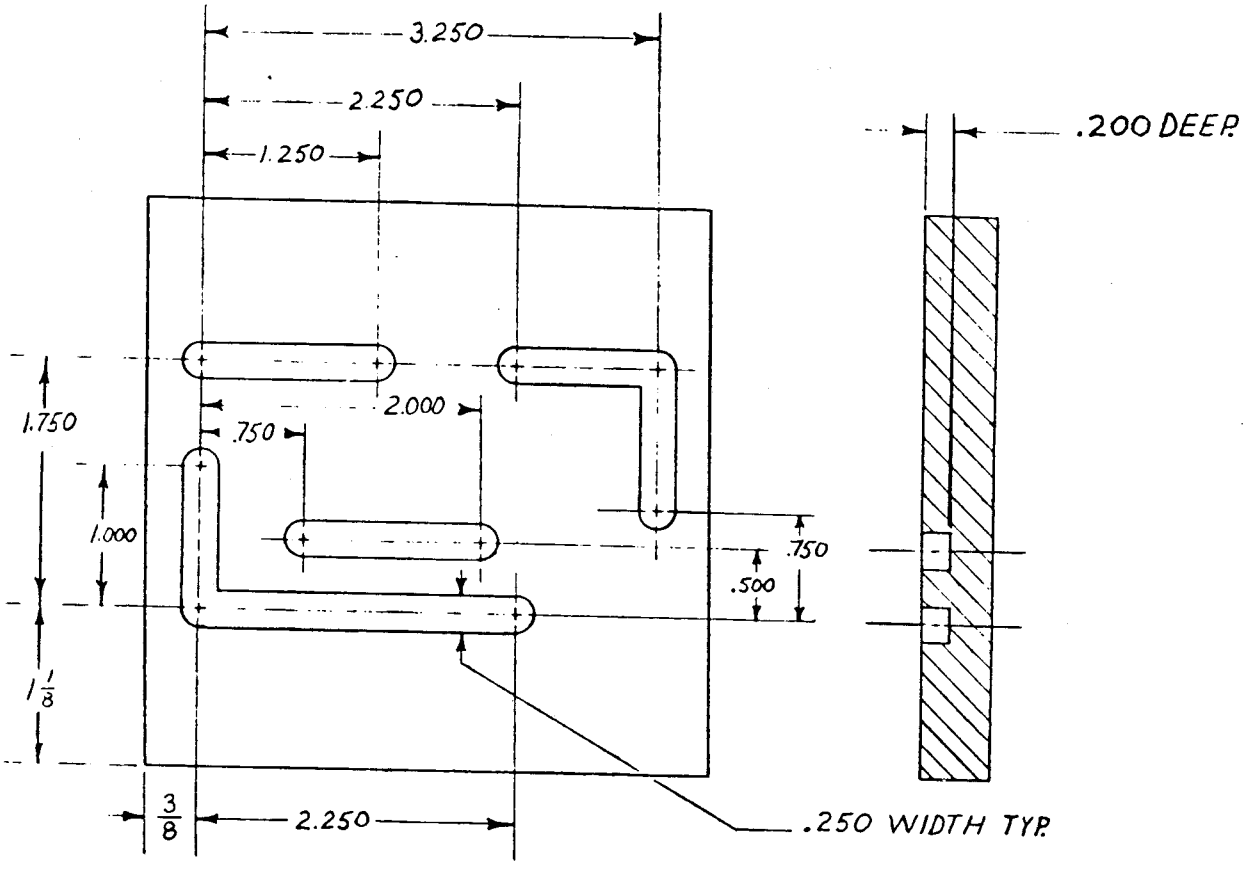
PROBLEM SET

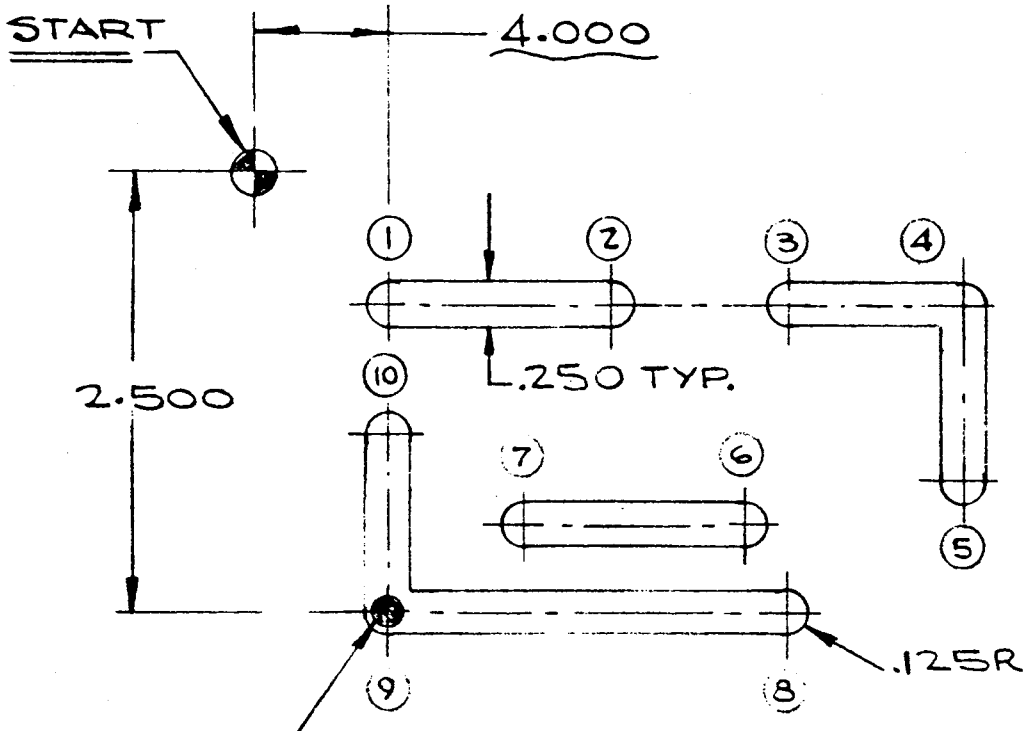
Bridgeport® TEXTRON
Bridgeport Machines Division of Textron Inc.

Program 1


3.250
2.250
1.250
.200 DEEP
1.750
2.000
.750
1.000
.750
.500
1 1/8
3/8
2.250
.250 WIDTH TYP.

Program 1

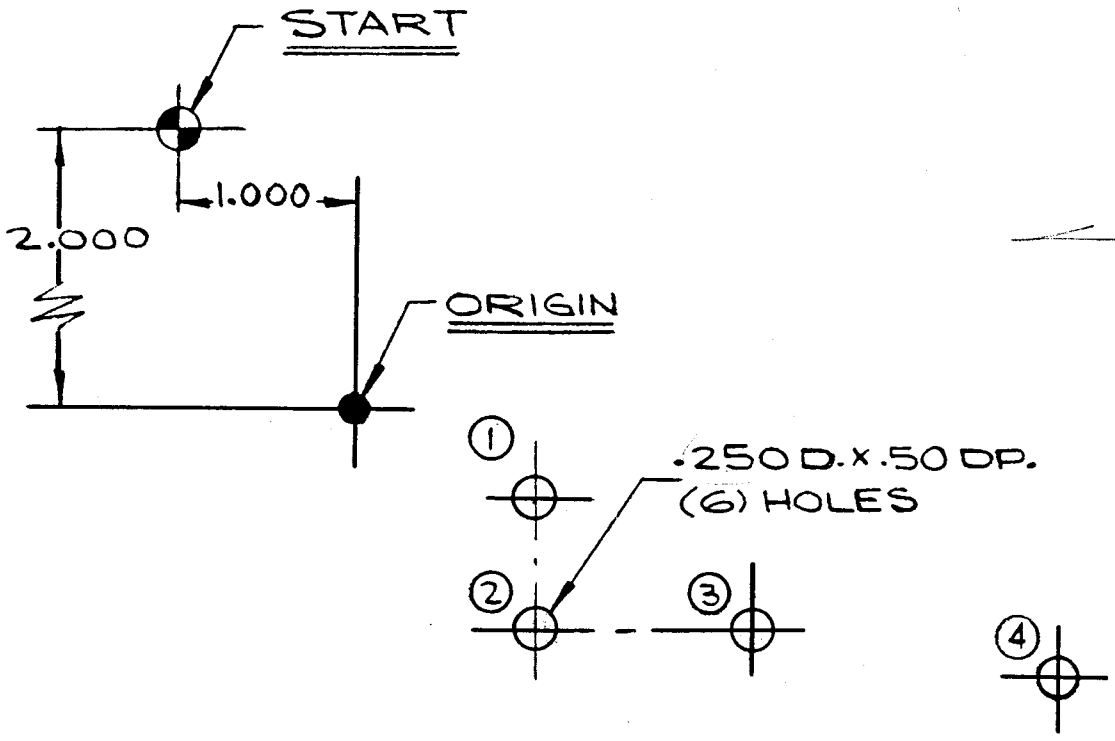
ALL SLOTS TO BE .200 DP.
CLEARANCE TO BE .050 ABOVE TOP OF PART
FEED RATE TO BE 15 I.P.M.



Loc.	X CO'ORD.	Y CO'ORD
START	- 4.000	2.500
1.	.000	1.750
2.	1.250	1.750
3.	2.250	1.750
4.	3.250	1.750
5.	3.250	.750
6.	2.000	.500
7.	.750	.500
8.	2.250	.000
9.	.000	.000
10.	.000	1.000

Program 2

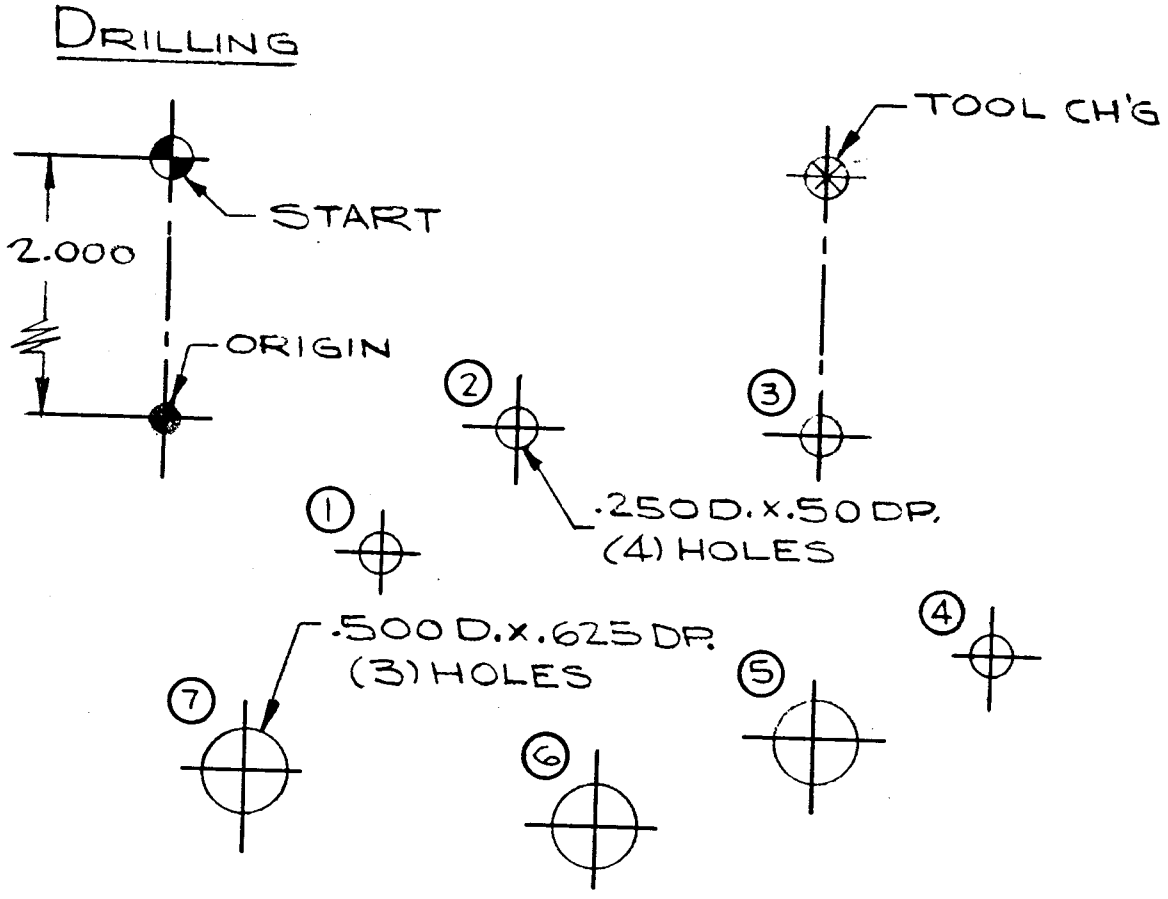
DRILLING



LOC.	X COORD.	Y COORD.
START	-1.000	2.000
1.	1.000	- .500
2.	1.000	-1.250
3.	2.250	-1.250
4.	4.000	-1.500
5.	2.750	-2.500
6.	.500	-2.250

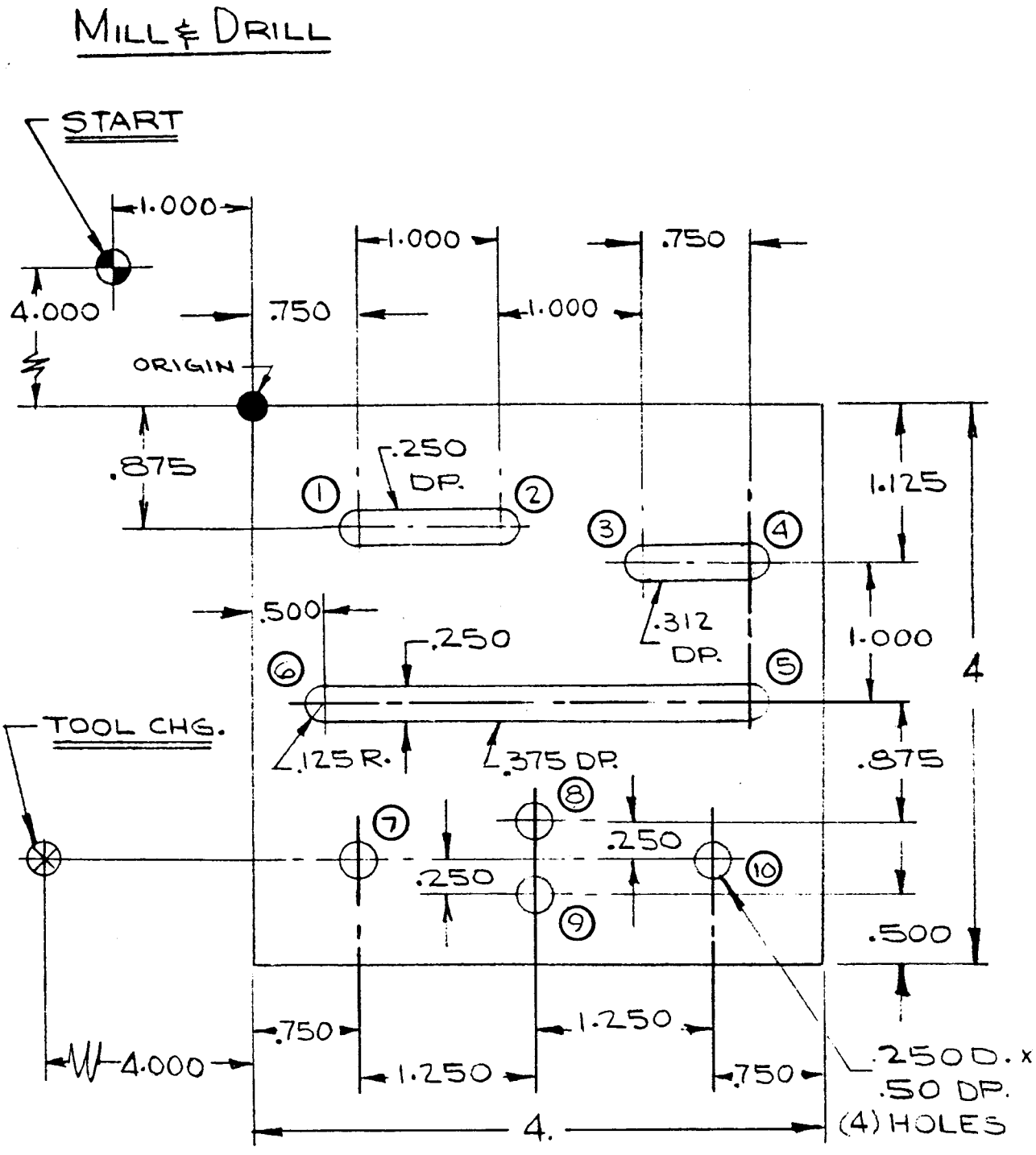
FEED RATE TO BE 8 IPM
CLEARANCE TO BE .05 ABOVE TOP OF PART

Program 3

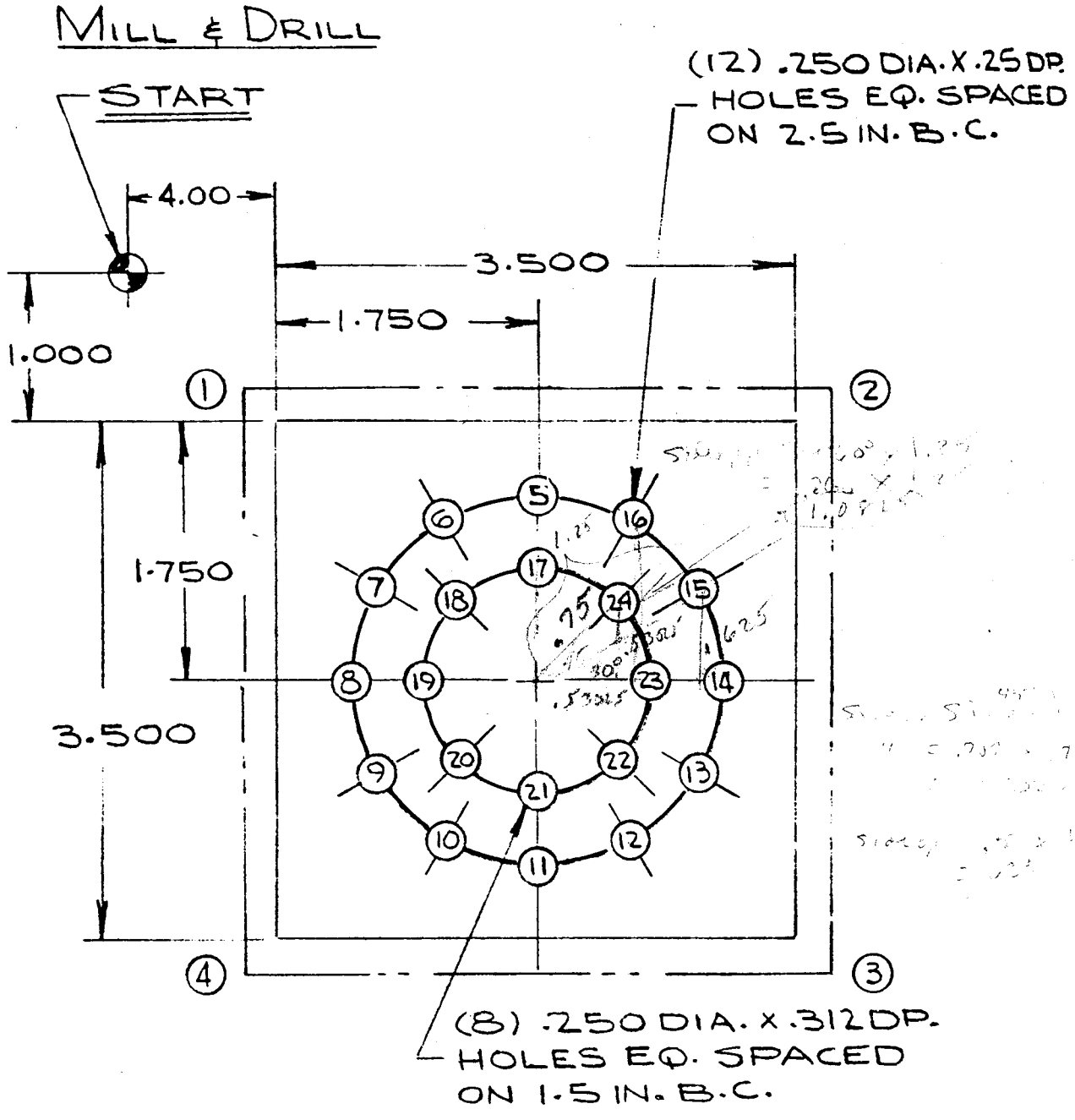


LOC	X CO'ORD	Y CO'ORD
START	.000	2.000
1.	1.250	- .750
2.	2.000	.000
3.	3.000	.000
4.	3.500	- 1.250
TOOL CHG.	3.000	2.000
5.	3.000	- 1.750
6.	2.500	- 2.250
7.	.500	- 2.000

Program 4

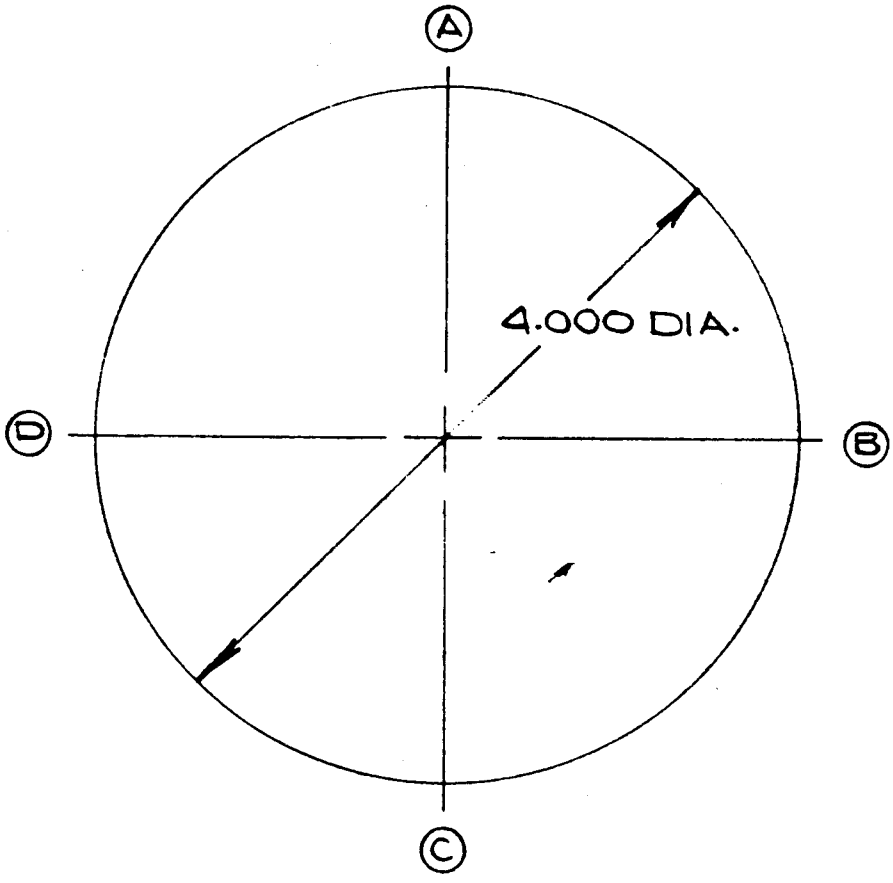

MILL & DRILL
START
1.000
4.000
.750
1.000
1.000
.750
ORIGIN
.875
.250
DP.
1
2
3
4
1.125
.500
.250
.312
DP.
1.000
4
6
5
TOOL CHG.
.125 R.
.375 DP.
.875
8
7
.250
.250
10
9
.500
.750
1.250
.250 D. x
.50 DP.
(4) HOLES
4.000
1.250
.750
4.

Program 5



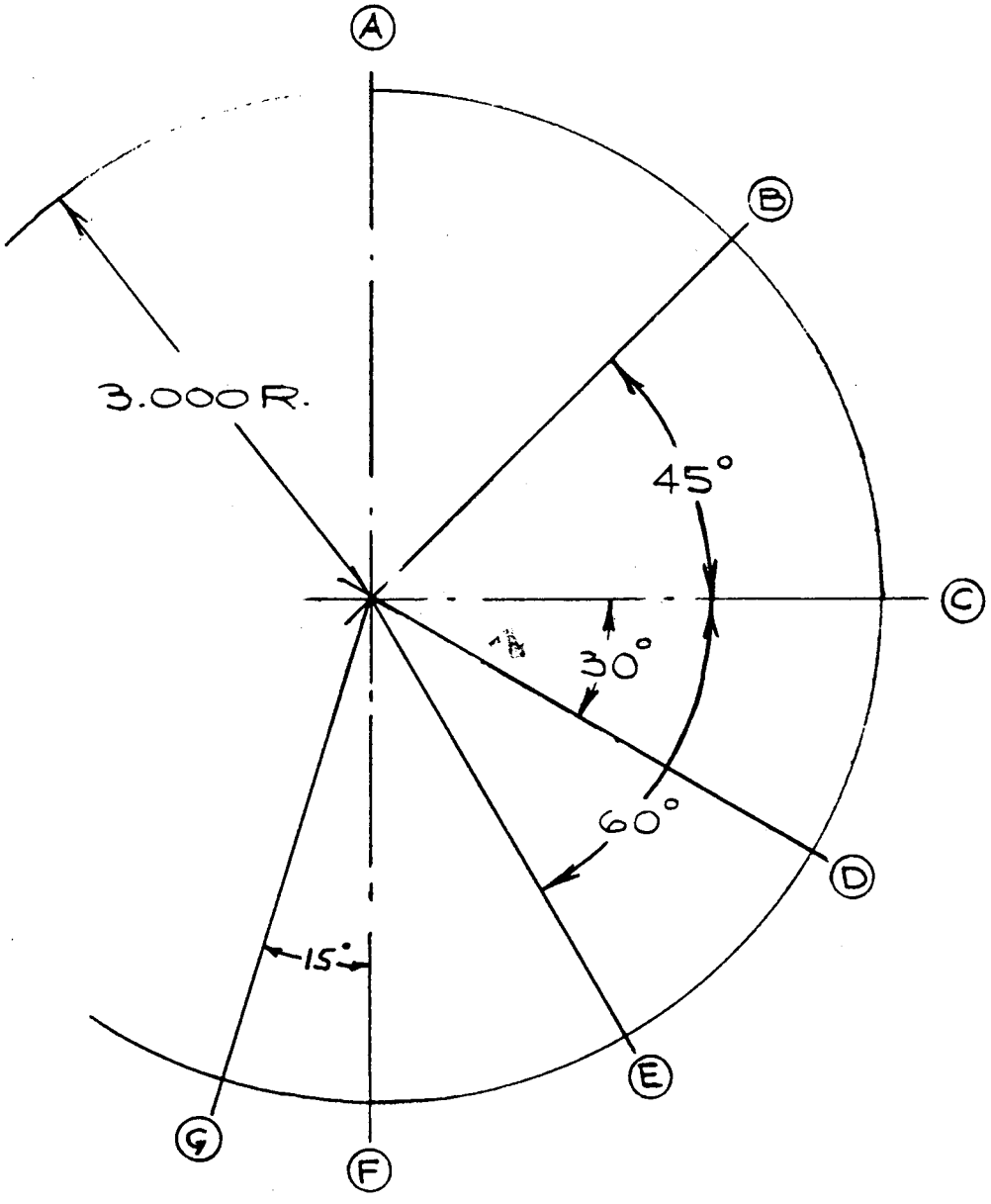
	SINE	COSINE	TAN	COTAN.	SECANT	COSEC.
30°	0.500	0.866	0.577	1.732	1.155	2.000
60°	0.866	0.500	1.732	0.577	2.000	1.155
45°	0.707	0.707	1.000	1.000	1.414	1.414

Program 6



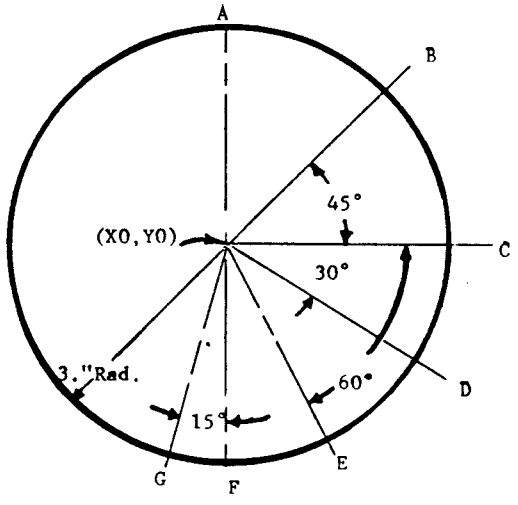
NOTE: PROGRAM CUTTER ON THE LINE...

Program 7



Program 7

Multi-Quadrant Circular



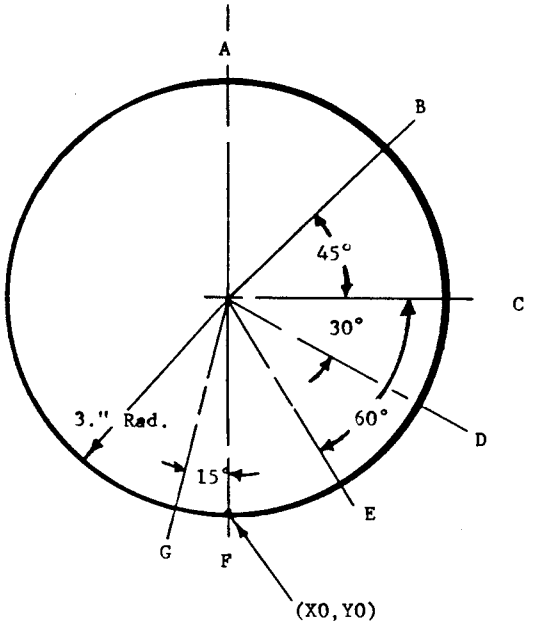
1. Cutter on line go from A to D (XO,YO) center of Arc.
2. .5" Dia. end mill cut outside of line B to G (XO,YO) center of Arc.

Speed & Feed for above:

1800 RPM
Plunge 7.5 IPM, Mill 15 IPM

Tool Change Position: X-3, YO

Depth of Cut: .100



3. .5" Dia. end mill cut inside of line G to D (XO,YO) on line at "F".
4. 1" Dia. end mill cut inside of line E to A (XO,YO) on line at "F".

Speed & Feed for above:

1450 RPM
Plunge 6.0 IPM, Mill 12. IPM

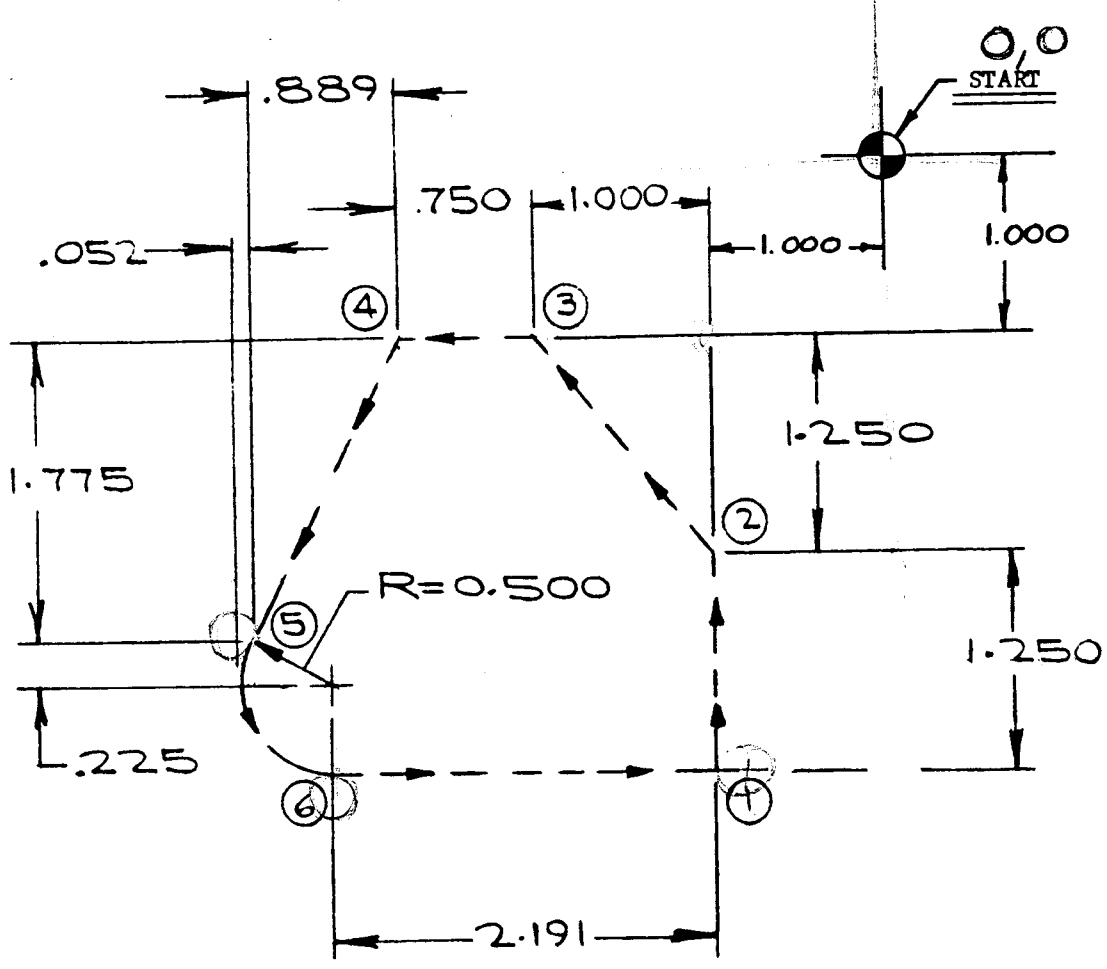
Tool Change Position: X-3, YO

Depth of Cut: .100

Program 8

INCREMENTAL PROGRAM

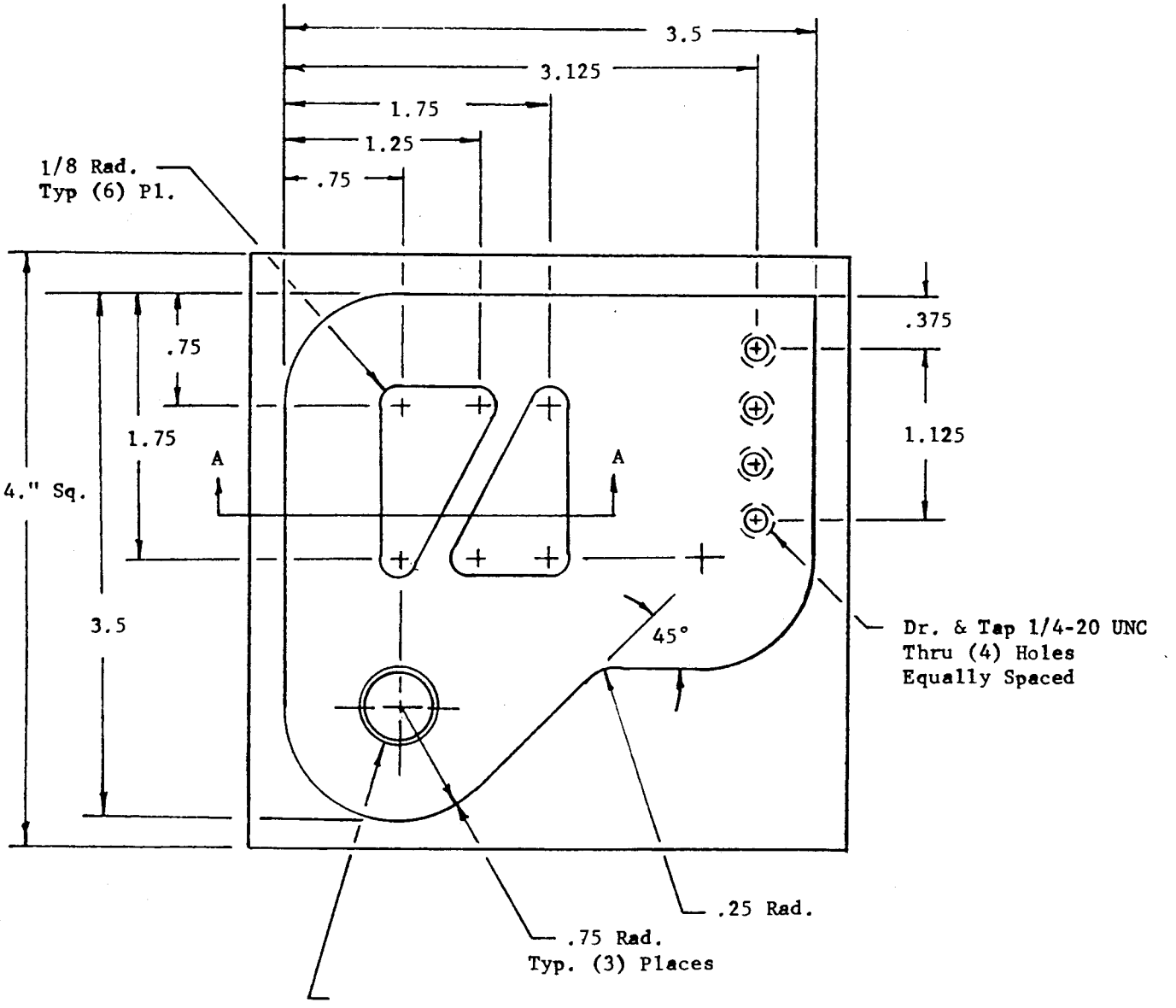
Note: Use G75 Circular

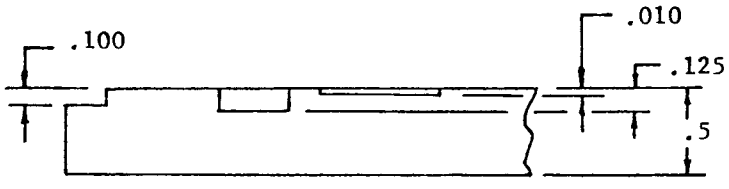


CUTTER PATH

	X	Y	I	J
1	-1	-3.5		
2	-1	-2.250		
3	-2	-1		
4	-2.75	-1		
5	-3.191	-2.75		
6	-3.191	3.5		

Program 9





Section "A-A"

Program 9

Instructions:

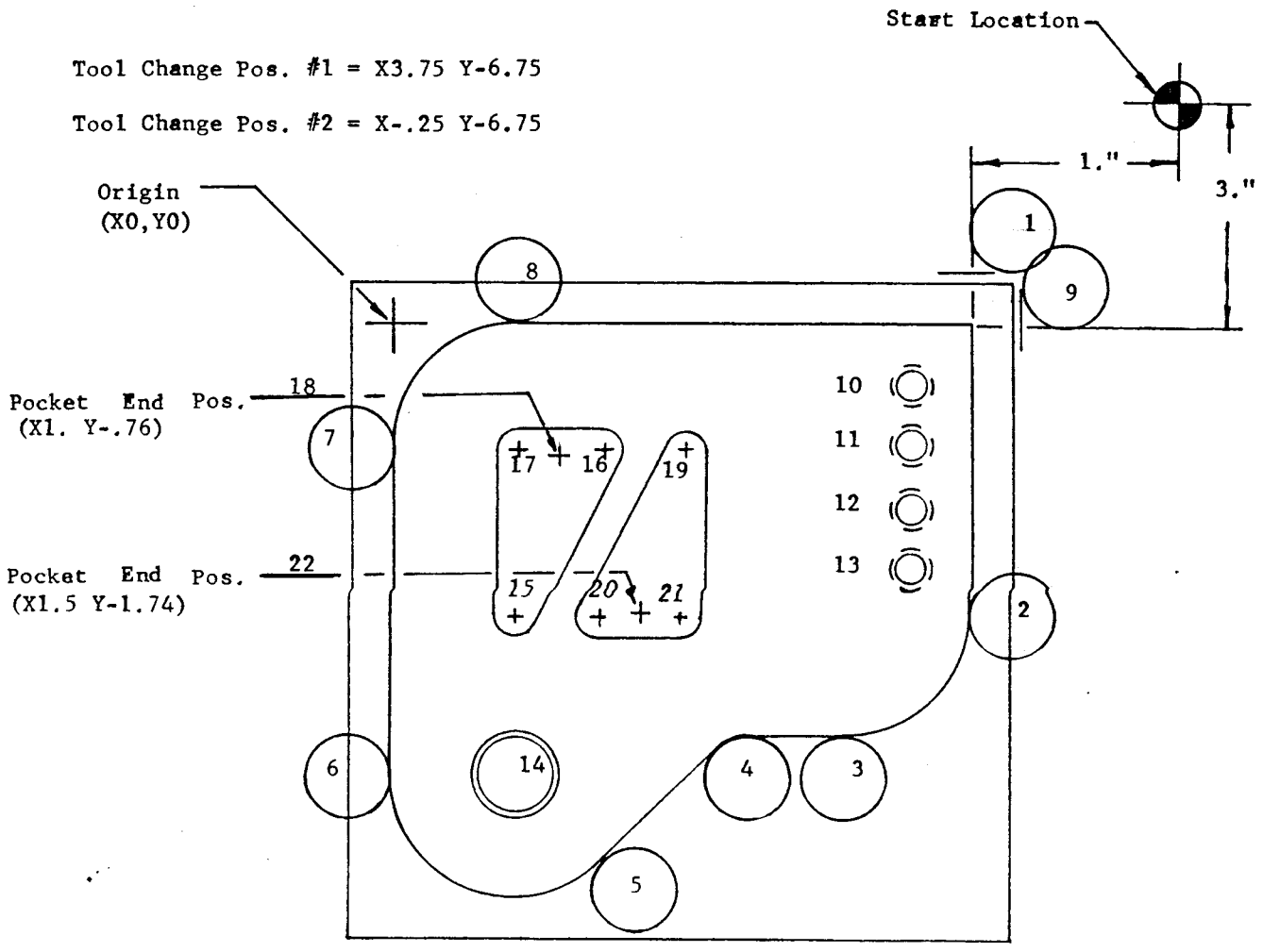
Material 1/2 X 4." Sq. 2024 Aluminum
Top Surface of Part = "Z" Zero
Chamfer all Holes .025/side

Tooling:

Tool#	Description	RPM	Feedrate	Tool Chg. Loc.
1	.5 Dia End Mill	1270	12. IPM	Start Loc.
2	.5 Dia 90° Spot Drill	1500	8.	T.C.#1 Pos.
3	.437 Dia. Drill	1450	8.	T.C.#2
4	.4505 Dia. Reamer	500	5.	T.C.#2
5	.25 Dia. End Mill	3120	12.	T.C.#2
6	.209 Dia. Drill	3300	8.	T.C.#1
7	1/4-20 Unc Tap	200	9.9	T.C.#1

Tool Change Pos. #1 = X3.75 Y-6.75

Tool Change Pos. #2 = X-.25 Y-6.75



Program 10

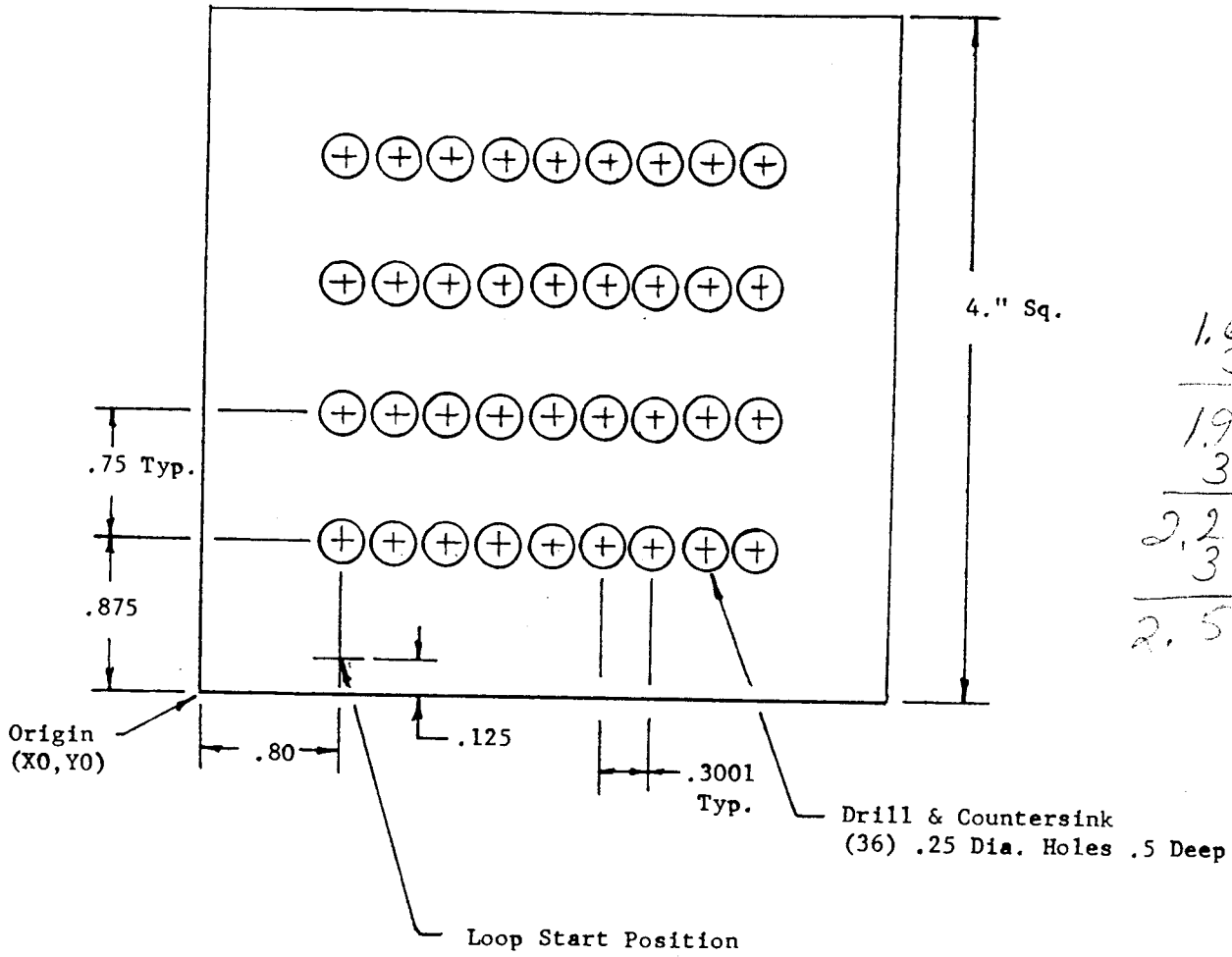
LOOPING

Tool #1 .5 Dia 90° Spot Drill 1500 RPM 8. IPM

Tool #2 .25 Dia Drill 3300 RPM 9.IPM

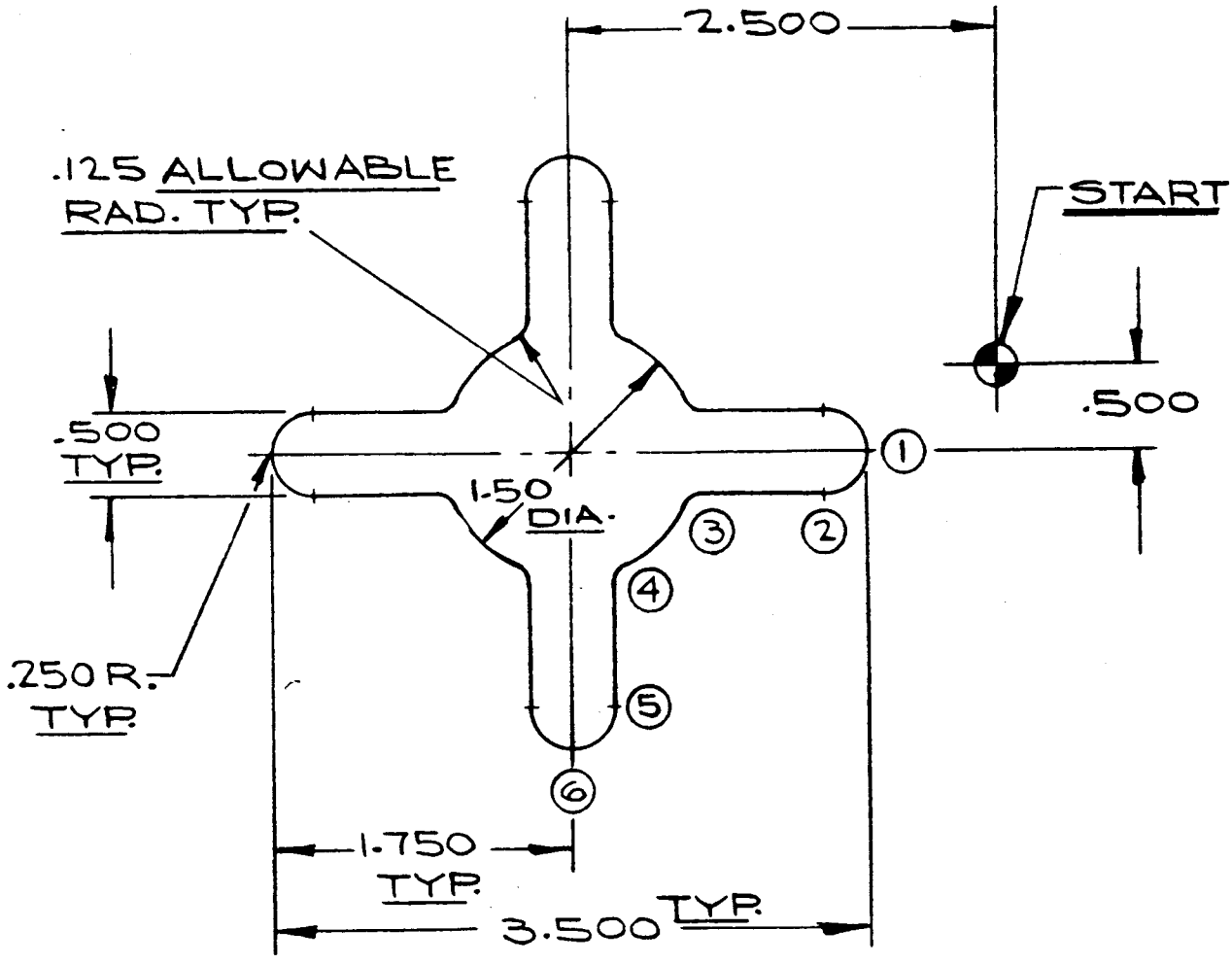
Note: Top of part = "Z" Zero

Tool Change Position X-4. Y3.



1.625
3001
1.9251
3001
2.2252
3001
2.5253

Program 11



Programming Instructions:

"Z" Zero = Top of the Part
Use G75 Circular

Tooling- .25Dia. End Mill 3300 RPM

Plunge Feed 5. IPM -- .1 Deep ; Contour Mill 10. IPM