NC PART PROGRAMMING

WHAT IT IS COMPRISED OF?

COLLECTION OF DATA

ARRANGEMENT OF INFORMATION IN A STANDARD FORMAT.

CALCULATION OF TOOL PATH
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.
  - Precedes 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.
ANGULAR DIMENSION WORD : A, B, C AROUND X, Y, Z, RESPECTIVELY

FOR ANGULAR DIMENSION AROUND SPECIAL AXIS D, E.

DIMENSION 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.
**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 QUADRANT – 2 OR MORE BLOCKS OF INFORMATION.

**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.
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’ ACCEPTS 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)
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
- 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
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 DRILLING 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.
6. X values are to be programmed as diameters.
% 
N005 G90 G70  
N010 G98 G42 T01  
N015 G00 X2200 Z2600 F0  
N020 X1800 M03 S1200 F0  

N025 G01 Z500 F12  
N030 X1900  
N035 G00 Z2600 F0  
N040 X1600 F0  
N045 G01 Z500 F12  
N050 X1700  
N055 G00 Z2600 F0  
N060 X1500 F0  
N065 G01 Z500 F12  
N070 X2200  
N075 G00 X5000 Z5000 F0  
N080 M30  

Indicates start of program  
Specifies absolute programming, inch units  
Specifies units for speed and feed rate, loads 1st tool  
Rapid positioning of tool to tool start position  
Position tool to remove 0.1 in. off part diameter, start spindle  
Feed tool into workpiece  
Retract tool (overlap previous cut)  
Move tool clear of workpiece  
Position tool to remove 0.1 in. off part diameter  
Feed tool into workpiece  
Retract tool (overlap previous cut)  
Move tool clear of workpiece  
Position tool to take finish cut  
Feed tool into workpiece  
Retract tool clear of the workpiece  
Move to safe position  
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

ABSOLUTE DIMENSION, METRIC UNIT

UNIT S FOR SPEED, FEED & TOOL CHANGE

N001 G90 G71
N005 G98 G95 T01
N010 G00 X26000 Z66000 F0
N015 M03 S750 M08
N020 G31 X23000 F225
N025 Z23000
N030 X23500
N035 G00 Z66000 F0
N040 G01 X21000 F225
N045 Z25000
N050 X21500
N055 G00 Z66000 F0
N060 G01 X18000 F225
N065 X21000 Z50000
N070 X21500
N075 G00 Z66000 F0
N080 G01 X16000 F225
N085 X21000 Z40000
N090 X21500
N095 G00 Z66000 F0
N100 G01 X15000 F225
N105 X20000 Z40000
N110 Z25000
N115 G03 X25000 Z20000 I5000 K5000
N120 G01 X26000 M09
N125 G00 Z66000 F0 M30

Rapid move to tool start position (SP)
Turn on spindle and coolant
Position tool for first cut (DOC=2mm)
First rough cut
TAKE AWAY FROM WP
GO TO RHS RAPIDLY
Position tool for second cut (DOC=2mm)
Second rough cut
TAKE AWAY FROM WP
RHS SAFE POSITION RAPIDLY
Position for start of rough taper
First rough taper
OFF THE WP
RHS SAFE POSITION
Position for second rough taper
Second rough taper
OFF THE WP
RHS SAFE POSITION
Position for start of finishing cut
Finish taper
Finish 40 mm diameter
Finish 5 mm radius
Clear the part, turn off coolant
Move to safe place, turn off all machine functions

END
Write an NC program to machine a \( \frac{1}{2} \)-in.-wide L-shaped slot in a mild steel workpiece with dimensions \( 3 \times 2.75 \times 0.75 \) 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.
Indicates start of program
Specifies absolute dimensions, inch units
Specifies units for speed and feed rate; loads first tool
Rapid positioning of tool to start point
Turns on spindle, feeds tool to required depth
Machines the vertical portion of the L
Machines the horizontal portion of the L
Retracts tool to 0.25 in above part surface
Moves to safe location at rapid rate
Turns off all machine functions
• TOOL 0.500 ABOVE SURFACE
• ABSOLUTE DIMENSION
<table>
<thead>
<tr>
<th>n</th>
<th>x</th>
<th>y</th>
<th>z</th>
<th>f</th>
<th>m</th>
<th>*</th>
</tr>
</thead>
<tbody>
<tr>
<td>001</td>
<td>1500</td>
<td>4000</td>
<td>0000</td>
<td>1500</td>
<td></td>
<td>*</td>
</tr>
<tr>
<td>002</td>
<td></td>
<td>-2289</td>
<td>20</td>
<td></td>
<td></td>
<td>*</td>
</tr>
<tr>
<td>003</td>
<td></td>
<td>0000</td>
<td>1500</td>
<td></td>
<td></td>
<td>*</td>
</tr>
<tr>
<td>004</td>
<td>10000</td>
<td>2000</td>
<td>0500</td>
<td>1500</td>
<td>06</td>
<td>*</td>
</tr>
<tr>
<td>005</td>
<td>5000</td>
<td></td>
<td></td>
<td></td>
<td>03</td>
<td>*</td>
</tr>
<tr>
<td>006</td>
<td></td>
<td>-1144</td>
<td>20</td>
<td></td>
<td></td>
<td>*</td>
</tr>
<tr>
<td>007</td>
<td></td>
<td>0000</td>
<td>1500</td>
<td></td>
<td></td>
<td>*</td>
</tr>
<tr>
<td>008</td>
<td>10000</td>
<td>2000</td>
<td>0500</td>
<td>f</td>
<td>06</td>
<td>*</td>
</tr>
<tr>
<td>009</td>
<td>5000</td>
<td>6000</td>
<td>0500</td>
<td>1500</td>
<td>03</td>
<td>*</td>
</tr>
<tr>
<td>010</td>
<td></td>
<td>-100</td>
<td>20</td>
<td></td>
<td></td>
<td>*</td>
</tr>
<tr>
<td>011</td>
<td>70000</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>*</td>
</tr>
<tr>
<td>012</td>
<td></td>
<td>6500</td>
<td></td>
<td></td>
<td></td>
<td>*</td>
</tr>
<tr>
<td>013</td>
<td>5000</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>*</td>
</tr>
<tr>
<td>014</td>
<td>7000</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>*</td>
</tr>
<tr>
<td>015</td>
<td>7000</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>*</td>
</tr>
<tr>
<td>016</td>
<td></td>
<td>0000</td>
<td>1500</td>
<td></td>
<td></td>
<td>*</td>
</tr>
<tr>
<td>017</td>
<td>10000</td>
<td>2000</td>
<td>0500</td>
<td>30</td>
<td></td>
<td>*</td>
</tr>
</tbody>
</table>

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 $90^\circ$. For such systems, if a circular path is required to
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