ. NO. 2 - sets the control in the Z axis drill cycle (m51) with a drill feed of 20ipm (f20) and spot drills the first hole to a depth of .05".

4. SEQ NO. 3 through 7 - spot drills the remaining 5 holes on level 1 .05" deep +.05" clearance,

5. SEQ. NO. 8 - rapid traverse to level 2. Note that the control is still in the mode "z" axis drill cycle routine" but using a rapid traverse feedrate (f0) overrides the cycle.

6. SEQ. NO. 9 through 11 - spot drills 3 holes .05" deep +.05" clearance. Note that a feedrate of 20ipm (F20) resets the control in the Z axis drill cycle mode.

7. SEQ. NO. 12 - returns the tool to the XY startpoint and turns the knee drill cycle off. Note that X, Y and Z data must be entered in the block of information, ending the cycle (m50) even though the Z move =.0". The table then moves down 3.0" (+Z) to provide clearance for insertion of a new tool. The m06 code sets the control in the "tool change" condition (Program Stop, Spindle Inhibit).

8. SEQ. NO. 13 - moves the tool clearance plane .05" above to level 1. The control is then set in the knee drill cycle (m51) with a drill feed of 4ipm (f4). The first 13/32" thru hole is drilled, allowing the tool to break through the workpiece by .05". Z = 1.188+.125+.05+.05 = 1.413.

9. SEQ. NO. 14 through 16 - drills the remaining three 13/32 thru holes.

10. SEQ. NO. 17 - ends the drill cycle, moves the tool 3" above the part (+Z = 3.812), returns the tool to the XY startpoint, stops the machine (spindle inhibit) for a tool change. After the operator restarts the machine, the tool is moved to within .05" of the work surface.

11. SEQ. NO. 19 through. 22 - The control is set in the knee drill cycle with a drill feed of 4ipm. Counterbores four 19/32" holes .375 deep +.05 clearance.

12. SEQ. NO. 23 - ends the drill cycle, moves the tool 3" above the part, returns the tool to the XY startpoint and stops the machine for a tool change. After the operator restarts the machine, the tool is moved to within .05" of the work surface.

13. SEQ. NO. 24 and 25 - drills two 19/64" thru holes (1.188+.05+.069+.05 = 1.377).

14. SEQ. NO. 26 - ends the drill cycle, etc. See SEQ. NO. 23.

15. SEQ. NO. 27 through 30 - drills three .201 holes .75 deep (.75+.06+.05 = .86).

16. SEQ. NO. 31 - ends the drill cycle and returns the tool to the XY startpoint. A "Z" move of -2.175" returns the tool to the Z startpoint, while the tape is being rewound. Note that the sequence of returning to the startpoint avoids damaging the workpiece.

NOTE: All tools are set to the same length for this demonstration.

Bridgeport machines
a **textron** company · 500 LINDLEY STREET, BRIDGEPORT, CONN. 06606

CONTINUOUS PATH CONTOURING CONTROL PROGRAM

EXAMPLE #4

PREPARED	DATE	CHECKED BY	DATE	SHEET	TAPE NO.	REMARKS	JOB NO.	PART NO.	PART NAME	COMPANY NAME / ADDRESS
MK	8/18/72			1 OF 2		Demonstrate Z Axis Multi hole drill routine	713	050115-2	END CAP	

N SEQ. NO.	T OR E	X	T OR E	Y	T OR E	Z/I	T OR E	J	T OR E	F FEED RATE	T OR E	M MISC. FUN.	E	REMARKS
RMST														
1	T	.0	T	.0	T	1.363	T			0	T	4	E	spindle on
2	T	.75	T	3.313	T	.1	T			20	0 T	7 · 51	E	Coolant on, rapid to clearance plane — Z axis drill 6(h) X
3	T	.813	T	.5	E									.05 dpt +.05 clearance
4	T	.937	T	.562	E									@ 20 ipm
5	T	2.5	T	.0	E									
6	T	.937	T	-.562	E									
7	T	.813	T	-.5	E									
8	T	.0	T	.0	T	.812	T			0	E			rapid to new clearance plane
9	T	-3.0	T	-.469	T	.1	T			20	E			Z axis drill (3)h X.05p +.05 clearance @ 20 ipm
10	T	1.164	T	-2.016	E									
11	T	-2.328	T	.0	E									
12	T	-2.586	T	-.828	T	.0	T			0	T	50	E	cycle off / return to XY SP
	T	.0	T	.0	T	3.0	T					6	E	stop (part) L2
13	T	.0	T	.0	T	-3.812	T					4	E	spindle on
14	T	.75	T	3.313	T	1.413	T					7	E	coolant on
15	T	1.75	T	1.062	E						4 T	51	E	Z drill (4) 13/32h X time
16	T	2.5	T	.0	E									
17	T	-1.75	T	1.062	E									
	T	.0	T	.0	T	3.812	T			0	T	50	E	end drill cycle
	T	-6.75	T	-3.313	T							6	E	return XY sp (part) L3

Bridgeport machines
· textron company · 500 LINDLEY STREET, BRIDGEPORT, CONN. 06606

CONTINUOUS PATH CONTOURING CONTROL PROGRAM

PREPARED BY: MK DATE 9/18/72
CHECKED BY: ____ DATE ____
SHEET 2 OF 2
TAPE NO. ____ DEPT. ____

REMARKS: ____

PART NO. 050115-2
PART NAME: END CAP
COMPANY NAME: ____
ADDRESS: ____
JOB NO. 713

N SEQ. NO.	T/E	X	T/E	Y	T/E	Z/I	T/E	J	T/E	F FEED RATE	T/E	M MISC FUN	E	REMARKS
	T		T		T		T				T	4	E	spindle on
19	T	.0	T	.0	T	-3.812	T				T	7	E	rotary transplane / near to transplane
20	T	.75	T	3.313	T	.425	T			4	T	5I	E	c'bore (4) 19/32 h x .375
21	T	1.75	T	1.062	E									+.05 clearance @ 4 ipm
22	T	2.5	T	.0	E									
	T	1.75	T	-1.062	E									
23	T	.0	T	.0	T	3.812	T			0	T	50	E	end drill cycle
											T	6	E	stop lead 1/4
	T	-6.75	T	-3.313	T						T	4	E	spindle on
24	T	.0	T	.0	T	-3.812	T				T	7	E	rotary transplane / near to transplane
	T	1.563	T	3.813	T	1.377	T			4	T	5I	E	drill (2) 19/64 thru
25	T	4.374	E	.0										
26	T	.0	T	.0	T	3.812	T			0	T	50	E	end drill cycle
	T	-5.937	T	-3.813	T						T	6	E	stop lead 1/5
	T	.0	T	.0	T						T	4	E	spindle on
28	T	2.586	T	.828	T	-3.0	T				T	7	E	rotary transplane / near to transplane
29	T	1.164	E	2.016	E	.86	T			6	T	5I	E	drill (3) .201 h
30	T	1.164	T	-2.016	E									
31	T	-4.194	T	-.828	T	.0	T			0	T	50	E	return to XY SP
	T	.0	T	.0	T	-2.175	T				T	2	E	return to start point

4-16

4.2   CONTINUOUS PATH PROGRAMMING.

For continuous path programming, the programmer follows the same steps and is guided by the same overall principles as in programming positioning work. The engineering drawing is studied, a manuscript is written, a clerk prepares a program tape and the manuscript and tape are checked for errors.

One of the principal tasks facing the programmer will be that of allowing for proper cutter offset. The programmed coordinate points, as in positioning, locate the center of the spindle, which is normally the center line of the cutting tool, but the final machined configuration is made by the cutting tool edge.

In drilling a hole, this is no problem. The center of the hole is the programmed drill location and the hole diameter is determined by the diameter of the drill. When contour milling, the path of the milling cutter centerpoint, which normally corresponds to the spindle center line, and the outline of the workpiece are not the same. Figure 4-6 illustrates how an allowance must be made for the necessary cutter offset. Such an offset is not too difficult when programming straight-line cuts that are exactly parallel to one of the three machine axes. The offset is perpendicular to the axis. Offsets must always be perpendicular to the surface being machined regardless of the direction that surface faces. Calculating offsets not parallel to a machine axis will involve trigonometric calculations.

Some other continuous path programming considerations are:

1.   End Mill Diameter:
       The part programmer, when calling for a type and size of end mill on any operation sheet, must recognize factory tolerances on mills.

       HHS
              2 and 3 flute (Regular and Stub length) + .000/-.0015
              2 and 3 flute (Long and Extra Long) + .005/-.0015
              4 or more flutes (Not center cutting) + .005/-.0015
              Ditto double end mills shank equals flute diameter
              (Center cutting) + .000/-.0025
       Solid Carbide
              Up to and including 1/4" diameter + .000/-.002
              Over 1/4" and including 3/4" diameter + .000/-.003

       These mills will be concentric as received from the factory but if a special diameter is requested, ensure that it be cylindrically ground to maintain perfect concentricity with the shank.

The toolpath that will be developed by the control system is at a normal distance from the part surface equal to the radius of the cutting tool. Two methods can be used to execute roughing passes:

a. A modified tool radius can be substituted into the toolpath calculation. For example, if the tool diameter for the finish cut is 1.000" and it is desired to leave .060" material in the roughing pass for finish milling, the tool diameter for making the toolpath calculations would be 1.12". The same 1.000" diameter tool would actually be used for both the rough milling and finish milling passes. However, two different input tapes would have to be prepared, one for roughing, one for finishing.

b. An undersize tool can be used for roughing which follows the finish pass toolpath. The tool is undersize in diameter by twice the amount of stock to be left in the finish pass. Referring to the previous sample, an .88 diameter tool used with the finish pass tape will leave approximately .060" in excess material.

2. Climb Milling vs. Conventional Milling:
Figure 4-7 shows the relationship between cutter rotation and feed direction in climb milling and in conventional milling.

In climb milling, chips are cut to maximum thickness at the initial engagement of the cutter teeth with the work, and decrease to zero thickness at the end of the engagement - in conventional milling, the reverse occurs.

Climb milling (which requires a rigid machine with minimum backlash such as the Bridgeport N/C Series II Miller) has several advantages over conventional milling:

a. Chips are less likely to be carried by the tooth, reducing the possibility of marring the machined surface.

b. Chip disposal is easier, because chips pile up behind the cutter rather than in front of it.

c. Cutter wear is less, because chip thickness is maximum at the start of the cut.
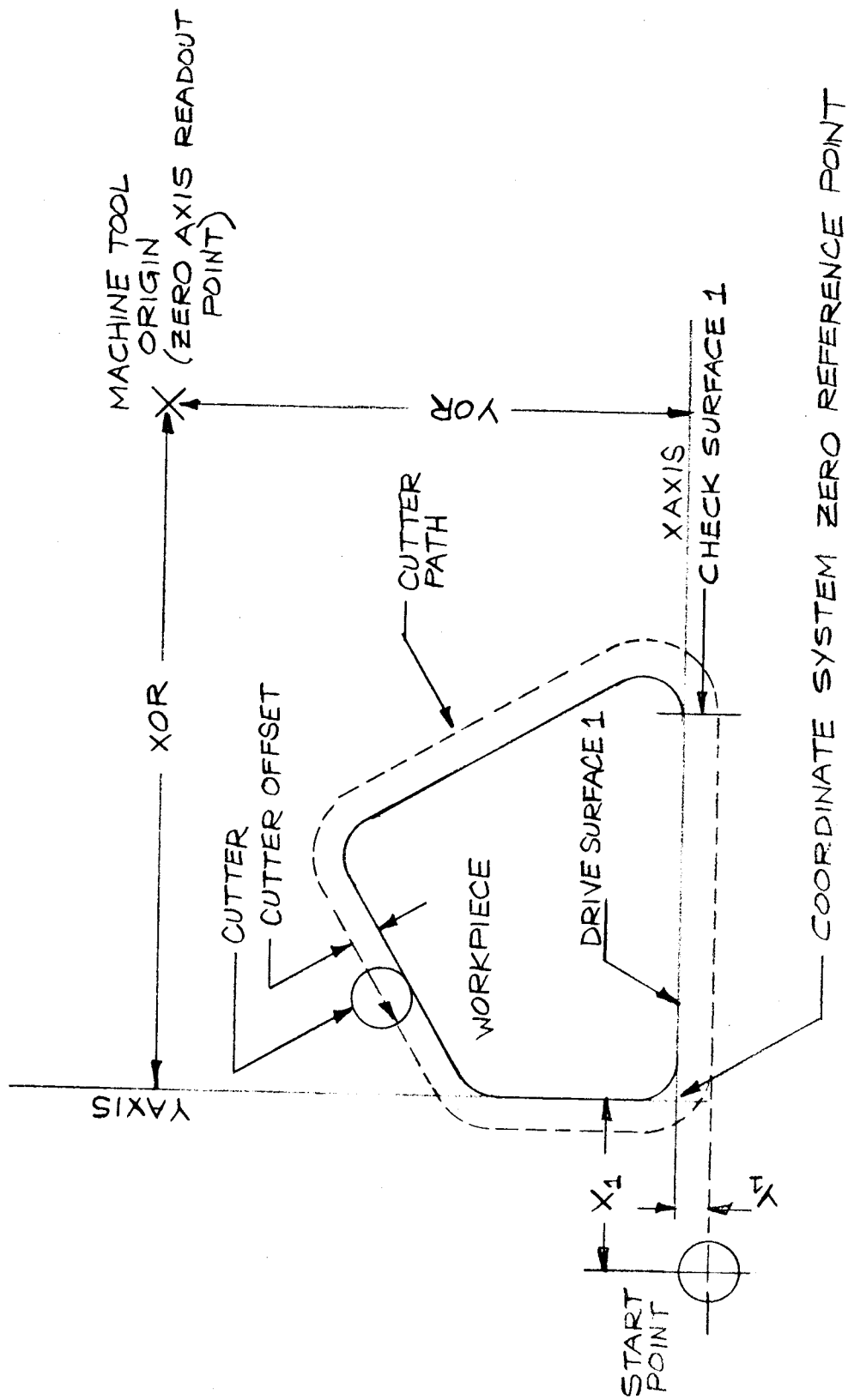
Figure 4-6.   Allowance for Cutter Offset

The main advantage of conventional milling is in the lower impact encountered at the initial tooth-workpiece engagement. Furthermore, the direction of milling force compensates for the backlash of the table feed mechanism, such as that present in standard (as opposed to N/C) machines. However, with numerical control, conventional milling is still preferred in:

    a.   Milling of surfaces on which depth of cut varies by more than 20%.

    b.   Milling of castings or forgings with very rough surfaces due to sand or scale.

3.   <u>Corners</u>:

Climb milling in internal corners with an endmill diameter to suit the corner radius is poor practice since the mill will tend to whip and pull into the corner. For instance, the use of a .5 diameter mill to cut two sides blended by a 1/4 radius will not yield the best radius for size and finish. It is better practice to program an undersize mill (say 7/16" diameter) and contour a 0.031" toolpath radius if two sides are mutually perpendicular.
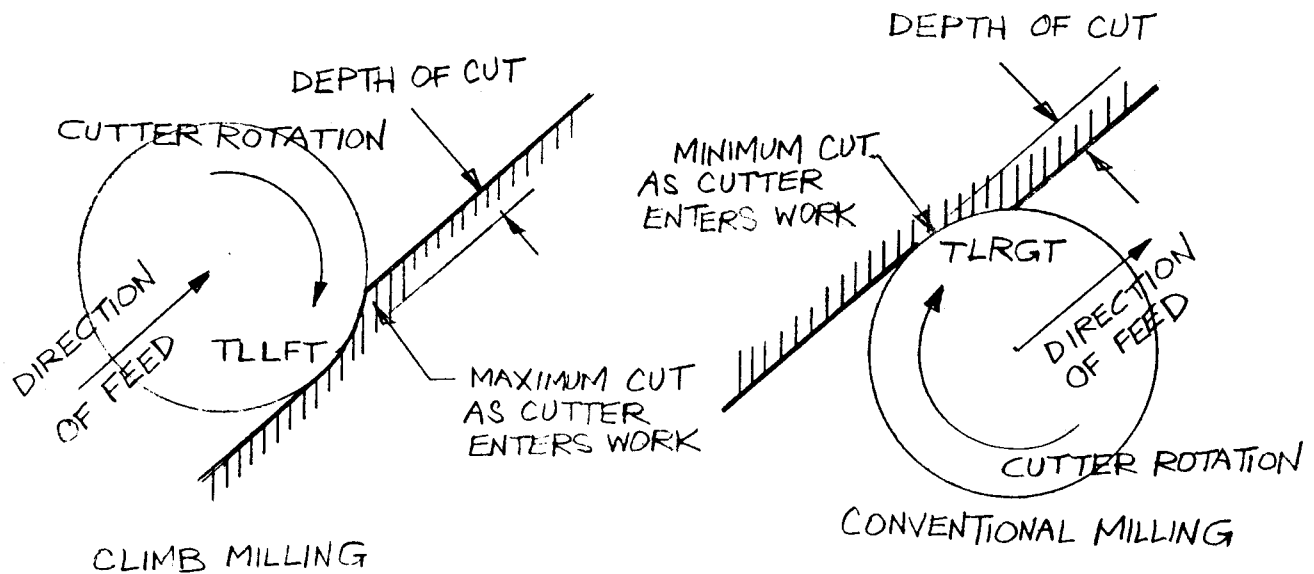

Figure 4-7 Climb Milling vs. Conventional Milling

4. Scallop Heights:
    Using a ball end mill will cause "scallops" between successive milling
    passes. The following example will show how to calculate the space
    between ball mill passes relative to an allowable scallop height.

    $$S=2*(SQRT(2*R*H))$$ where: R=tool radius
    S=Space between Centers   H=Allowable scallop height

    If the allowable scallop height = .01" and a .25 RADIUS ball
    end mill is used, the space between centers = 2* (SQRT (2*.25*.01))
    = .14" (approximately).

5. Cutter Deflection:
    End milling is characterized by the unsupported cutter length. As this
    unsupported length increases, it becomes more difficult to maintain
    dimensional accuracy because of cutter deflection. For most applica-
    tions, an unsupported length of more than five diameters is excessive.
    Cutter deflection can be reduced by using solid carbide cutters which
    have a greater modulus of rigidity than high speed steel.

6. Machinability Data:
    Sources of accurate and applicable machinability data are necessary
    for success in N/C machining. This includes all the variables which
    interact and affect machining performance such as:

    a. Cutting tool design and material

    b. Tooling characteristics

    c. Workpiece properties

    d. Machine – tool characteristics

    e. Cut dimensions

    f. Cutting fluids

    The following reference materials provides information about the rela-
    tive machinability of various materials with data giving recommendations
    for cutting speeds, feed rates, tool materials, tool types and geometries,
    cutting fluids, etc.

    a. N/C MACHINABILITY DATA SYSTEMS. (Society of Manu-
        facturing Engineers, 20501 Ford Road, Dearborn, Michigan 48128).
        This book summarizes the present current available information
        on machinability data systems.

b. MACHINING DATA HANDBOOK. (METCUT Research Asso-
ciates, Inc., 3980 Rosslyn Drive, Cincinnati, Ohio 45209 –
$35.00). The machining data contained in this handbook
provides starting recommendations for all types of material
removal operations and materials. Data for the handbook
was collected with the help of the U.S. Army Weapons Command
Metal Cutting Advisory Committee, and approximately 250
companies in industry.

c. MACHINING DATA FOR NUMERICAL CONTROL. (Air Force
Machinability Data Center, 3980 Rosslyn Drive, Cincinnati,
Ohio 45209). These reports (designated AFMDC 66-1 and 68-2)
include N/C machining data for turning, face milling, drilling,
peripheral end milling, end-mill slotting, tapping and reaming.

d. AIR FORCE MACHINABILITY DATA CENTER.

(1) Scope.

The Air Force Machinability Data Center (AFMDC) collects,
evaluates, stores, and disseminates material removal informa-
tion including specific and detailed machining data for the
benefit of industry and government. Strong emphasis is given
to engineering evaluation for the purpose of developing optimized
material removal parameters, such as speeds, feeds, depths
of cut, tool material and geometry, cutting fluids and other
significant variables. Data are being processed for all types
of materials and for all kinds of material removal operations
such as turning, milling, drilling, tapping, grinding, elec-
trical discharge machining, electrochemical machining, etc.
Services are available without charge to the aerospace industry,
Department of Defense (including all of the military technical
institutions, and nonmilitary industries in a position to assist
the defense effort.)

To request specific machining information......

        Telephone: 513-271-9510
        TWX: 810-461-2840
        Write: Air Force Machinability Data Center
                     3980 Rosslyn Drive
                     Cincinnati, Ohio 45209

e. METALS HANDBOOK, eighth Edition. Volume III
    MACHINING. (American Society for Metals,
    Metals Park, Ohio 44073). The text contains
    1099 tables of machining data, 2836 illustrations,
    and 1009 examples that describe actual machining
    procedures or problems solved, and that give
    details of selection of equipment, tools and
    process variables.

7. Feedrate Modification
    The ratio of the cutter tool path radius to the part surface radius affects
    the cutting rate since the vector velocity is that at the center of the
    cutter, not at the surface of the material. For cutting around the out-
    side of a circle, the feedrate number should be increased by a factor
    of (PR+CR)/PR. For cutting around the inside of a circle, the feed-
    rate number should be increased by (PR-CR)/PR (where PR is the
    part surface radius, CR is the cutter radius), Example No. 5 shows
    the calculation of feedrate number to maintain a relatively constant
    surface speed. See Figure 4-8.

8. Geometry and Trigonometry
    In order to perform the necessary calculations to determine the XY
    tool motions around a profile, some knowledge of geometry and trig-
    onometry is required. Fundamentally, a good visualization of right
    angle triangles and their solution is all that is necessary. (See
    Figures 4-9 and 4-10). As an extension of this requirement, Figure 4-11,
    A, B and C illustrate the general solution of line-line intersection.
    Other examples in this manual show line-circle intersection and the
    computer programming discussion (Section V) contains a new discipline
    where trigonometry is not usually required.

9. Linear and Circular Interpolation
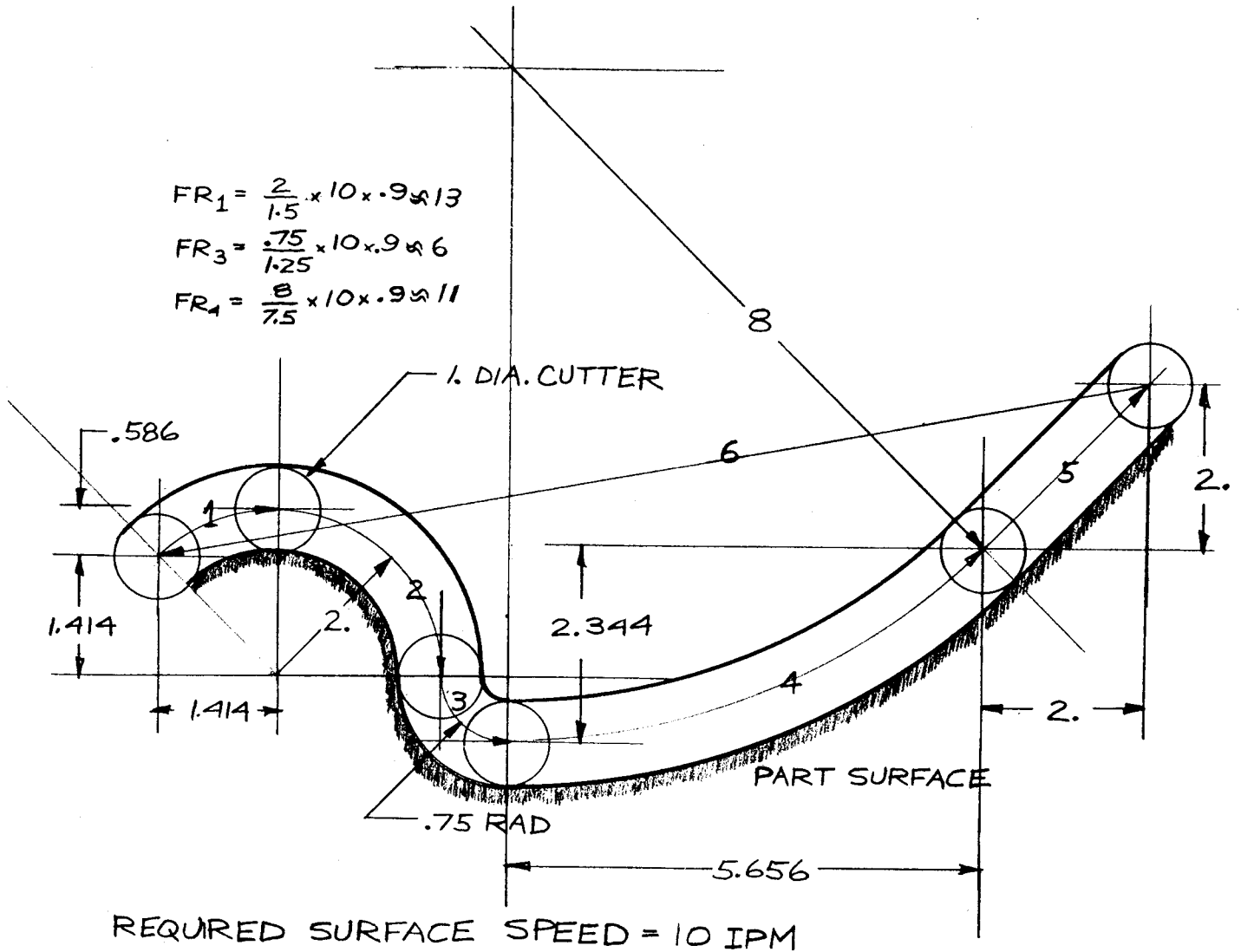   Example No. 6 (Figure 4-12) has been programmed to show linear and
    circular interpolation in the XY plane. This example uses data
    tabulated in the notes to assist the programmer. Example 7 (Figure 4-13)
    requires the programmer to solve various triangles to obtain the
    solutions shown before proceeding with the linear and circular motions
    to be programmed.

Purpose: To illustrate the use of:

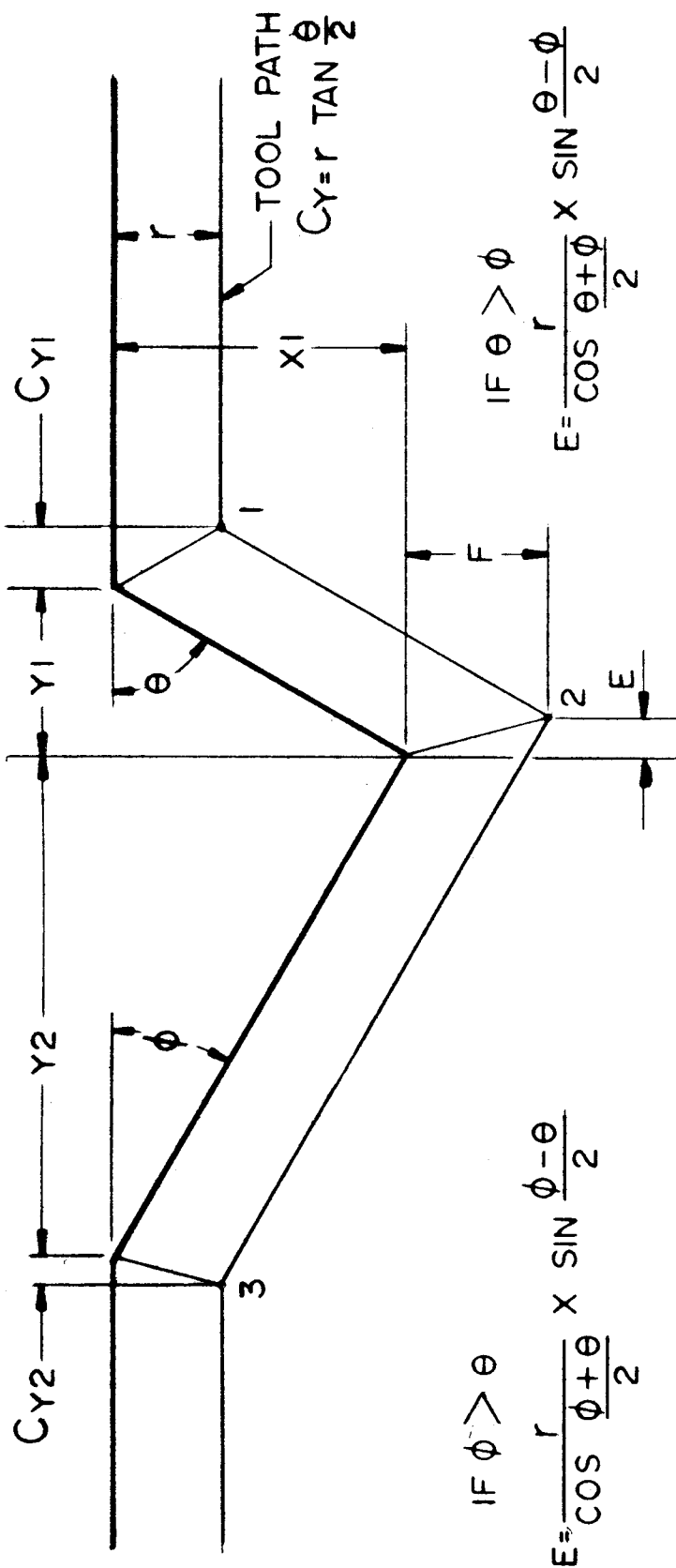1.   Radius Format
2.   Feedrate Correction
3.   Acceleration/Deceleration Override

Example No. 5. Contour Milling a Slot

$$FR_1 = \frac{2}{1.5} \times 10 \times .9 \approx 13$$

$$FR_3 = \frac{.75}{1.25} \times 10 \times .9 \approx 6$$

$$FR_4 = \frac{8}{7.5} \times 10 \times .9 \approx 11$$

REQUIRED SURFACE SPEED = 10 IPM

N SEQ NO	T E	X	T E	Y	T E	Z/I	T E	J	T E	F FEED RATE	T E	M MISC FUN.	T E
1	,	1.414	,	.586	,	−1.414	,	1.414	,	13-	,	57	$
2	,	• 2.0	,	−2.0	,	.0	,	2.0	,	−	$		
3	,	.75	,	−.75	,	−.75	,	.0	,	6-	$		
4	,	5.656	,	2.344	,	.0	,	−8.0	,	11-	$		
5	,	2.0	,	2.0	,					10	$		
6	,	−11.82	,	−2.18	,					0	,	6	$

Figure 4-8.   Contour Milling a Slot

TOOL PATH $C_Y = r \ \text{TAN} \ \dfrac{\theta}{2}$

IF $\theta > \phi$

$$E = \frac{r}{\cos\frac{\theta+\phi}{2}} \times \sin\frac{\theta-\phi}{2}$$

$$F = \frac{r}{\cos\frac{\theta+\phi}{2}} \times \cos\frac{\theta-\phi}{2}$$

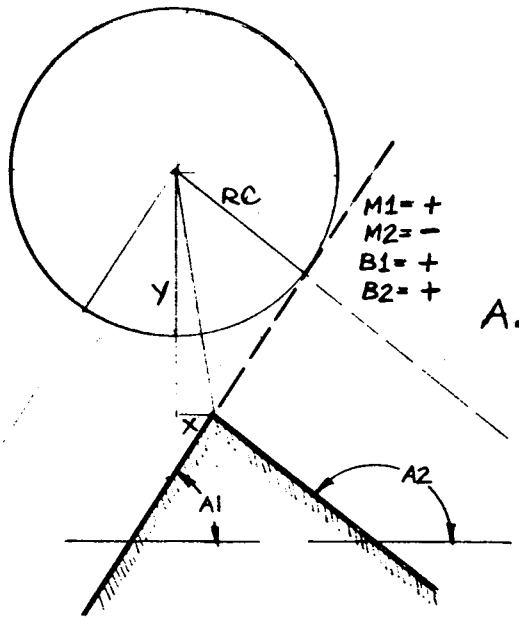IF $\phi > \theta$

$$E = \frac{r}{\cos\frac{\phi+\theta}{2}} \times \sin\frac{\phi-\theta}{2}$$

$$F = \frac{r}{\cos\frac{\phi+\theta}{2}} \times \cos\frac{\phi-\theta}{2}$$

PROGRAM (for $\theta > \phi$)

FROM	TO	Y AXIS	X AXIS
1	2	$Y_1 + C_{Y_1} - E$	$X_1 - r + F$
2	3	$Y_2 + E + C_{Y_2}$	$X_1 - r + F$

Figure 4-9. Trigonometry Example

$$C_Y = r\,\tan\frac{\theta}{2}$$

TOOL PATH

IF $\theta > \phi$

$$E = \frac{r}{\cos\dfrac{\theta+\phi}{2}} \times \sin\frac{\theta-\phi}{2}$$

$$F = \frac{r}{\cos\dfrac{\theta+\phi}{2}} \times \cos\frac{\theta-\phi}{2}$$

IF $\phi > \theta$

$$E = \frac{r}{\cos\dfrac{\phi+O}{2}} \times \sin\frac{\phi-\theta}{2}$$

$$F = \frac{r}{\cos\dfrac{\phi+\theta}{2}} \times \cos\frac{\phi-\theta}{2}$$

PROGRAM (for $\theta < \phi$)

FROM	TO	Y AXIS	X AXIS
1	2	$Y_1 - C_{Y1} - F$	$X_1 + r - F$
2	3	$Y_2 + E - C_{Y2}$	$X_1 + r - F$

$M = TAN(A)$ A is measured CCLW from X axis
if $0 < A < 90$ M is positive
if $90 < A < 180$ M is negative
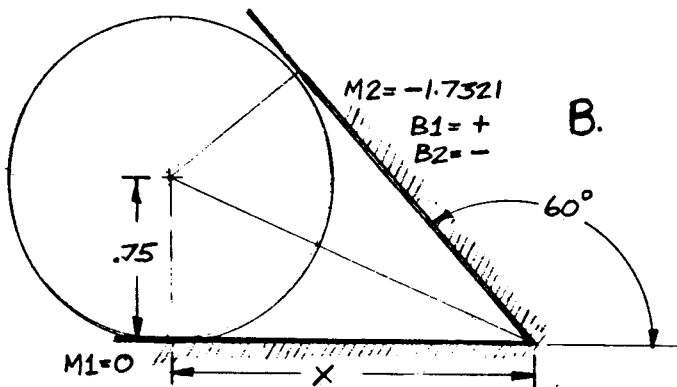
$B1 = RC * SQRT(1 + M1 * M1)$ or $RC * SEC(A1)$
$B2 = RC * SQRT(1 + M2 * M2)$ or $RC * SEC(A2)$

If the normal from the cutter center to the line goes down (ie, the cutter is above the line), B is positive. If the normal from the cutter center to the line goes up (ie, the cutter is below the line), B is negative.
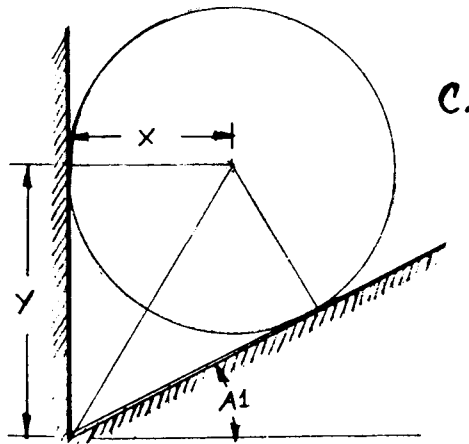
$X = (B2 - B1) / (M1 - M2)$
$Y = M1 * X + B1$

EXAMPLE

$B1 = .75 + SEC(0) = .75$
$B2 = .75 * SEC(60) = -1.5$

$X = (-1.5 - .75) / (0 - (-) 1.7321)$
$X = -2.25 / 1.7321 = -1.2990$

$Y = .75$ (by inspection)

In the special case shown $A2 = 90°$
Therefore $TAN(A2)$ & $SEC(A2) \rightarrow \infty$

$X = RC$ (by inspection)
$B1 = RC * SEC(A1)$
$Y = RC * (TAN(A1) + SEC(A1))$

Figure 4-11.   General Solution of Line-Line Intersection

Example No. 6. Valve Seat, PARTNO 050118

Purpose: To demonstrate milling using 2-axis linear and circular interpolation.

PROGRAM NOTES:

1. The program is written to drill (12) 5/16 thru holes and then mill 6 arcs. The milling feedrate will be 2 ipm. The coordinates of P1 through P12 can be calculated as:

P1	−1.532	−1.2856
P2	−.3472	−1.9696
P3	.3472	−1.9696
P4	1.532	−1.2856
P5	1.8794	−.684
P6	1.8794	.684
P7	1.532	1.2856
P8	.3472	1.9696
P9	−.3472	1.9696
P10	−1.532	1.2856
P11	−1.8794	.684
P12	−1.8794	−.684

2. Data blocks 1 and 2 turn the spindle and coolant on.
3. SEQNO 1 - Moves the tool from the start point to the first hole where the hole is drilled automatically (m51). The feedrate will be rapid traverse (f0).
4. SEQNO 2 through 12 - Drills the remaining 11 holes.
5. SEQNO 13 - Return to startpoint. The MO6 Code will turn the spindle and coolant off, end the drill cycle and stop the control system so that the tool can be changed. The next two data blocks are necessary to turn the spindle and coolant on.
6. SEQNO 14 - Rapid move to P1.
7. SEQNO 15 - this statement provides the X and Y axis tool motions from P1 to P2 (note that they are the same as the point-to-point values between P1 and P2, SEQNO2) and include the I and J values which locate the center of the arc. The f2 code will set the feedrate at 2 ipm. The M57 code will cause the tool to be fed into the work before positioning along the arc begins.
8. SEQNO 16 - Raises the tool (M50) and moves it at rapid Traverse (f0) to the beginning of the second arc.
9. SEQNO 17 - mills the second arc.
10. SEQNO 18 - Raises the tool and moves it at rapid traverse to the beginning of the third arc.
11. SEQNO 19 - The arc P5-P6 straddles two quadrants. Therefore two blocks of information are required to mill the arc from P5 to P6. The first block mills from P5 to the X axis, the second block mills from the X axis to P6. SEQNO 19 sets the feedrate at 2 ipm and lowers the tool into the work before positioning along the arc begins.
12. SEQNO 20 - Mills the portion of the arc P5-P6 that is in the first quadrant. Note the I and J Values of 2.0,.0 respectively.
13. SEQNO 21 through 27 - Provides the information to mill the remaining arcs.
14. SEQNO 28 - raises the tool, returns it to the starting point at rapid traverse (f0) and rewinds the tape.
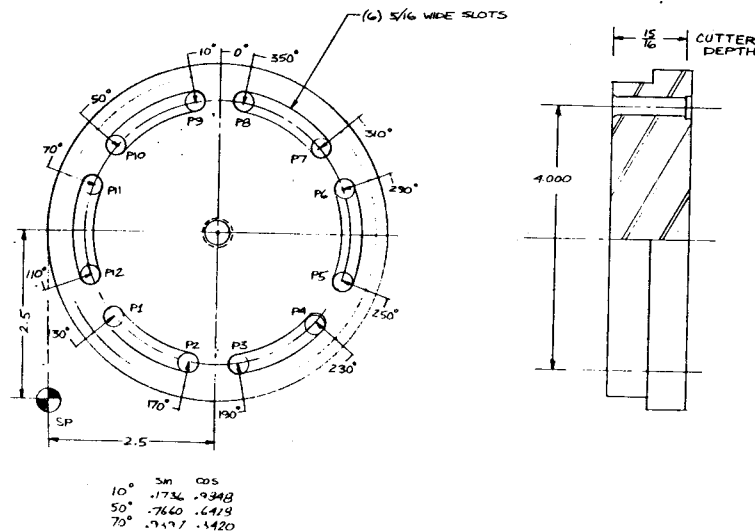

Figure 4-12. Valve Seat, PARTNO 050118

CONTINUOUS PATH CONTOURING CONTROL PROGRAM

EXAMPLE #6

PREPARED BY	MK	DATE 7/13/71
CHECKED BY	/	DATE
SHEET	/	OF
TAPE NO.		DEPT.

JOB NO. 7/3

PART NO. 050118
PART NAME: VALVE SEAT
COMPANY NAME
ADDRESS

REMARKS: Start point -2.5, -2.5 † origin
dr (12) 5/16 h, mill (6) 5/16 arcs

SEQ. NO.	T OR E	X	T OR E	Y	T OR E	Z/1	T OR E	J	T OR E	F FEED RATE	T OR E	M MISC FUN.	E	REMARKS
RMS												4	†	spindle/on
												7	†	coolant/on
1	↗	.968	↗	1.2144						0	↗	51	†	cycle/drill (12) 5/16 h
2	↗	1.1848	↗	-.684	†									
3	↗	.6944	†	.684	↗									
4	↗	1.1848	↗	.684	†									
5	↗	.3474	↗	.6016	†									
6	↗	0	↗	1.362	†									
7	↗	-.3474	↗	.6016	†									
8	↗	-1.1848	†	.684	†									
9	↗	-.6944	†	-.684	†									
10	↗	-1.1848	↗	-.684	†									
11	↗	-.3474	↗	-.6016	†									
12	↗	0	↗	-1.362	↗									
13	↗	-.6206	↗	-1.816	↗								6	†	quick off/ GOTO/SP
												4	†	spindle/on
											↗	7	†	coolant/on
14	↗	.968	↗	1.2144	↗					0	†			
15	↗	1.1848	↗	-.684	↗	-1.532	↗	-1.2856	↗	2		57	†	mill slot P1-P2
16	↗	.6944	↗							0	↗	50	†	rapid to P3
17	↗	1.1848	↗	.684	↗	.3472	↗	-1.9696	↗	2		57	†	mill slot P3-P4
18	↗	.3474	↗	.6016	↗					0	↗	50	†	rapid to P5

4-29

Bridgeport machines

(textron) company 500 LINDLEY STREET, BRIDGEPORT, CONN. 06606

CONTINUOUS PATH CONTOURING CONTROL PROGRAM

EXAMPLE # 6

PREPARED BY	CHECKED BY	SHEET	TAPE NO.	REMARKS	JOB NO.	PART NO.	PART NAME	COMPANY NAME	ADDRESS
MK DATE 5/14/1	DATE	2 OF 2 DEPT.		Cont'd	713	050118	VALVE SEAT		

N SEQ. NO.	T OR E	X	T OR E	Y	T OR E	Z/!	T OR E	J	T OR E	F FEED RATE	T OR E	M MISC FUN.	E	REMARKS
19	→	.1206	→	.684	→	1.8794	→	-.684	→	2	→	57	4	mill slot P5-X axis
20	→	-.1206	→	.684	→	2.0	→	.0	5		→			Mill X axis-P6
21	→	-.3474	→	.6016	→				→	50	→	50	4	rapid to P7
22	→	-1.1848	→	.684	→	1.532	→	1.2856	→	2	→	57	4	mill slot P7-P8
23	→	-.6944	→						→	0	→	50	4	rapid to P9
24	→	-1.1848	→	-.684	→	-.3472	→	1.9696	→	2	→	57	4	mill slot P9-P10
25	→	-.3474	→	-.6016	→				→	0	→	50	4	rapid to P11
26	→	-.1206	→	-.684	→	-1.8794	→	.684	5	2	→	57	4	mill slot P11-X axis
27	→	.1206	→	-.684	→	2.0	→	.0			→			null X axis-P12
28	→	-.6206	→	-1.816	→				→	0	→	2	4	return to sp.

Example No. 7: Plate, PARTNO 050117

Purpose: To demonstrate typical 2-axis program-
ming involving the use of trigonometric
solutions.

PROGRAM NOTES:

1. Two drawings of the part are shown (Fig-
   ures 4-13 and 4-14). The first shows the
   part to be milled. The second shows the
   path the tool will follow. Since the tool
   diameter is .5 inch, the center path of the
   cutter is located .25 inch outside the de-
   sired part form. The second drawing also
   shows the points that must be determined
   to define the centers of the arcs and end
   points of the angular moves.
   The trigonometric solutions for three typi-
   cal points are shown:

2. SEQNO 1 moves the tool from the start
   point in rapid traverse to the point near
   where milling will begin. The second data
   block of SEQNO 1 lowers the tool (M57)
   and mills at 4 ipm (f4) along the X axis
   to the beginning of the first arc.

3. SEQNO 2 - Contains the X,Y,I, and J in-
   formation needed to perform the first
   quadrant of the arc. SEQNO 3 contains
   the information needed to complete the
   arc.

4. SEQNO 4 - Mills the 60 degree angle using
   linear interpolation.

5. SEQNO 5 through 10 complete the remain-
   ing sides and arcs.

6. SEQNO 11 - Raises the tool, returns it in
   rapid traverse to the start point and re-
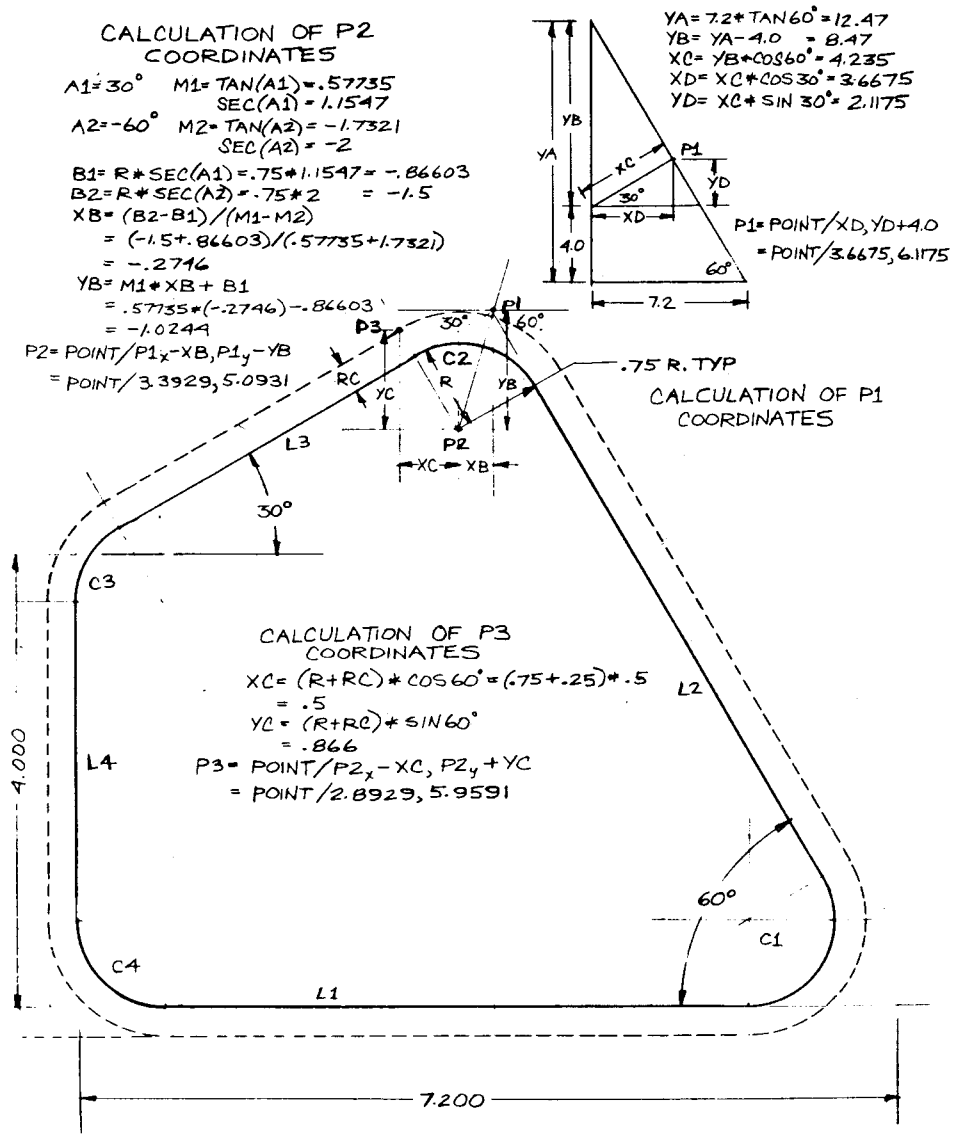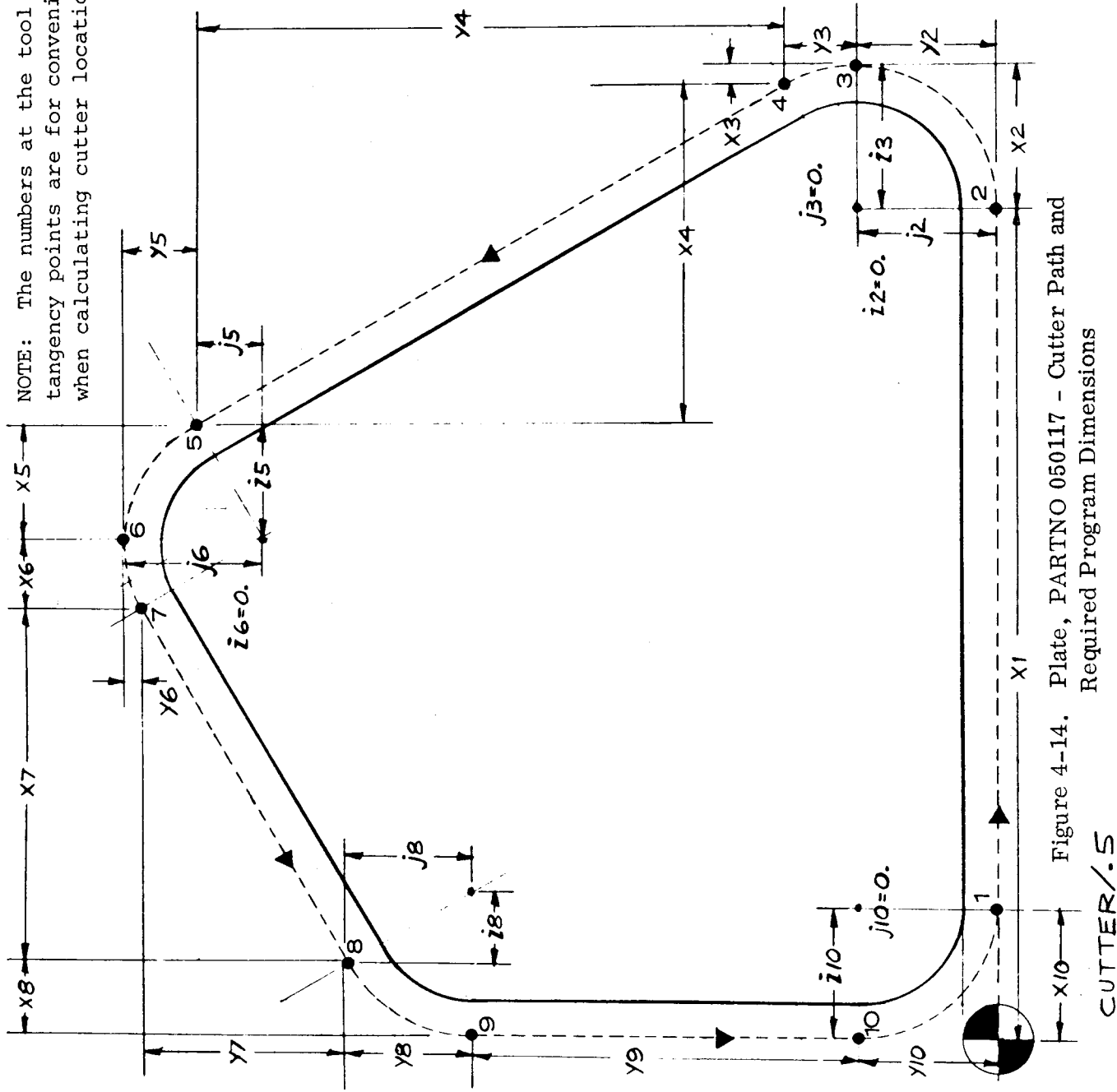   winds the tape.

Figure 4-13. Plate, PARTNO 050117

Figure 4-14. Plate, PARTNO 050117 - Cutter Path and Required Program Dimensions

NOTE: The numbers at the tool path tangency points are for convenience when calculating cutter locations.

CUTTER/.5

EXAMPLE #7

CONTINUOUS PATH CONTOURING CONTROL PROGRAM

PREPARED BY M Keith   DATE 7/15/11
CHECKED BY _____   DATE
SHEET __1__ OF __1__
TAPE NO. ____   DEPT. ____

PART NO. 050117
PART NAME PLATE
JOB NO. 713
COMPANY NAME
ADDRESS

REMARKS: startpoint -1, -.25 from bottom left hand side. Use .5 dia cutter

N SEQ. NO.	T OR E	X	T OR E	Y	T OR E	Z/I	T OR E	J	T OR E	F FEED RATE	T OR E	M MISC FUN.	E	REMARKS
RNS	→											4	⌀	Sendbfn
1	→	.75	→							→		7	⌀	controls
	→	6.151	→							0 ⌀				
2	→	1.⌀	→	1.⌀	;	.⌀	→	-1.⌀	⌀	4	→	57 ⌀		X1
3	→	-.134	→	.5	⌀	1.⌀	→	.⌀	⌀					X2, Y2
4	→	-2.5075	→	4.3432	⌀		→							X3, Y3
5	→	-.8661	→	.5	→	.866	→	.5	⌀					X4, Y4
6	→	-.5	→	-.134	→	.⌀	→	1.⌀	⌀					X5, Y5
7	→	-2.6434	→	-1.5262	⌀		→		⌀					X6, Y6
8	→	-.5	→	-.866	⌀	-.5	→	.866						X7, Y7
9	→	.⌀	→	-2.817	⌀		→							X8, Y8
10	→	1.⌀	→	-1.⌀	→	-1.⌀	→	.⌀	⌀					Y9
11	→	-1.75	→								0	→	2 ⌀		X10, Y10 Gott startpont