MANUFACTURING PROCESSES (TA-202)

NC PART PROGRAMMING

Dr. V. K. Jain

Mechanical Engineering Department

Indian Institute of Technology

Kanpur (India)

NC PART PROGRAMMING

CLASSIFICATION OF DATA REQUIRED TO PRODUCE A PART ?

INFORMATION FROM THE DRAWING

- DIMENSIONS OF FEATURES
- SEGMENT SHAPE

MISCELLANEOUS INFORMATION

- SURFACE QUALITY
- TOLERANCES
- TOOL & W/P MATL.
- MACHINING CONDITIONS
- AUXILIARY FUNCTION

DATA DETERMINED BY THE PART PROGRAMMER

- DIRECTION OF THE CUTTING
- TOOL CHANGE
- SEQUENCING (REQUIRES FAMILIARITIES WITH NC PROCESS)
- FAMILIARITY WITH NC M/C TOOL SYSTEMS

HOW THE TAPES CAN BE PRODUCED ?

MANUAL

COMPUTER ASSISTED PROGRAMMING

VARIOUS FUNCTIONS

- **SEQUENCE NUMBER N:**
  - DISPLAYED IN THE CONSOLE. EACH BLOCK.
- **PREPARATORY FUNCTION G:**
  - PREPARES **MCU** TO BE READY TO PERFORM SPECIFIC MODE OF OPERATION.
  - PRECEDS THE DIMENSION WORD / NUMBER.

    EX. G21 – METRIC DATA INPUT.
- **DIMENSION WORD:**
  - DISTANCE DIMENSION WORDS X, Y, Z.
  - CIRCULAR DIMENSION – IN CIRCULAR INTERPOLATION & THREAD CUTTING.
  - I, J, K – DISTANCE TO ARC CENTER (OR THREAD LEAD) PARALLEL TO X, Y, Z.

VARIOUS FUNCTIONS

- **ANGULAR DIMENSION WORD : A, B, C AROUND X, Y, Z, RESPECTIVELY**
- **FOR ANGULAR DIMESNION AROUND SPECIAL AXIS D, E.**
- **DIMESNION WORDS – CO-ORDINATES (INCREMENTAL / ABSOLUTE)**
- **THE MISCELLANEOUS FUNCTION M:**
  - 2 DIGITS AUXILIARY INFORMATION NOT RELATED TO DIMENSIONS – SPINDLE COMMAND, COOLANT ON/OFF ETC.
  - 'STOP' (M00,M01) AND 'END' ARE EXECUTED AFTER COMPLETION OF OTHER COMMANDS IN THE BLOCK.

VARIOUS FUNCTIONS

- CIRCULAR INTERPOLATION:
  - **PREPARATORY FUNCTIONS (G17,G18,G19) FOR PLANE OF ARC**
  - **FOR DIRECTION OF TOOL ON THE ARC (G02,G03….)**
  - 4 DIMENSION WORDS/BLOCK – 2 TO THE END OF THE ARC + 2 FOR THE DISTANCE OF THE ARC CENTER.
  - AN ARC MUST END IN THE SAME QUADRANT.
  - IF MORE THAN 1 QUANDRANT – 2 OR MORE BLOCKS OF NFORMATION.
- **THREAD CUTTING**
  - THREAD CUTTING MODE BY PREPARATORY FUNCTIONS G33-G35
  - FOUR DIMENSION WORDS/BLOCK.
  - LEADS PARALLELED TO X, Y, Z AXES – BY i, j, k
  - NO ALGEBRAIC SIGN (+ OR -).

MANUAL PART PROGRAMMING

- **TYPES OF MANUAL PROGRAMMING: POINT TO POINT, CONTOURING, 3-D.**
- *(3-D programming, only with the help of a computer) .*

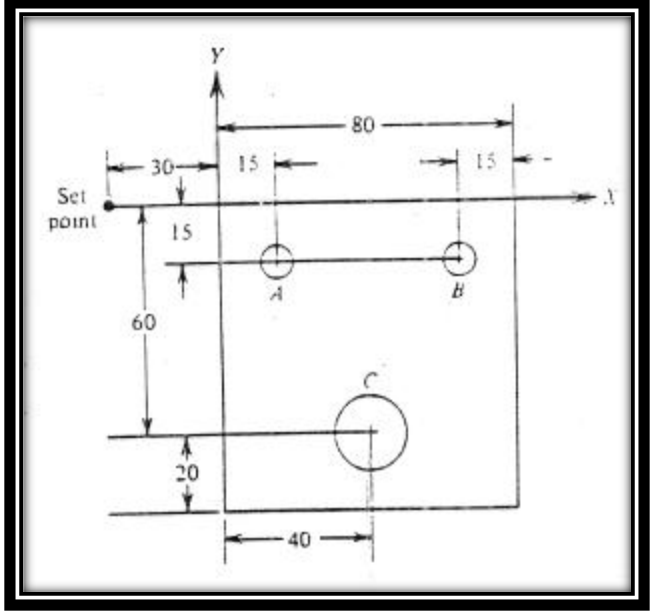
- WRITE THE PART PROGRAM IN A STANDARD FORMAT
- SPECIAL MANUSCRIPT AND FLEXO WRITER – TAPE, LISTING etc. *(Nowadays, these are is not required)*
- MOSTLY POINT TO POINT programming – SIMPLE
- COMPLICATED CONFIGURATION OF THE PATH – CALCULATIONS with the help of a COMPUTER.
- SEVERAL SPECIAL PURPOSE LANGUAGES FOR NC ROGRAMMING – SAY, APT.

BASIC CONCEPTS OF MANUAL PART PROGRAMMING

- WHO PREPARES THE TAPE / CD ? – PROGRAMMER –> SHOULD BE FAMILIAR WITH THE MANUFACTURING PROCESSES.
- **OPTIMAL SEQUENCE OF OPERATIONS SHOULD BE KNOWN.**
- **PROGRAM SHOULD BE WRITTEN IN THE MANUSCRIPT.**
- **EACH LINE OF THE MANUSCRIPT TELLS WHAT IS BEING DONE ?**
- **TRANSFER OF CUTTING TOOL INCLUDING OTHER INSTRUCTIONS.**
  - **EXAMPLE :**N – SEQUENCE #, G – PREPARATORY FUNCTION, X&Y – DIMENSIONAL WORDS,F (or f)- FEED RATE, S – SPINDLE SPEED,T – TOOL #, M – MISCELLANEOUS FUNCTION, EB – END OF BLOCK.
- **X, Y – WORD ADDRESS.**
- **EB – READING COMPLETED & MOTION STARTS**

FEED FUNCTION

- **‘f’ – RESTRICTED TO CONTOURING OR STRAIGHT - CUT.**
- **FEED RATES OF LINEAR OR CIRCULAR MOTION – INDEPENDENT OF SPINDLE SPEEDS – EXPRESSED AS inches/min OR mm/min.**
- **‘MCU’ ACCCEPTS SPECIFIC METHOD OF EXPRESSING ‘FRN’ :**

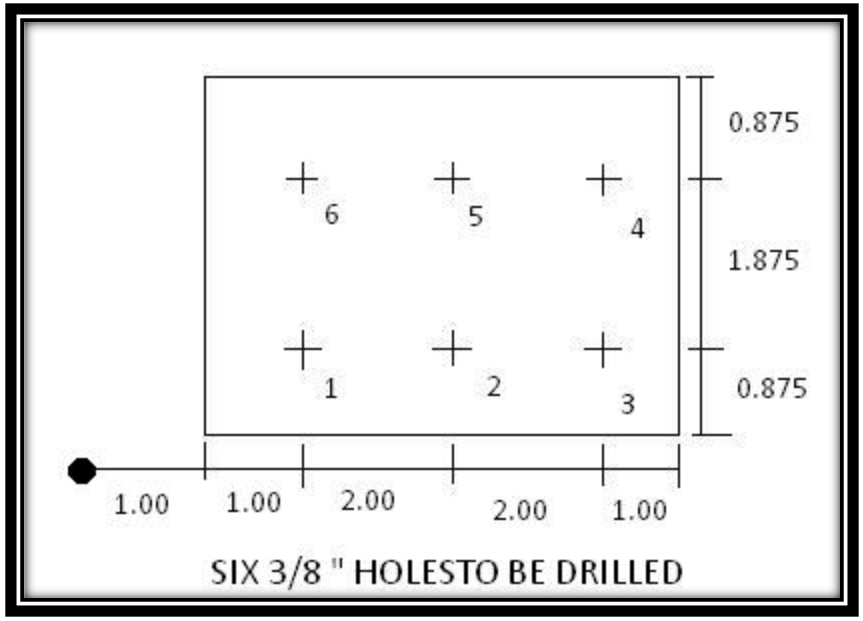


***ATTEMPT YOUR SELF.**
***FOUR HOLES ARE TO BE DRILLED.**
TWO OF 5mm DIA. AND ONE OF 10mm DIA.
*** WRITE THE FULL PART PROGRAM**

- **M03 – SPINDLE CW**
  SPINDLE START ROTATION CW
- **M06 – TOOL CHANGE**
  MANUAL / AUTOMATIC (TOOL SELECTION IGNORED)
- **M30 – END OF THE TAPE**
  (INCLUDES REWINDING OF TAPE READY FOR NEXT W/P)

EXAMPLE NC PART PROGRAMMING

R WORK PLANE (OR GAGE HEIGHT)
IT IS GENERALLY 0.100 INCH ABOVE THE SURFACE OF THE W/P. IT IS USED AS A REFERENCE, AND ALL OTHER WORK SURFACES ARE RELATIVE TO THIS LOCATION.



FIXED OR CANNED CYCLES
(G1–G89)

THEY ARE PRESET COMBINATION OF OPERATIONS, SUCH AS DRILLING, WHERE ALL M/C – AXES MOTIONS ARE PROGRAMMED AND WILL REPEAT THEMSELVES UNTIL CANCELLED BY A G80 CODE.

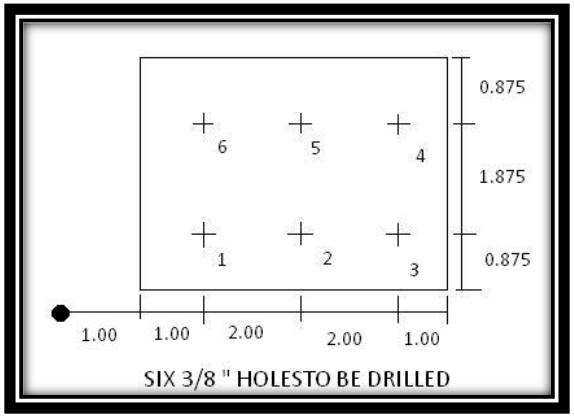
N040 G81 X2.000 Y1.500 R0.100 Z-1.000 f5

EXAMPLE NC PART PROGRAMMING

- **G81 –** A FIXED DRILLING CYCLE
- **R 0.1000 -** THE GAGE HEIGHT IS SET AT 0.100 ABOVE THE WORK SURFACE
- **Z -1.000 -** THE DRILL WILL BE FED INTO THE WORK 1.000 INCH DEEP
- **f 5 -** THE FEED RATE FOR THE DRILL WILL BE 5 inch / min.
- **AFTER REACHING THE 'Z' DEPTH THE DRILL WILL AUTOMATICALLY RETRACT IN THE *RAPID MODE* TO THE GAGE HEIGHT.**

TO DRILL ALL THE SIX HOLES

- **N010G91** - INCREMENTAL MODE
- **N020G70** - INCH MODE

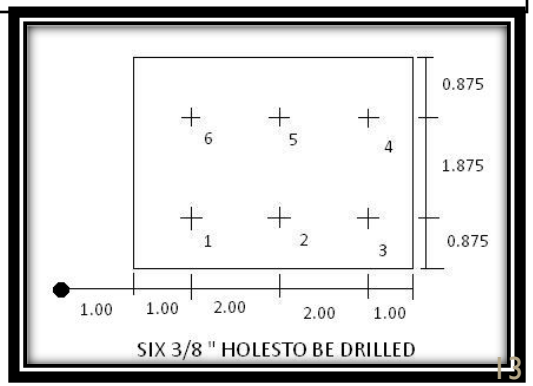


EXAMPLE NC PART PROGRAMMING

- **N030G81X2.000Y0.875R0.100Z-1.000f5M03**

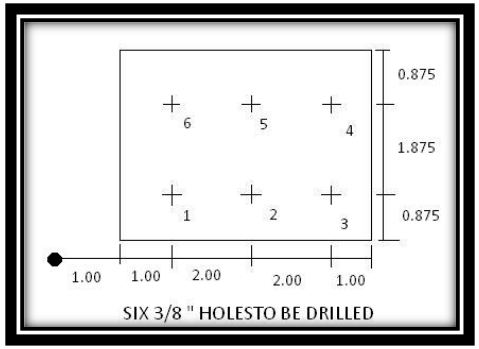
EXPLANATION (SOME STEPS ARE LEFT OUT)

- **G81** — **FIXED CYCLE**
- **X2.000, Y0.875** — **THE M/C TABLE WILL RAPID TO HOLE #1 POSITION**
- **R0.100** — **THE M/C SPINDLE WILL RAPID DOWN SO THAT THE DRILL POINT IS 0.100 INCH ABOVE THE SURFACE OF THE PART.**
- **M03** — **START SPINDLE ROTATION CLOCKWISE DIRN.**
- **Z-1.000, f5** — **THE DRILL WILL ADVANCE 1.000 INCH INTO THE W/P AT A FEED RATE OF 5IN./MIN. THE DRILL WILL RAPID OUT OF THE HOLE BACK TO GAGE HEIGHT (0.100 INCH ABOVE WORK).**



EXAMPLE: NC PART PROGRAMMING

- **N040X2.000** — THE TABLE WILL RAPID 2.000INCH TO HOLE #2 G81 WILL BE REPEATED & A HOLE WILL BE DRILLED.
- **N050X2.000** — HOLE # 3
- **N060Y1.875** — HOLE # 4
- **N070X-2.000** — HOLE # 5
- **N080-2.000** — HOLE # 6



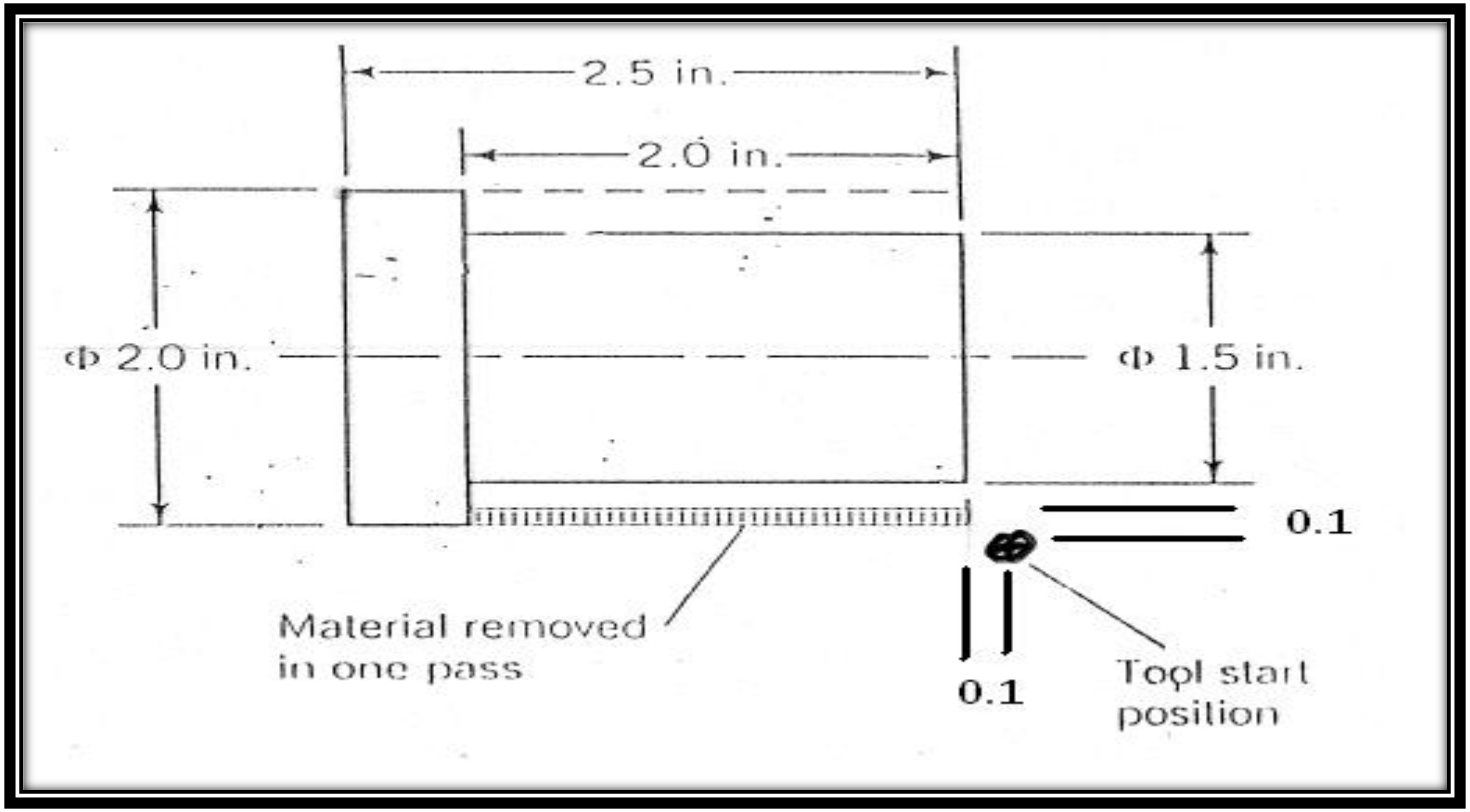
- **N090G80** — **CANCELS THE DRILL CYCLE AND AUTOMATICALLY PUTS THE M/C IN THE RAPID MODE.**
- **N100X-2.000Y-2.275M06** — **THE TABLE RAPIDS SIMULATNEOUSLY ALONG THE XY AXES AND RETURNS TO THE XY ZERO. M06 STOPS THE M/C SPINDLE AND RAISES THE CUTTING TOOL TO THE FULL RETRACT POSITION.**
- **N110M30** — **REWINDS THE TAPE IN PREPARATION FOR USE IN DRLLING THE NEXT PART**

SELF READING EXERCISES

Write an NC program to machine the simple aluminum pin shown in Figure 6.7. A 2-in.-diameter blank, $2\frac{1}{2}$-in. long, is to be used.

Assumptions

1. The center of the left face of the pin will be used for program zero.
2. The tool start position is 0.2 in. off the diameter and 0.1 in. off the right face.
3. Two roughing cuts (0.1 in. deep) and one finish cut (0.05 in. deep) will be taken.
4. A spindle speed of 1200 rpm and feed rate of 12 in. /min are used for machining.
5. Machine specification: N3G2X± 43Y± 43Z± 43R± 43F40S4T2M2.
6. X values are to be programmed as diameters.

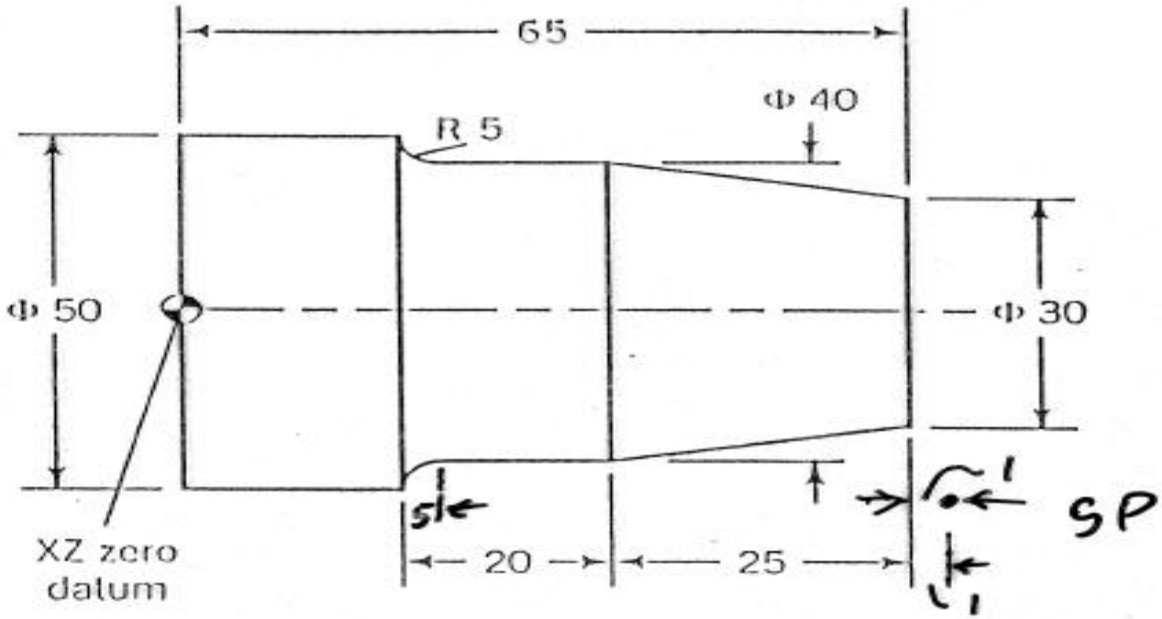


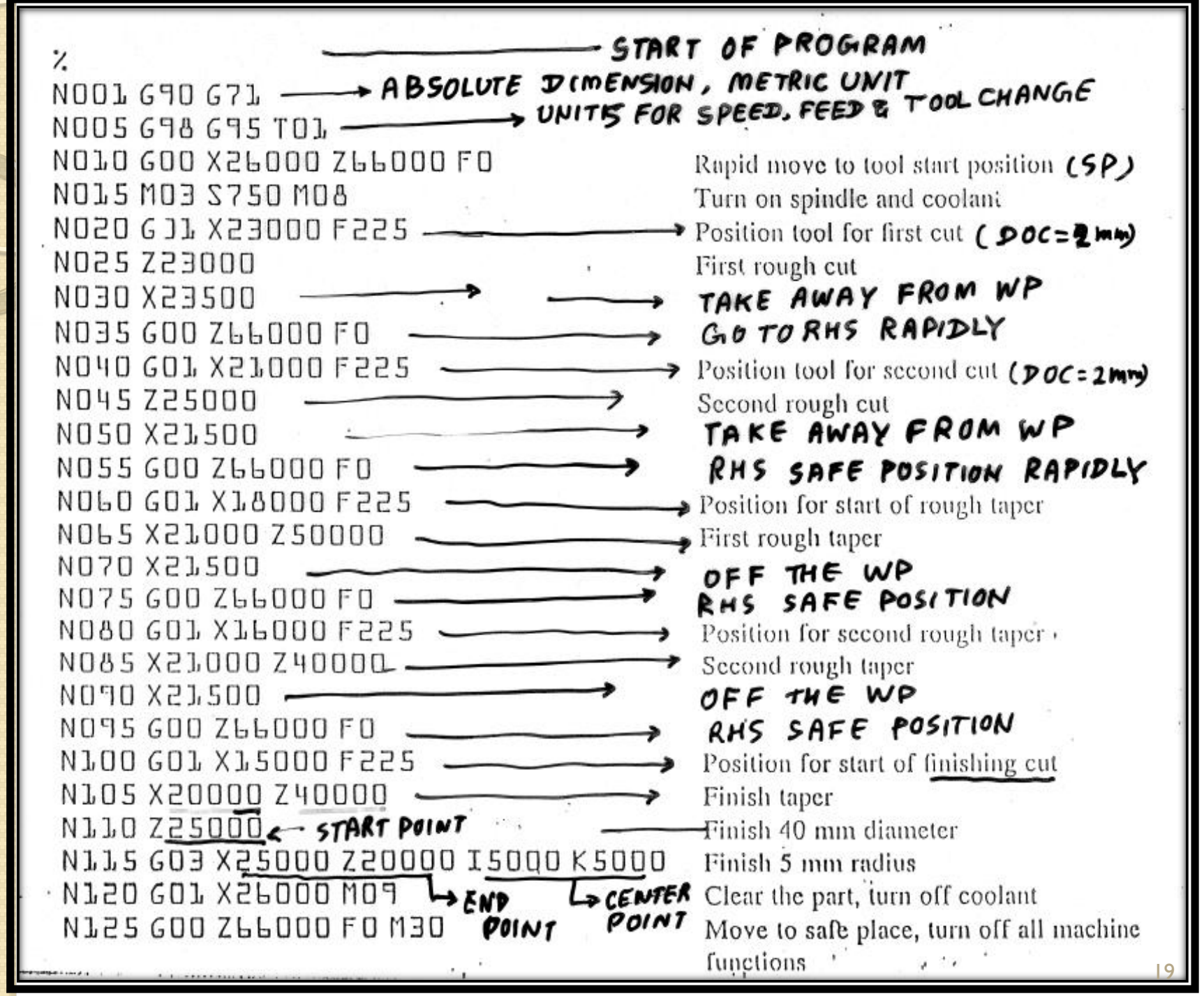
%	Indicates start of program
N005 G90 G70	Specifies absolute programming, inch units
N010 G98 G92 T01	Specifies units for speed and feed rate, loads 1st tool
N015 G00 X2200 Z2600 F0	Rapid positioning of tool to tool start position
N020 X1800 M03 S1200 F0	Position tool to remove 0.1 in. off part diameter, start spindle
N025 G01 Z500 F12	Feed tool into workpiece
N030 X1900	Retract tool (overlap previous cut)
N035 G00 Z2600 F0	Move tool clear of workpiece
N040 X1600 F0	Position tool to remove 0.1 in. off part diameter
N045 G01 Z500 F12	Feed tool into workpiece
N050 X1700	Retract tool (overlap previous cut)
N050 G00 Z2600 F0	Move tool clear of workpiece
N060 X1500 F0	Position tool to take finish cut
N065 G01 Z500 F12	Feed tool into workpiece
N070 X2200	Retract tool clear of the workpiece
N075 G00 X5000 Z5000 F0	Move to safe position
N080 M30	Turn off all machine functions

Write an NC program to machine the aluminum part shown in Figure 6.8. A 50-mm-diameter blank, 65-mm long, is to be used.

Assumptions

This is the process sequence used: face off to final length, rough cut 40-mm diameter in two passes, rough turn taper in two passes, finish machine to final dimensions. Absolute programming has been used; spindle speed is specified in rev/min and feed rate in mm/min. Note the use of F0 to specify rapid feed rate. X values are to be programmed as radii. The specification of the machine to be used is N3G2X±43Y±43Z±43R±43F4S4T2M2.

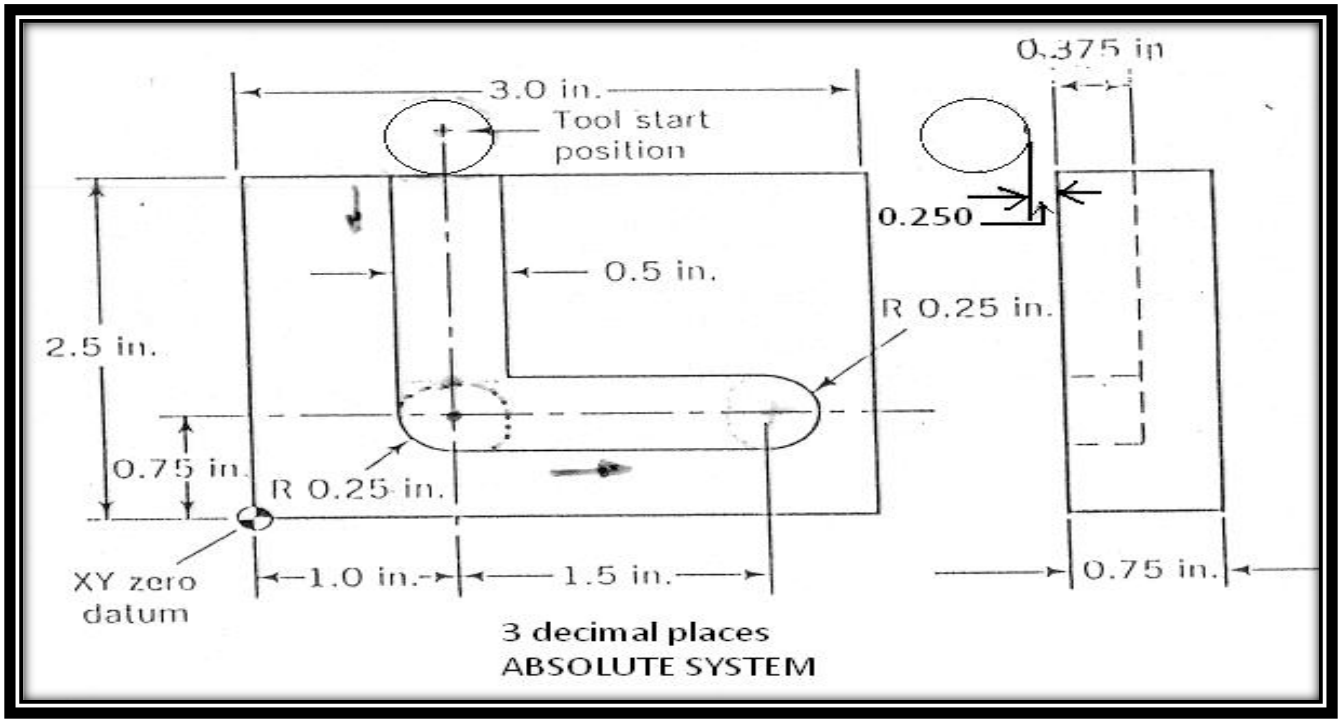



%
START OF PROGRAM
N001 G90 G71
ABSOLUTE DIMENSION, METRIC UNIT
N005 G98 G95 T01
UNITS FOR SPEED, FEED & TOOL CHANGE
N010 G00 X26000 Z66000 F0
Rapid move to tool start position (SP)
N015 M03 S750 M08
Turn on spindle and coolant
N020 G01 X23000 F225
Position tool for first cut (DOC=2mm)
N025 Z23000
First rough cut
N030 X23500
TAKE AWAY FROM WP
N035 G00 Z66000 F0
GO TO RHS RAPIDLY
N040 G01 X21000 F225
Position tool for second cut (DOC=2mm)
N045 Z25000
Second rough cut
N050 X21500
TAKE AWAY FROM WP
N055 G00 Z66000 F0
RHS SAFE POSITION RAPIDLY
N060 G01 X18000 F225
Position for start of rough taper
N065 X21000 Z50000
First rough taper
N070 X21500
OFF THE WP
N075 G00 Z66000 F0
RHS SAFE POSITION
N080 G01 X16000 F225
Position for second rough taper
N085 X21000 Z40000
Second rough taper
N090 X21500
OFF THE WP
N095 G00 Z66000 F0
RHS SAFE POSITION
N100 G01 X15000 F225
Position for start of finishing cut
N105 X20000 Z40000
Finish taper
N110 Z25000
START POINT
Finish 40 mm diameter
N115 G03 X25000 Z20000 I5000 K5000
Finish 5 mm radius
N120 G01 X26000 M09
END POINT
CENTER POINT
Clear the part, turn off coolant
N125 G00 Z66000 F0 M30
Move to safe place, turn off all machine functions

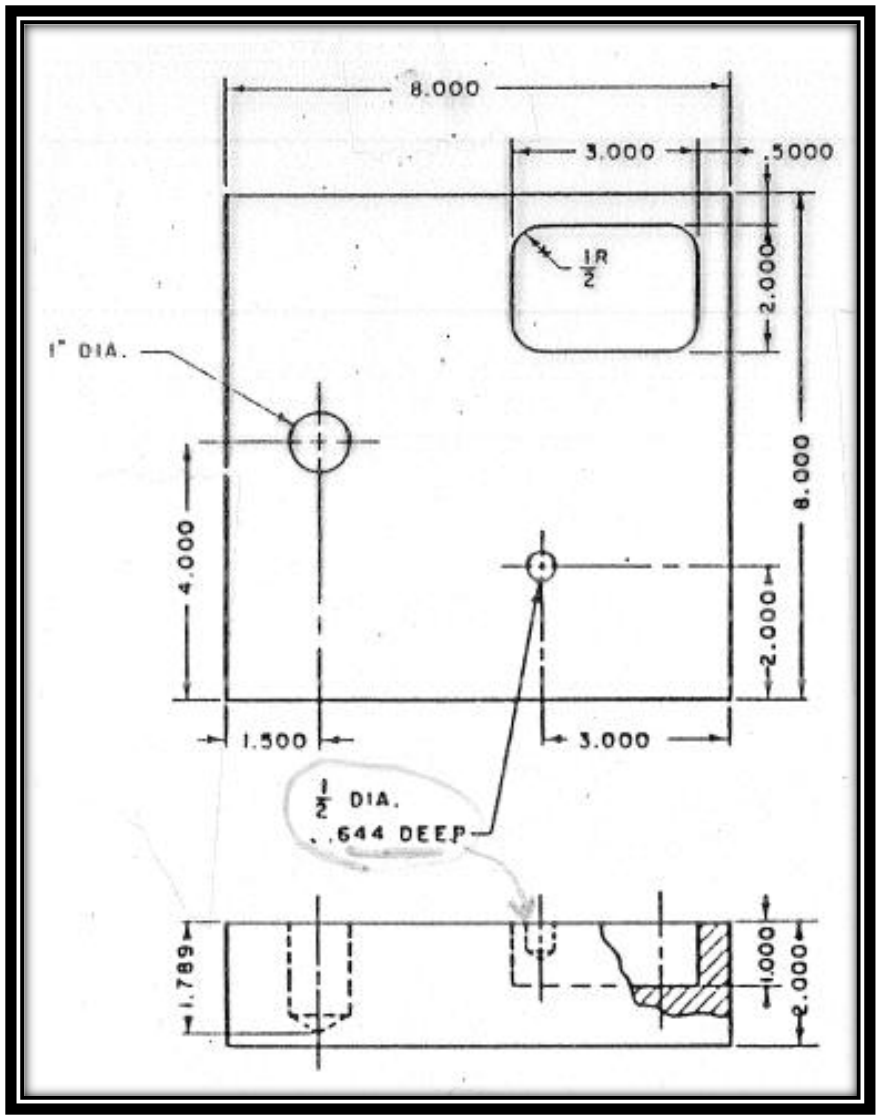
Write an NC program to machine a $\frac{1}{2}$-in.-wide L-shaped slot in a mild steel workpiece with dimensions 3 × 2.75 × 0.75 in. as shown in Figure 6.6.

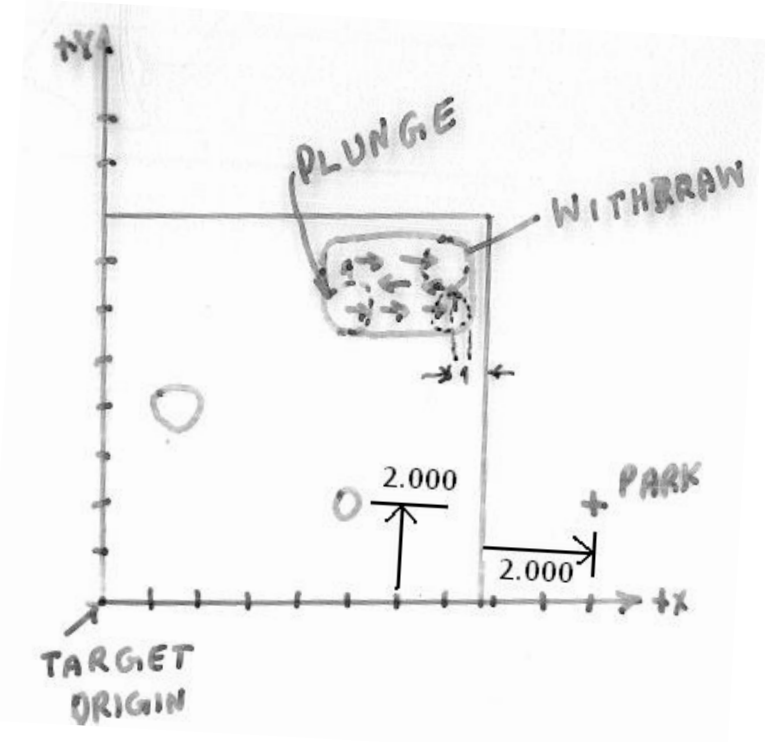
Assumptions:

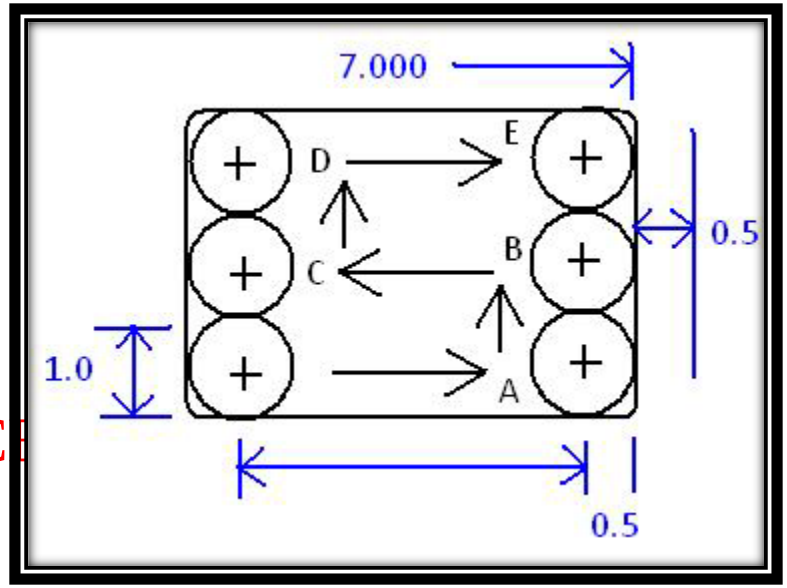
1. The top lower left corner will be used for program zero.
2. Machining motion will start in the indicated position.
3. The tool is $\frac{1}{4}$ in. above the top surface of the part prior to start of machining.
4. The tool diameter used is $\frac{1}{2}$-in., so only one pass is required.
5. A cutting speed of 500 rpm and feed rate of 10 in./min are used for machining.
6. Machine specification: N3G2X±43Y±43Z±43R±43F4S4T2M2.



%	Indicates start of program
N005 G90 G70	Specifies absolute dimensions, inch units
N010 G97 G94 T01	Specifies units for speed and feed rate; loads first tool
N015 G00 X1000 Y3000 Z250 F0	Rapid positioning of tool to start point
N020 G01 Z-375 M03 S500 F10	Turns on spindle, feeds tool to required depth
N025 Y750	Machines the vertical portion of the L
N030 X2500	Machines the horizontal portion of the L
N035 Z250	Retracts tool to 0.25 in above part surface
N040 X-1000 Y-1000 F0	Moves to safe location at rapid rate
N045 M30	Turns off all machine functions







- TOOL 0.500 ABOVE SURFACE
- ABSOLUTE DIMENSION

n	x	y	z	f	m	*
001	1500	4000	0000	1500		*
002			-2289	20		*
003			0000	1500		*
004	10000	2000	0500	1500	06	*
005	5000				03	*
006			-1144	20		*
007			0000	1500		*
008	10000	2000	0500	f	06	*
009	5000	6000	0500	1500	03	*
010			-100	20		*
011	70000					*
012		6500				*
013	5000					*
014		7000				*
015	7000					*
016			0000	1500		*
017	10000	2000	0500		30	*

DRILLING 1” DIA HOLE

-TOOL PENETRATES

* - EOB

- CHANGE TOOL – PARK POSTION

- SPINDLE START CW

- TOOL PENETRATES

DRILLING 1/2” DIA HOLE

- CHANGE TOOL

- SPINDLE START CW

- TOOL PENETRATES

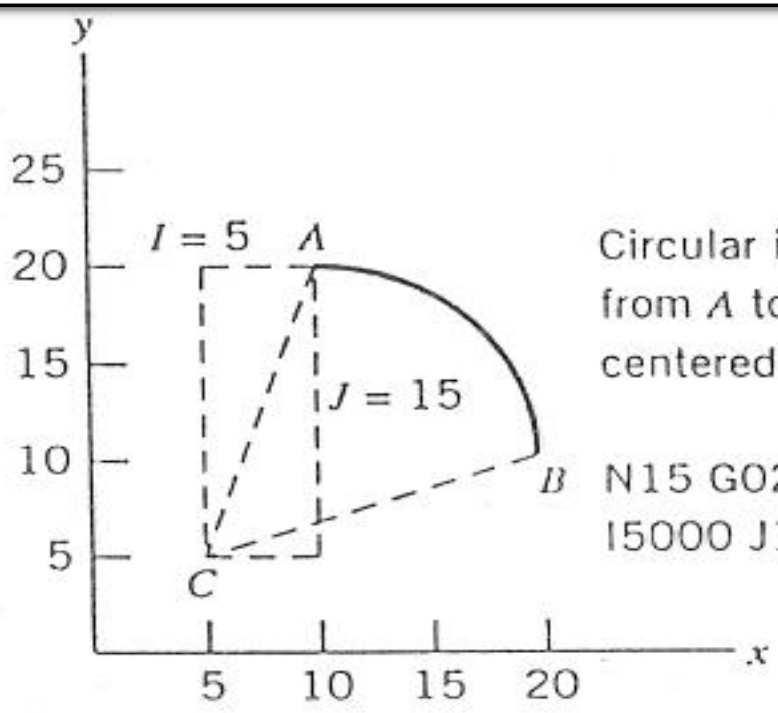
CAVITY MILLING

TOOL LIFTED UP 0.500 OF W/P SURFACE

M30 – END OF TAPE

THANK YOU

On most machines, circular interpolation can be carried out within only one of three possible planes at a time. The available planes are $x-y$, $z-x$, and $y-z$. Usually the $x-y$ plane is assumed if a plane is not explicitly specified. Codes G17–G19 are used to select the plane of operation. A further restriction on many machines is that a circular interpolation command can be effective within only one of the quadrants formed by the intersection of the axes of the coordinate system within the plane of operation, and the maximum angle of the arc is 90°. For such systems, if a circular path is required to



Circular interpolation motion from *A* to *B* about a circle centered at *C*.

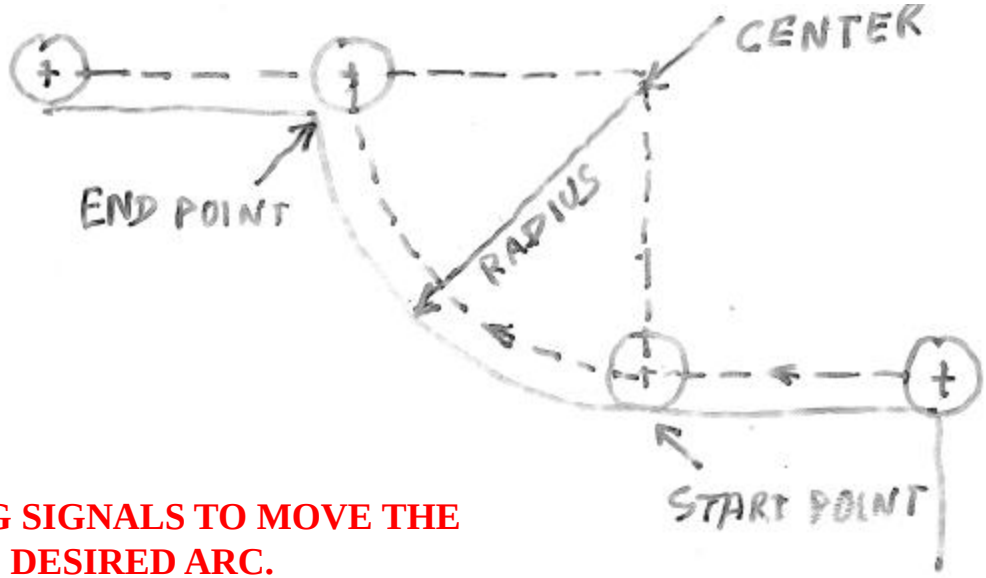
N15 G02 X20000 Y10000 I5000 J15000 F2500

CIRCULAR INTERPOLATION

FOUR PIECES OF INFORMATION

- THE DETECTION OF CUTTER TRAVEL (PREPARATORY FUNCTION
- START POINT OF THE ARC (X-Y COORDINATES)
- CENTER POINT OF THE ARC (I-J COORDINATES)
- END POINT OF THE ARC (X-Y COORDINATES)

• THE CIRCULAR INTERPLATOR AUTOMATICALLY (MCU) BREAKS UP THE ARC INTO VERY SMALL LINEAR MOVES, GENERALLY 0.0025 mm OR 0.005 mm EACH TO DESCRIBE THE CIRCULAR PATH



• MCU GENERATES CONTROLLING SIGNALS TO MOVE THE CUTTING TOOL TO PRODUCE THE DESIRED ARC.

• THE START POINT OF THE ARC IS USUALLY THE END POINT OF THE LINEAR LINE OR THE END POINT OF A PREVIOUS ARC.

• I,J,K WORDS ARE INCREMENTAL VALUES REGARDLESS – ABSOLUTE / INCREMENTAL