LECTURE #10

Computer Aided Manufacturing

Computer aided manufacturing

- Introduction to CAM
- Part programming
- CNC programming
- Automatically programmed tools (APT)
- Computer aided part programming
- Mould tool design
- Rapid prototyping
- CMM inspection

What is manufacturing ?

Manufacturing is a set of correlated operations and activities which includes product design, material selection, planning, production inspection, management, and marketing of the products, for the manufacturing industries.

Manufacturing Processes are the lower level activities (casting, forming, machining, joining etc.) used to make products.

Manufacturing System is an organization that comprises several interrelated manufacturing subsets. Its objective is to interface with outside production functions in order to optimize the total productivity performance of the system.

[Wang et al, 98]

Four Centuries of Manufacturing

Early 18th Century	19th Century	20th Century	21th Century
A person with an anvil and hammer Poorly understood process Craftspeople Cottage industry	Steam-powered machinery Improved understanding of process Factory conditions in cities	Computer aided design, planning, and manufacturing Limited process models using closed loop control Increased factory automation	Systemwide networks and information Robust processes and intelligent control Global enterprises and virtual manufacturing corporation

CAM

CAM is the use of computers and computer technology to assist in all phases of manufacturing a product, including process and production planning, machining, scheduling, management and quality control.

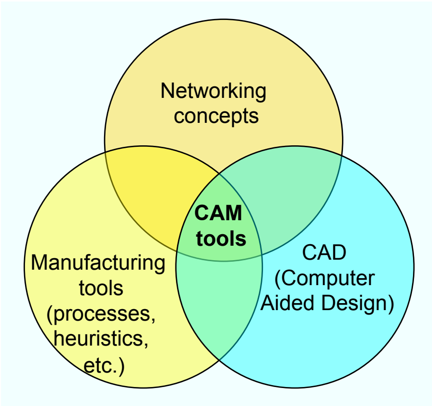
In the integrated CAD/CAM Systems:

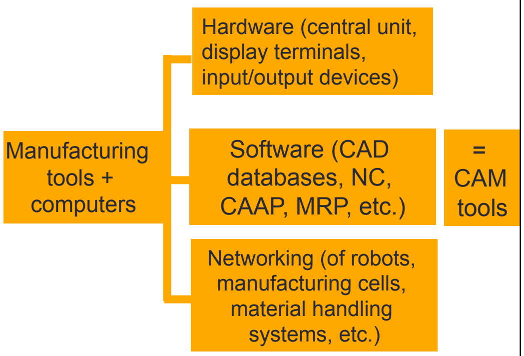
- CAD & CAM modules share common database
- No need for data exchange or recreation
- Automatic tool path generation
- Toll path verification and prevention of possible tool collisions with jigs, fixtures or other interferences
- Coding and classification of parts using alphanumeric coding
- Standardized product development
- Reduced design effort, tryout and prototype work
- Reduced cost and increased productivity

Into to CAM

- the use of computer-based software tools that assist engineers and machinists in manufacturing or prototyping product components.
- a programming tool that allows you to make 3D models using computer-aided design (CAD).
- been considered as an NC programming tool wherein 3D models of components generated in CAD software are used to generate CNC code to drive numerical controlled machine tools.

Definition of CAM tools based on their constituents





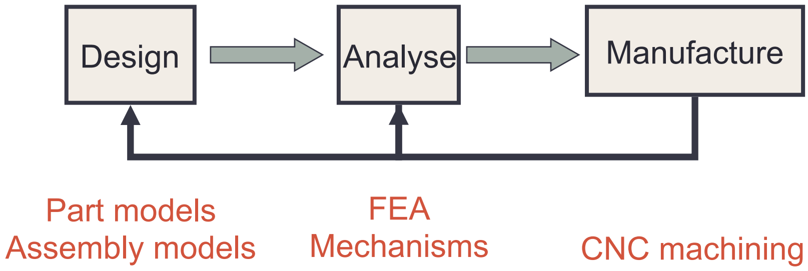
Definition of CAM tools based on their implementation in a design environment

Typical areas of concern

- High Speed Machining, including streamlining of tool paths
- Multi-function Machining
- 5 Axis Machining
- Ease of Use

The Design & Manufacturing Process

Interface between CAD and Manufacturing process to make a part



Background

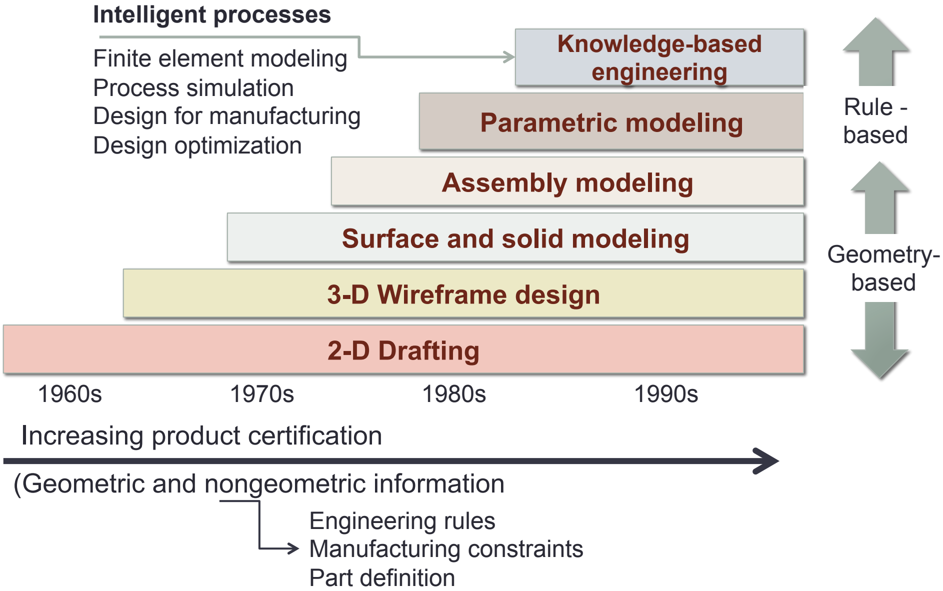
- In late 1940s John Parson devised a method for manufacture of smooth shapes for templates of aircrafts wing section using punched cards.
- At first work started at MIT USA and in parallel in UK.
- Appreciating this idea, by the mid to late 1950s Numerically Controlled tools were used in production fairly routinely.

John T. Parsons

Background

- The first commercial applications of CAM were in large companies in the automotive and aerospace industries for example UNISURF in 1971 at Renault (Bezier) for car body design and tooling.

Evaluation of CAD and CAM



Anatomy of CAM

- ***Tooling Model Analysis***
  Tooling model received from CAD
  Surface integrity analysis
  Draft analysis
  Gouge analysis.
- ***Cutter Path Definition***
  Type of cut
  Amount of stock required
  Roughing or finishing requirements
- ***Cutter Path Generation***
  CL data visualization
  Observation of retracts
  Editing of data as necessary
  Post-processing for specific control
- ***Cutter Path Verification***
  Solid model vs. hard tool
  Collision avoidance
  Gouge detection
  Testing of various machining techniques
- ***Transfer to CNC***
  Transfer data directly to CNC
  Operator setup information
  Clamp location

CAM contents & tools …

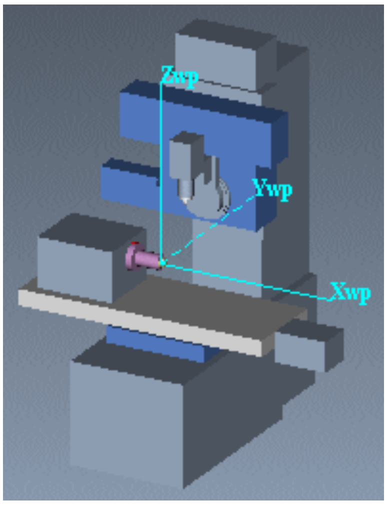
- 2 main factors that determine the success of CAM implementation
  - the link between CAD and CAM must be a two-way route
  - the hardware and software networking of the various CAM elements
- Timely synchronization among robots, vision systems, manufacturing cells, material handling systems, and other shop-floor tasks is most important

CAM contents & tools …

- Manufacturing process begins with the process planning.
- Process planning is the backbone of the manufacturing process.
- The outcome of the process planning is a production plan, tools procurement, material order, and machine programming.

Spectrum of CAM Capabilities

- CNC machinery to produce high-quality tooling for both prototype and production applications.
- Conventional CNC Milling
- High speed CNC Milling
- Electrostatic Discharge Maching (EDM)



Integrated CAD /CAM Tools

Mastercam (from CNC software Inc.)
  A system for generating 2-through 5-axis milling, turning, wire EDM, lasers, mold base development and 3D design and drafting

Virtual Gibbs (from Gibbs & Assoc.)
  A powerful, full featured CAM system for NC programming

Varimetrix (from Varimetrix Corp.)
  A system with design modeling, CAM (planning, resource management and CNC programming) and drafting.

Pro/MANUFACTURING (integrated with Pro/E)
  A system for generating machine code (CNC code for 3 axis milling, turning and wire EDM) to produce parts.

Integrated CAD /CAM Tools

SURFCAM (from Surfware Inc.)
  An outgrowth of the Diehl family's machine shop
  A system for generating 2 – 5 axis milling, turning, drilling and wire EDM
  Toolpath verification (MachineWorks Ltd.)

Rhinoceros (NURBS modeling)
  Industrial, marine and jewelery designs, CAD/CAM, rapid prototyping and reverse engineering.

PART PROGRAMMING

Graphical Part Programming

Basically,

1. Part geometry is entered in 2D or 3D.
2. Tool geometry and machine tool type are entered.
3. Speeds and feeds are entered or calculated based on tool and work material.
4. Inside/outside of geometry, and initial stock sizes are selected.
5. Cutter paths are generated.
6. Cutter paths are converted to a machine specific language (eg, G-codes).

These programs are usually built into better CAD systems or are available as stand alone software

Some machine tools have these programmers built into the controller.

Manual NC Programming

In the control of NC machines, the programmer (usually called the part programmer) writes the instructions for the machine tool.

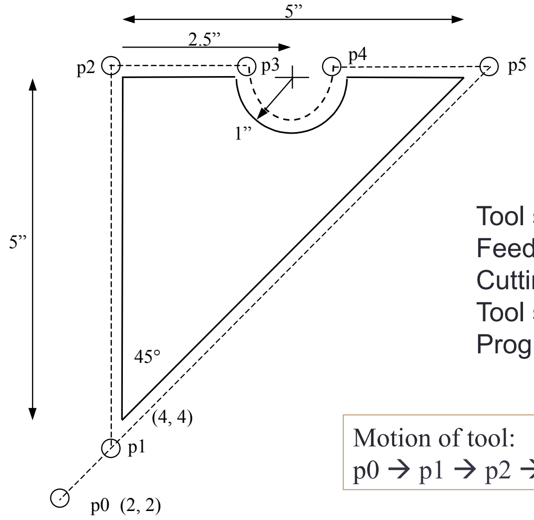
Standard Part programming language:
RS 274-D (Gerber, GN-code)

Manual NC programming

Part program: A computer program to specify

- Which tool should be loaded on the machine spindle;
- What are the cutting conditions (speed, feed, coolant ON/OFF etc)
- The start point and end point of a motion segment
- how to move the tool with respect to the machine.

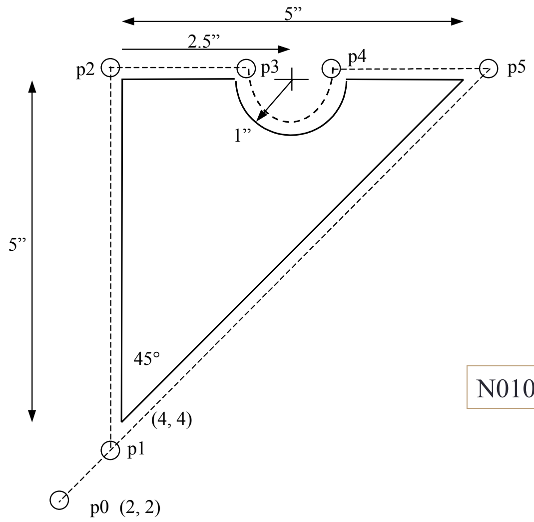
Manual Part Programming Example

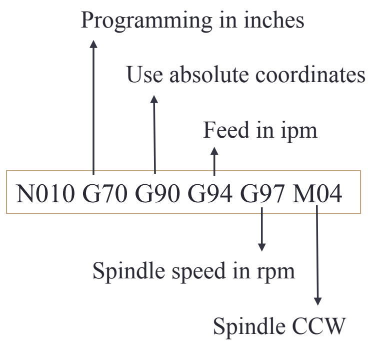


Tool size = 0.25 inch,
Feed rate = 6 inch per minute,
Cutting speed = 300 rpm,
Tool start position: 2.0, 2.0
Programming in inches

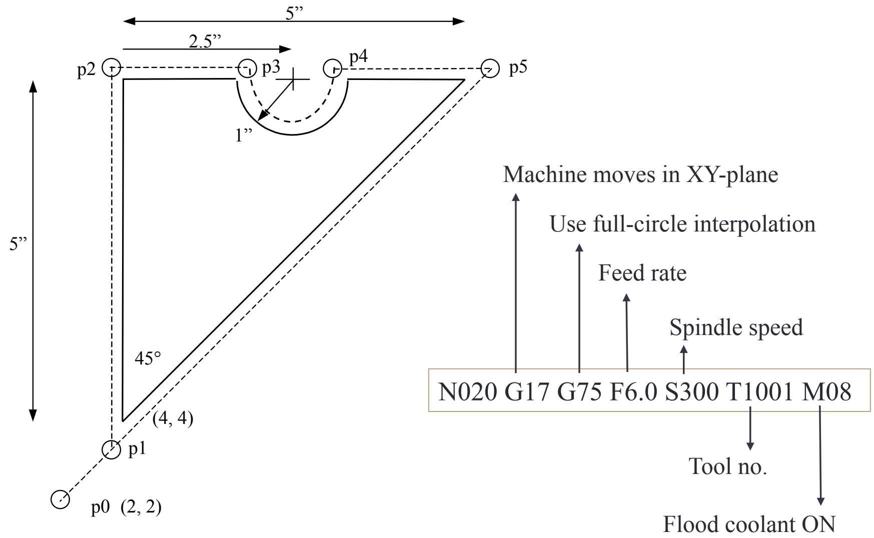
Motion of tool:
p0 → p1 → p2 → p3 → p4 → p5 → p1 → p0

1. Set up the programming parameters

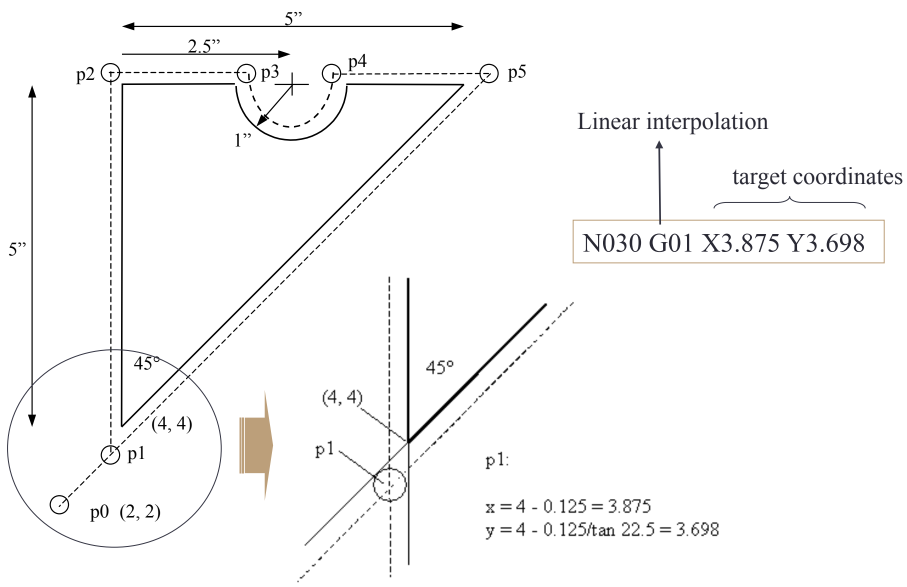




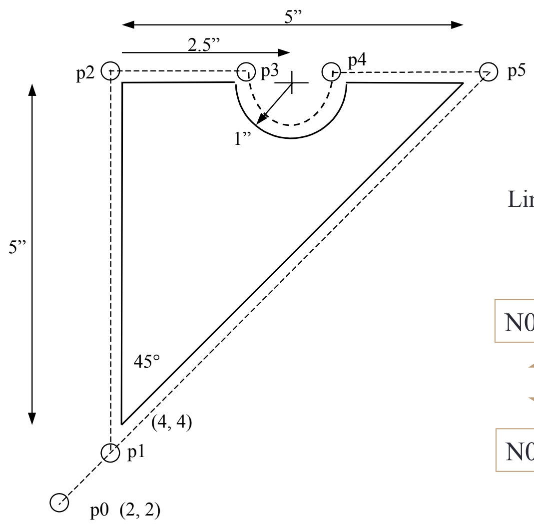
2. Set up the machining conditions



3. Move tool from p0 to p1 in straight line



4. Cut profile from p1 to p2



Linear interpolation

target coordinates

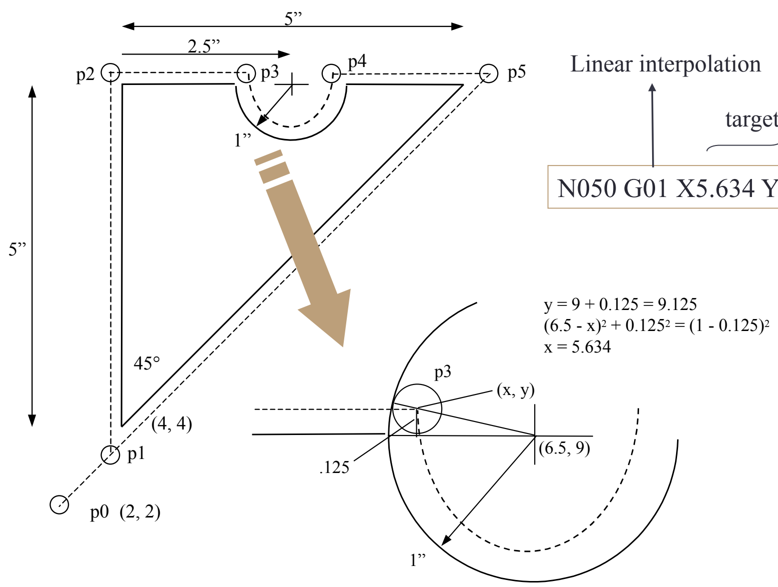
N040 G01 X3.875 Y9.125

or

N040 G01 Y9.125

X-coordinate does not change ➔ no need to program it

5. Cut profile from p2 to p3

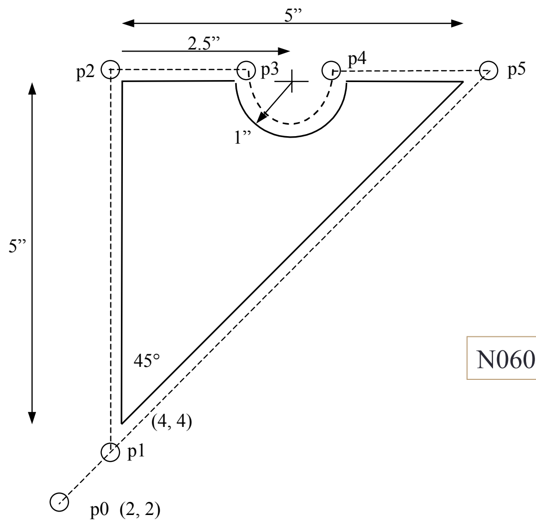


Linear interpolation

target coordinates

N050 G01 X5.634 Y9.125

6. Cut along circle from p3 to p4



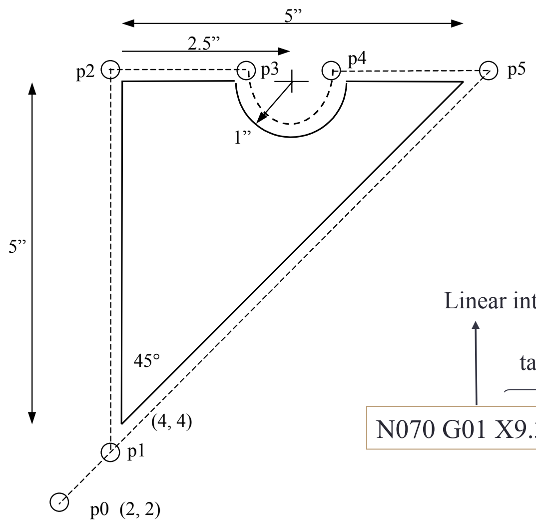
circular interpolation, CCW motion

target coordinates

N060 G03 X7.366 Y9.125 I6.5 J9.0

coordinates of center of circle

7. Cut from p4 to p5

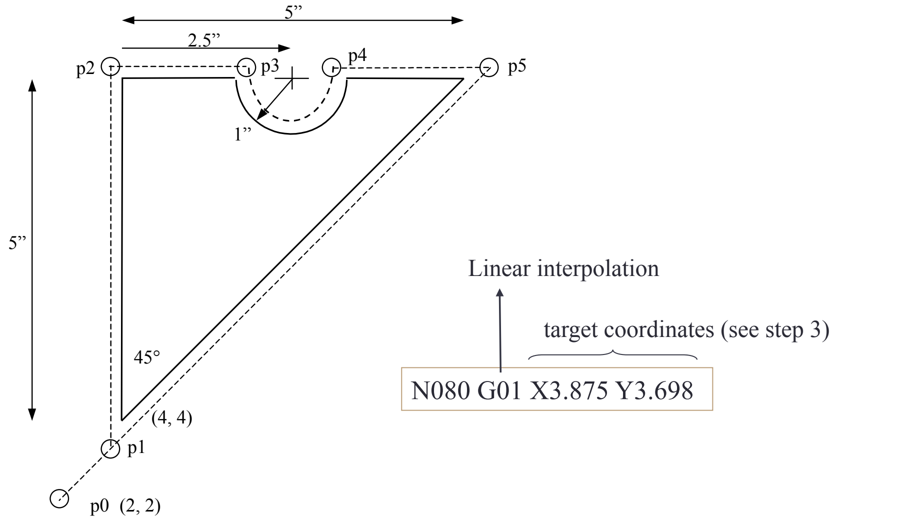


Linear interpolation

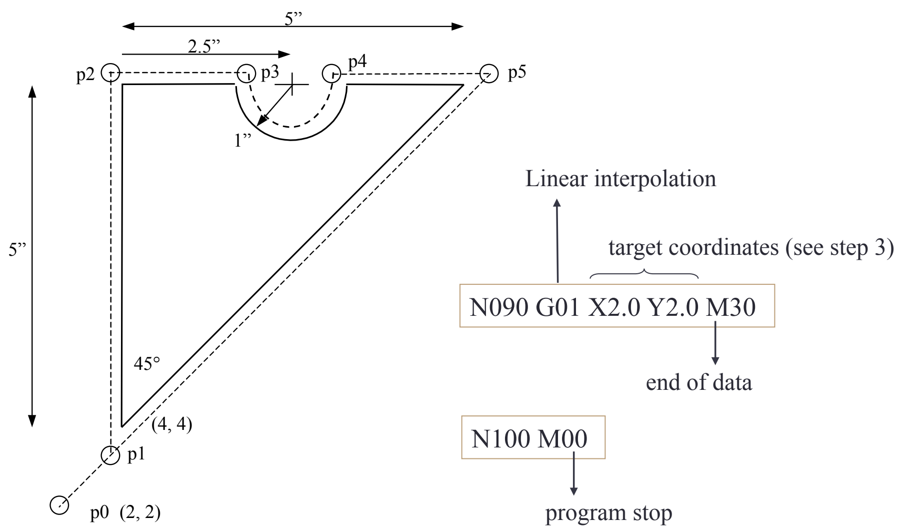
target coordinates (Y is unchanged)

N070 G01 X9.302

8. Cut from p5 to p1

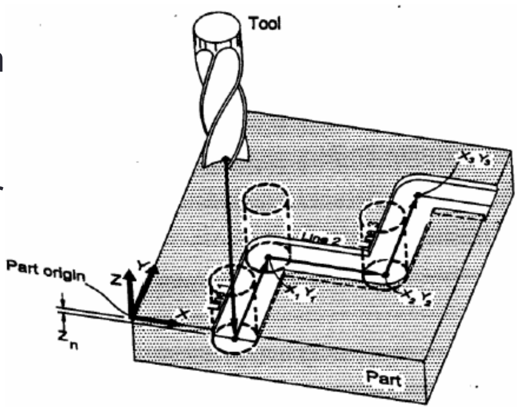


9. Return to home position,stop program



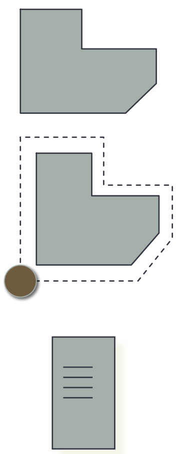
NC Part Programming

NC programmers coordinate the transition between mathematical CAD models to NC cutter paths required for CNC machining.



Linear tool motion (Milling) relative to the part

NC Part Programming Process



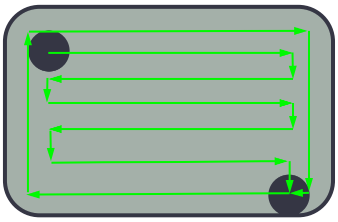
Part geometry	CAD data
Tool paths	Feeds, speeds, tools, paths, etc.
Cutter Location Data	Neutral file (APT)
G-code	Machine-specific G-code

NC Cutter Paths

When we have simple features, paths are easy to generate. These features include,

- steps
- pockets
- holes
- etc.

Typically paths for these will repeat as shown below,

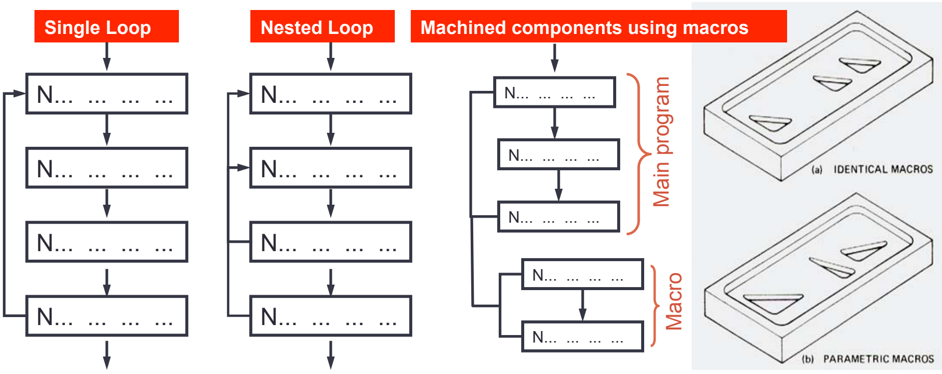


Elements of NC Programming

- Tool selection, feeds and speeds
- Cutter path specification
  - interpolation
  - tool offset calculations
  - tool interference (gouge avoidance)
- Generation of NC instructions

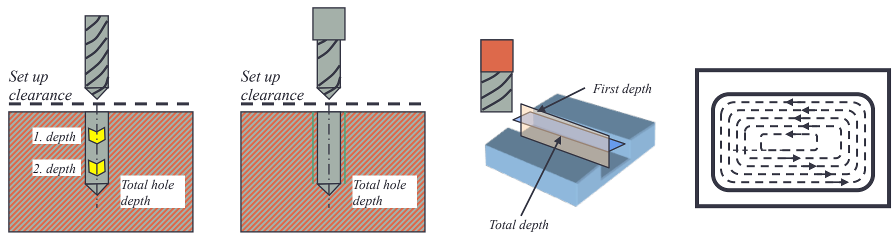
Program Features

- **Program Loops**
  - **Single and Nested Loops**
    - To minimize laborious programming steps (e.g. for a number of identical drill holes at an equal incremental distance
  - **Macros or Subroutines**
    - This may be called repeatedly in a part program, possibly with variable parameters to provide variable numerical data to the program
    - Macros may be system type (integral with system software) or user defined

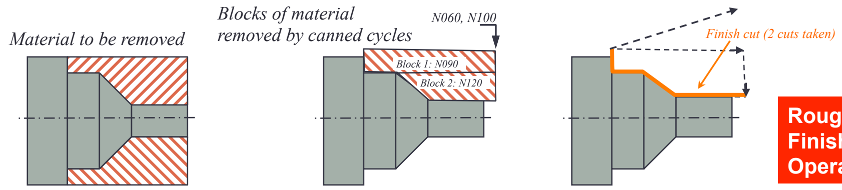


Program Features

- **Canned cycles or fixed cycles**
  - Automatic procedures for common machining operations that involve repeated moves
  - Applied to operations such as rough cutting of typical volumes, drilling, tapping and threading cycles etc



Cyle 1: Drill/Peck Drill
Cyle 2: Tapping
Cyle 3: Slot milling
Cyle 4: Pocket milling



Roughing & Finishing Operation

A simple algorithm to cut the surface

```
dim_flag = 1; a direction flag
n=10 ; number of passes to cut the surface
step=1.0/n ; step sizes for u and v directions
start=step/2 ; the start offset in the u and v directions
[xp,yp,zp] = p(start,start) ; calculate the start position
print("G00 X",xp," Y",yp," Z",zp+0.2) ; move the tool to above the start position
for i=0 to (n-1) ; will increment in the u direction
        for j=0 to (n-1) ; will increment in the v direction
        ; calculate next point
        if dim_flag=-1 then [xp,yp,zp]=p(start+i*step,start+j*step)
        if dim_flag=1 then [xp,yp,zp]=p(start+i*step,start+(n-j)*step)
        print("G01 X",xp," Y",yp," Z",zp) ; issue instruction to cut to next point
        next j ; make next step in v direction until done
        dim_flag = -dim_flag ; reverse direction to cut in opposite direction
next i ; move to next cut line in the u direction
print("G00 Z",zp+0.2) ; move the tool to above the end position
```

NC & CNC Programing

Some common programming languages include,
(note: standards are indicated with an *)

ADAPT - (ADaptation of APT) A subset of APT
***APT** - (Automatically Programmed Tool) A geometry based language that is compiled into an executable program.
AUTOSPOT - A 2D language developed by IBM. Later combined with ADAPT.
COMPACT/COMPACTII - A higher level language designed for geometrical definitions of parts, but it doesn't require compilation.
EXAPT - An European flavor of APT
***G-Codes** (EIA RS-274 G&M codes)
MAPT - (Microcomputer APT) - Yet another version of APT
UNIAPT - APT controller for smaller computer systems
Other Proprietary languages

CNC PROGRAMMING

CNC Programming Basics

- CNC instructions are called part program commands.
- When running, a part program is interpreted one command line at a time until all lines are completed.
- Commands, which are also referred to as blocks, are made up of words which each begin with a letter address and end with a numerical value.

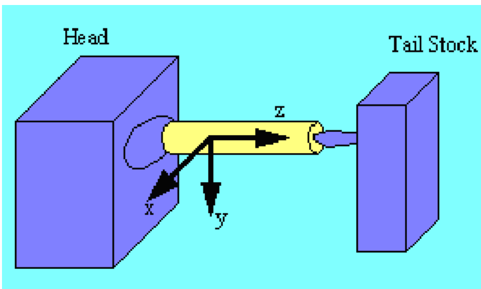
CNC programming

Important things to know:

- Coordinate System
- Units, incremental or absolute positioning
- Coordinates: X,Y,Z, RX,RY,RZ
- Feed rate and spindle speed
- Coolant Control: On/Off, Flood, Mist
- Tool Control: Tool and tool parameters

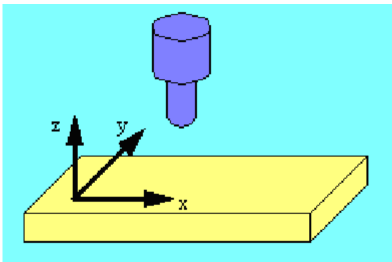
CNC Programing

For a lathe, the infeed/radial axis is the x-axis, the carriage/length axis is the z-axis. There is no need for a y-axis because the tool moves in a plane through the rotational center of the work. Coordinates on the work piece shown below are relative to the work.



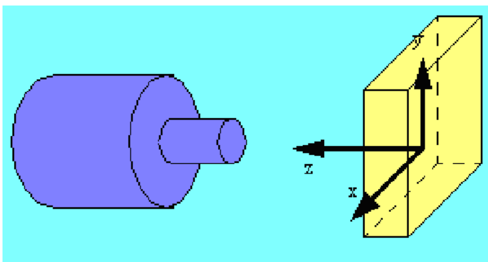
CNC Programing

For a tool with a vertical spindle the x-axis is the cross feed, the y-axis is the in-feed, and the z-axis is parallel to the tool axis (perpendicular to the table). Coordinates on the work piece shown below relative to the work.



CNC Programing

For a tool with a horizontal spindle the x-axis is across the table, the y-axis is down, and the z-axis is out. Coordinates on the work piece shown below relative to the work.



Tool Paths, Cutting & Plotting Methods

- Tool paths describes the route the cutting tool takes.
- Motion can be described as point to point, straight cutting or contouring.
- Speeds are the rate at which the tool operates e.g. rpm.
- Feeds are the rate at which the cutting tool and work piece move in relation to each other.
- Feeds and speeds are determined by cutting depth, material and quality of finish needed. e.g. harder materials need slower feeds and speeds.
- Rouging cuts remove larger amounts of material than finishing cuts.
- Rapid traversing allows the tool or work piece to move rapidly when no machining is taking place.

Programming Key Letters

O - Program number (Used for program identification)

N - Sequence number (Used for line identification)

G - Preparatory function

X - X axis designation

Y - Y axis designation

Z - Z axis designation

R - Radius designation

F – Feed rate designation

S - Spindle speed designation

H - Tool length offset designation

D - Tool radius offset designation

T - Tool Designation

M - Miscellaneous function

Int to CNC Programming

CNC machines are programmed using a simple language with words.

Each word is specified by one alphabetic character and is followed by a numeric value.

CNC machine consists of a line number (N word) followed by a sequence of words.

Each letter address relates to a specific machine function. “G” and “M” letter addresses are two of the most common.

Program Command Parameters

Optimum machine programming requires consideration of certain machine operating parameters including:

- Positioning control
- Compensations
- Special machine features

Positioning control is the ability to program tool and machine slide movement simultaneously along two or more axes. Positioning may be for point-to-point movement or for contouring movement along a continuous path. Contouring requires tool movement along multiple axes simultaneously. This movement is referred to as “Interpolation” which is the process of calculating intermediate values between specific points along a programmed path and outputting those values as a precise motion. Interpolation may be linear having just a start and end point along a straight line, or circular which requires an end point, a center and a direction around the arc.

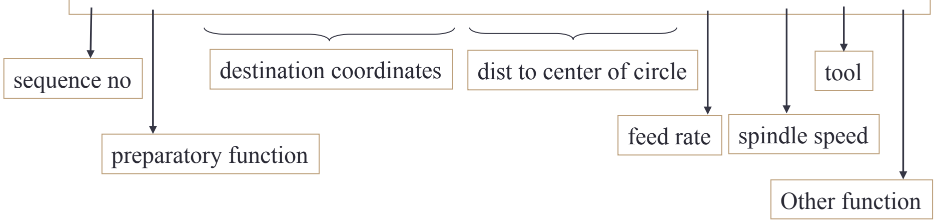
CNC Programming

The RS274-D is a **word address format**

Each line of program == 1 **block**
Each block is composed of several instructions, or (words)

Sequence and format of words:

N3 G2 X+1.4 Y+1.4 Z+1.4 I1.4 J1.4 K1.4 F3.2 S4 T4 M2



Int to CNC Programming

Word Definition	Typical value format	Meaning
N	4 digits int	Line number in program
G	2 digits int	Preparatory code (units)
M	2 digits int	Miscellaneous func(coolant)
X,Y,Z,A,B,W, I,J,K	real number	Axis position in desired units
F	4 digits int	Feed rate value (mm/min)
S	4 digits int	Spindle speed (rpm)
T	2 digits int	tool number

Part Programming Syntax Identifiers

- **Sequence Number (Identifier N)**
  - Identifying number for block, in ascending numerical order, not necessarily in continuous sequence
- **Preparatory Function (Identifier G)**
  - Prepare MCU for a given operation, typically cutter motion
  - So important that MCD is often called 'G-Code program'
- **Dimension Data (Identifier X, Y, Z, A or B)**
  - Contains location and axis orientation data for a cutter move
- **Feed Functions (Identifier F)**
  - Use to specify cutter feed rates to be employed
- **Speed Functions (Identifier S)**
  - Use to specify spindle speed or to setup parameters for constant surface speed options
- **Tool Functions (Identifier T)**
  - Use to specify the cutter to be used, particular cutter offsets
- **Miscellaneous Functions (Identifier M)**
  - Used to designate a particular mode of operation, typically to switch a machine function such as coolant supply or spindle on or off
- **End of Block (Identifier eob)**

  \- to end a block

G - codes

A "G" letter specifies certain machine preparations such as inch or metric modes, or absolutes versus incremental modes.

- common name for the programming language that controls NC and CNC machine tools.
- Developed by the Electronic Industries Alliance in the early 1960s.
- Extensions and variations have been added independently by manufacturers, and operators of a specific controller must be aware of differences of each manufacturers' product.

G-Code Generation

- CNC machines require programs in G-code format
- G-code is a low-level NC programming language
- There are variations between machines, so machine-specific post-processors are used
- This language was originally designed to be read from paper tapes. As a result it is quite simple.
- The language directs tool motion with simple commands

G-Code Generation

The typical sequence of one of these programs

1. Introductory functions such as units, absolute coords. vs. relative coords., etc.
2. Define coordinates.
3. Feeds, speeds, etc.
4. Coolants, doors, etc.
5. Cutting tool movements and tool changes
6. Shutdown

Explanation of commonly used G codes

- G00 – Preparatory code to control final position of the tool and not concerned with the path that is followed in arriving at the final destination.
- G01 – Tool is required to move in a straight line connecting current position and final position. Used for tool movement without any machining- point to point control. (linear interpolation)
- G02 – Tool path followed is along an arc specified by I, J and K codes.(circular interpolation)

Preparatory Functions (G Codes)

- Tool motion
  - Rapid traverse G00
    - Positioning command
    - Moves the tool at a rapid feed rate to a specific XYZ coordinate
    - Takes the shortest route to reach the specified point
  - Format
    - G00 Xx Yy Zz

Table of Important G codes

G00 Rapid Transverse
G01 Linear Interpolation
G02 Circular Interpolation, CW
G03 Circular Interpolation, CCW
G17 XY Plane,G18 XZ Plane,G19 YZ Plane
G20/G70 Inch units
G21/G71 Metric Units
G40 Cutter compensation cancel
G41 Cutter compensation left
G42 Cutter compensation right
G43 Tool length compensation (plus)
G43 Tool length compensation (plus)
G44 Tool length compensation (minus)
G49 Tool length compensation cancel
G80 Cancel canned cycles
G81 Drilling cycle
G82 Counter boring cycle
G83 Deep hole drilling cycle
G90 Absolute positioning
G91 Incremental positioning

Complete list of `G' operation codes

G00 - Rapid move (not cutting)
G01 - Linear move
G02 - Clockwise circular motion
G03 - Counterclockwise circular motion
G04 - Dwell
G05 - Pause (for operator intervention)
G08 - Acceleration
G09 - Deceleration
G17 - x-y plane for circular interpolation
G18 - z-x plane for circular interpolation
G19 - y-z plane for circular interpolation
G20 - turning cycle or inch data specification
G21 - thread cutting cycle or metric data specification
G24 - face turning cycle
G25 - wait for input #1 to go low (Prolight Mill)
G26 - wait for input #1 to go high (Prolight Mill)
G28 - return to reference point
G29 - return from reference point
G31 - Stop on input (INROB1 is high) (Prolight Mill)
G33-35 - thread cutting functions (Emco Lathe)
G35 - wait for input #2 to go low (Prolight Mill)
G36 - wait for input #2 to go high (Prolight Mill)
G40 - cutter compensation cancel
G41 - cutter compensation to the left
G42 - cutter compensation to the right
G43 - tool length compensation, positive
G44 - tool length compensation, negative
G50 - Preset position
G70 - set inch based units or finishing cycle
G71 - set metric units or stock removal
G72 - indicate finishing cycle (EMCO Lathe)
G72 - 3D circular interpolation clockwise (Prolight Mill)
G73 - turning cycle contour (EMCO Lathe)
G73 - 3D circular interpolation counter clockwise (Prolight Mill)
G74 - facing cycle contour (Emco Lathe)
G74.1 - disable 360 deg arcs (Prolight Mill)
G75 - pattern repeating (Emco Lathe)
G75.1 - enable 360 degree arcs (Prolight Mill)
G76 - deep hole drilling, cut cycle in z-axis
G77 - cut-in cycle in x-axis

Complete list of `G' operation codes

G78 - multiple threading cycle
G80 - fixed cycle cancel
G81-89 - fixed cycles specified by machine tool manufacturers
G81 - drilling cycle (Prolight Mill)
G82 - straight drilling cycle with dwell (Prolight Mill)
G83 - drilling cycle (EMCO Lathe)
G83 - peck drilling cycle (Prolight Mill)
G84 - taping cycle (EMCO Lathe)
G85 - reaming cycle (EMCO Lathe)
G85 - boring cycle (Prolight mill)
G86 - boring with spindle off and dwell cycle (Prolight Mill)
G89 - boring cycle with dwell (Prolight Mill)
G90 - absolute dimension program
G91 - incremental dimensions
G92 - Spindle speed limit
G93 - Coordinate system setting
G94 - Feed rate in ipm (EMCO Lathe)
G95 - Feed rate in ipr (EMCO Lathe)
G96 - Surface cutting speed (EMCO Lathe)
G97 - Rotational speed rpm (EMCO Lathe)
G98 - withdraw the tool to the starting point or feed per minute
G99 - withdraw the tool to a safe plane or feed per revolution
G101 - Spline interpolation (Prolight Mill)

M - codes

A "M" letter specifies miscellaneous machine functions and work like on/off switches for coolant flow, tool changing, or spindle rotation. Other letter addresses are used to direct a wide variety of other machine commands.

- referred to as a "Miscellaneous" function
- control the overall machine, causing it to stop, start, turn on coolant, etc., whereas other codes pertain to the path traversed by cutting tools.
- Different machine tools may use the same code to perform different functions; even machines that use the same CNC control.

Stop and End Functions

- M00 **unconditional stop** shuts down all drive motors until re-started by the machinist
- M01 **optional stop** will stop the program if the optional stop button is pressed
- M02/M30 **end of program statement**

Table of Important M codes

M00 Program stop
M01 Optional program stop
M02 Program end
M03 Spindle on clockwise
M04 Spindle on counterclockwise
M05 Spindle stop
M06 Tool change
M08 Coolant on
M09 Coolant off
M10 Clamps on
M11 Clamps off
M30 Program stop, reset to start

Complete list of M-Codes

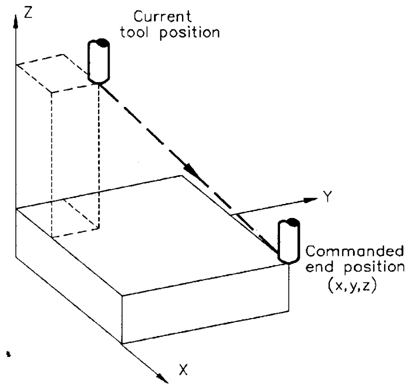
M00 - program stop
M01 - optional stop using stop button
M02 - end of program
M03 - spindle on CW
M04 - spindle on CCW
M05 - spindle off
M06 - tool change
M07 - flood with coolant
M08 - mist with coolant
M08 - turn on accessory #1 (120VAC outlet) (Prolight Mill)
M09 - coolant off
M09 - turn off accessory #1 (120VAC outlet) (Prolight Mill)
M10 - turn on accessory #2 (120VAC outlet) (Prolight Mill)
M11 - turn off accessory #2 (120VAC outlet) (Prolight Mill) or tool change
M17 - subroutine end
M20 - tailstock back (EMCO Lathe)
M20 - Chain to next program (Prolight Mill)
M21 - tailstock forward (EMCO Lathe)
M22 - Write current position to data file (Prolight Mill)
M25 - open chuck (EMCO Lathe)
M25 - set output #1 off (Prolight Mill)
M26 - close chuck (EMCO Lathe)
M26 - set output #1 on (Prolight Mill)
M30 - end of tape (rewind)
M35 - set output #2 off (Prolight Mill)
M36 - set output #2 on (Prolight Mill)
M38 - put stepper motors on low power standby (Prolight Mill)
M47 - restart a program continuously, or a fixed number of times (Prolight Mill)
M71 - puff blowing on (EMCO Lathe)
M72 - puff blowing off (EMCO Lathe)
M96 - compensate for rounded external curves
M97 - compensate for sharp external curves
M98 - subprogram call
M99 - return from subprogram, jump instruction
M101 - move x-axis home (Prolight Mill)
M102 - move y-axis home (Prolight Mill)
M103 - move z-axis home (Prolight Mill)

Other codes and keywords include

Annn - an orientation, or second x-axis spline control point
Bnnn - an orientation, or second y-axis spline control point
Cnnn - an orientation, or second z-axis spline control point, or chamfer
Fnnn - a feed value (in ipm or m/s, not ipr), or thread pitch
Innn - x-axis center for circular interpolation, or first x-axis spline control point
Jnnn - y-axis center for circular interpolation, or first y-axis spline control point
Knnn - z-axis center for circular interpolation, or first z-axis spline control point
Lnnn - arc angle, loop counter and program cycle counter
Nnnn - a sequence/line number
Onnn - subprogram block number
Pnnn - subprogram reference number
Rnnn - a clearance plane for tool movement, or arc radius, or taper value
Qnnn - peck depth for pecking cycle
Snnn - cutting speed (rpm), spindle speed
Tnnn - a tool number
Unnn - relative motion in x
Vnnn - relative motion in y
Wnnn - relative motion in z
Xnnn - an x-axis value
Ynnn - a y-axis value
Znnn - a z-axis value
; - starts a comment (proLight Mill), or end of block (EMCO Lathe)

Linear Interpolation

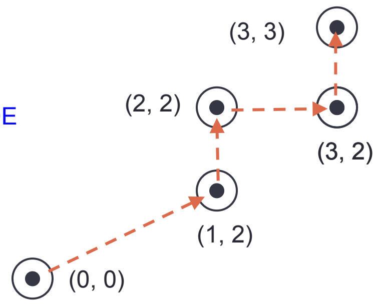
- Moves the tool from its current position to a specific XYZ coordinate at a specified feed rate
- Format
  - G01 Xx Yy Zz ff



Linear interpolation

- When cutting, it is useful to change our point of reference.
- When doing mathematics we tend to dimension relative to a main origin (absolute).
- In fact a machine will need to have coordinates specified with reference to a main origin.
- These relative points refer to as local origins.

N0010 G90; PUT IN ABSOLUTE MODE
N0011 G01X1Y2; MOVE TO (1,2)
N0012 G01X2Y2; MOVE TO (2,2)
N0013 G91; PUT IN INCREMENTAL MODE
N0014 G01X1 ; MOVE TO (3,2)
N0015 G92X2Y2; SET NEW ORIGIN
N0016 G01X1Y1; MOVE TO (3,3)
ABSOLUTE N0017 G92X0Y0Z0; RESET THE ZERO



Circular Interpolations

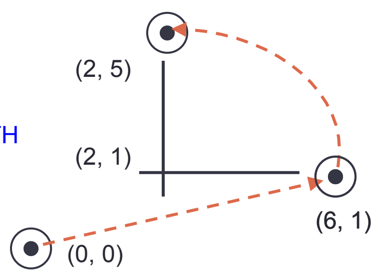
- Moves a tool around a circular arc to a specific XYZ coordinate
  - Requires **5 pieces** of information
    - Plane selection
    - Arc start point
    - Rotation direction
    - Arc end point
    - Arc center or arc radius
- G02 circular interpolation clockwise around an arc
  G03 circular interpolation counter clockwise around an arc

Circular interpolation

Valid for both milling and turning.
move to the start point, the command indicates the direction (clockwise or counterclockwise).
The I, J values indicate the center of rotation,
The X, Y values indicate the point to stop at.
Cut circular paths on other planes by resetting the cutting planes (G17, G18, G19).

N10 G01 X6Y1; MOVE TO (6,1)

N11 G03 X2Y5I2J1; CUT CIRCULAR PATH



Radius Method

- Requires two entry parameters in the command the **XYZ end point of the arc and the radius R**
- G02/G03 Xx Yy Zz Rr

Sequence and format of words

N3 G2 X+1.4 Y+1.4 Z+1.4 I1.4 J1.4 K1.4 F3.2 S4 T4 M2 **eob**

N3: Sequence number followed by three integers, e.g. N001, N100...
G2: Preparatory function followed by two integers.
X+1.4: X dimension, followed by sign (+ or -), one digit, a decimal, followed by 4 digits.
Y+1.4, Z+1.4: Same as above.
I1.4, J1.4, K1.4: Dimension for circular interpolation. Always positive, so no sign is specified.
F3.2: Feedrate specification, F followed by 2 digits, a decimal, then three digits.
S4: Spindle speed, 4 digits to specify the spindle rpm.
T4: Tool number is specified, using upto 4 digits.
M2: Miscellaneous function; two digits are specified.

CNC Programing

- When using the prolight mill, program elements to request that an external device (ie robot) load or unload parts can be added.
- The robot has been connected to the robotic interface port available.
- This port has four inputs and two outputs.
- The example below assumes that the input #1 indicates a part has been dropped off and the mill can start.
- Output #1 will be turned on to request that the robot pick up a part and load new stock.

```
N20 M26 ; SEND OUTPUT TO REQUEST ROBOT LOAD A PART
N21 G26 ; WAIT UNTIL THE INPUT FROM THE ROBOT INDICATES PART HERE
N22 M25 ; TURN OFF REQUEST TO ROBOT
N23 G00... ; START CUTTING THE
PART                                        .........
N89 G00... ; END PART CUTTING
N90 M26 ; SEND OUTPUT TO REQUEST PICKUP BY ROBOT, AND NEW PART
etc....
```

Rules for programming

Block Format

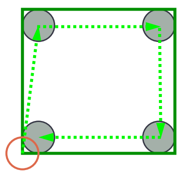
N135 G01 X1.0 Y1.0 Z0.125 F5

Sample Block

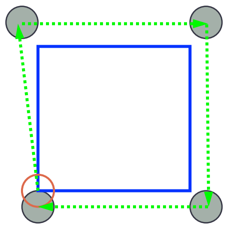
- Restrictions on CNC blocks
- Each may contain only one tool move
- Each may contain any number of non-tool move G-codes
- Each may contain only one feed rate
- Each may contain only one specified tool or spindle speed
- The block numbers should be sequential
- Both the program start flag and the program number must be independent of all other commands (on separate lines)
- The data within a block should follow the sequence shown in the above sample block

CNC Programing

- The cutter offsets calculated by hand.
- Modern NC machines keep a record of the tool geometry. (used to automatically calculate offsets)
- The best way to think of tool compensation is when cutting a profile, should we be to the left or right of the line.



G42
G00 X1.000 Y1.000
G01 Y2.000
G01 X2.000
G01 Y1.000
G01 X1.000

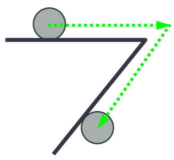


G41
G00 X1.000 Y1.000
G01 Y2.000
G01 X2.000
G01 Y1.000
G01 X1.000

CNC Programing

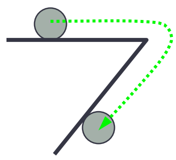
Additional commands to help with former problems.

M97 – compensate for corners larger than step (requires more time)



G41
G01 X4.000
G01 X1 Y1 M97

M96 – compensate for corners

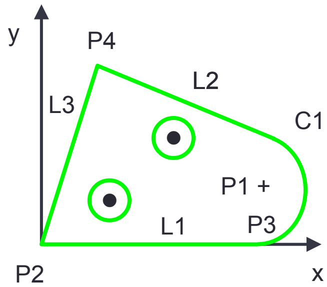


G41
G01 X4.000
G01 X1 Y1 M96

CNC Programing

Typical commanded cycles include,
- rectangular pocket milling
- circular pocket milling
- slot or elongated hole milling
- peck drilling
- tapping

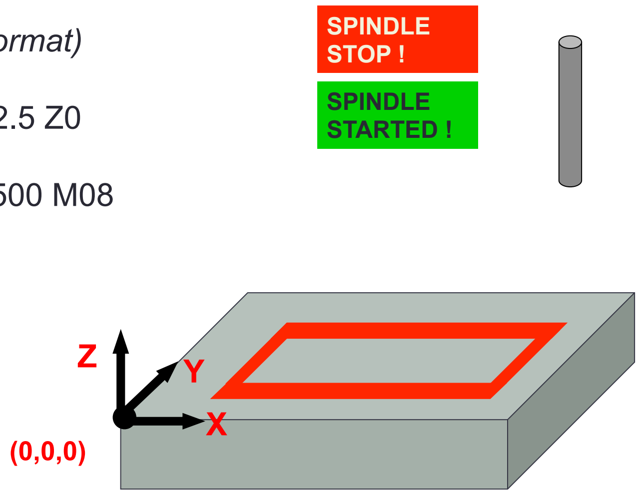
For practice, develop the part program for the component shown below



Example: A Milling Operation

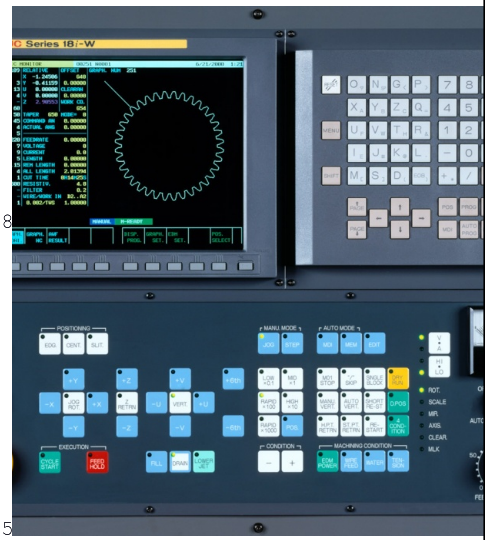
NC CODE
(Word Address Format)

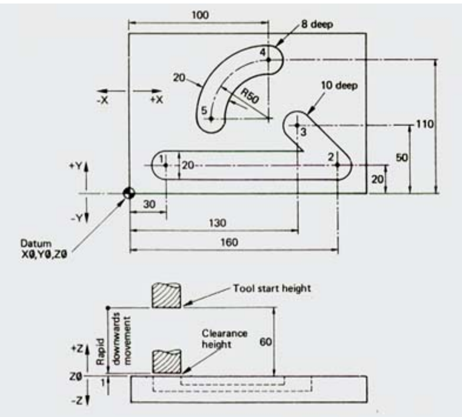
N50 G00 X15 Y12.5 Z0
N55 M03
N60 G01 Z-2.5 F500 M08
N65 G01 X50
N70 G01 Y45
N75 G01 X15
N80 G01 Y12.5
N85 G00 Z0 M09
N90 G79 M04



Sample G-Code

```
N1 G20
N2 G92 G90
N3 T01 M06
N4 M03
N5 G00 X-0.1633 Y0.8176 Z1.0008
N6 F5.0
N7 G01 Z-0.25
N15 G01 Z-0.253 F3.0
N16 X-0.1155 Y0.7022
N17 G03 X0.0 Y0.625 I0.1155 J0.0478
N18 G01 X0.75
N19 G02 X1.125 Y0.25 I0.0 J-0.375
N20 G01 Y-0.25
N21 G02 X0.75 Y-0.625 I-0.375 J0.0
N22 G01 X-0.75
N23 G02 X-1.125 Y-0.25 I0.0 J0.375
N24 G01 Y0.25
N25 G02 X-0.75 Y0.625 I0.375 J0.0
N26 G01 X0.0
N27 G03 X0.1083 Y0.6875 I0.0 J0.125
```

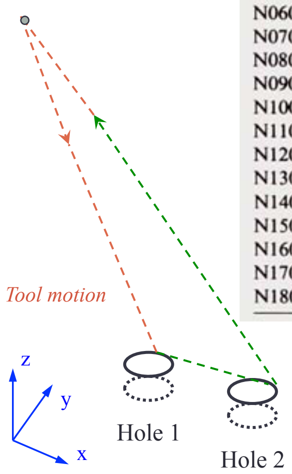




Example: Part program for Slot-Milling Operation (speeds & feeds omitted)

- Each line is a called a 'Block'
- Block is made up of a number of commands
- Each command consists of a capital letter called 'Address' followed by a numeric value to make complete command word
- Format (Arrangement of words in block for standard order)
- Each block commences with Block Sequence Number

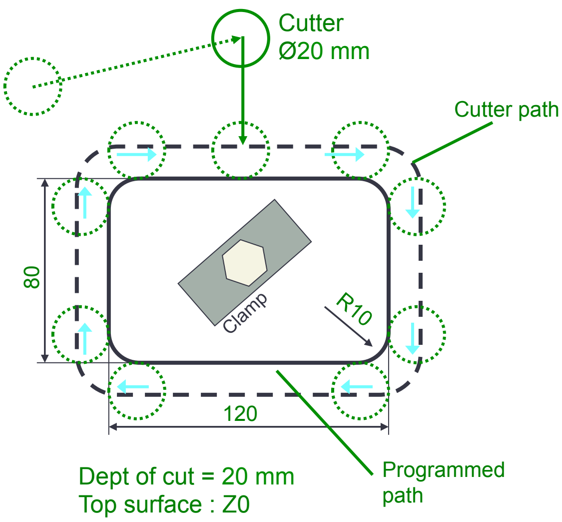
PART-PROGRAM EXTRACT						INTERPRETATION
N006	G00	X30	Y20	Z60	M03	SPINDLE ON, RAPID MOVEMENT TO POINT1 (AT Z60)
N007				Z1	M08	RAPID DOWNWARDS MOVEMENT TO Z1, COOLANT ON
N008	G01			Z-10		FEED LINEAR INTERPOLATION TO Z-10 (DEPTH OF CUT)
N009		X160				FEED LINEAR INTERPOLATION TO POINT2.
N010		X130	Y50			FEED LINEAR INTERPOLATION TO POINT3.
N011	G00			Z1		RAPID UPWARDS MOVEMENT TO Z1 AT POINT3
N012		X100	Y110			RAPID MOVEMENT TO POINT4 AT Z1.
N013	G01			Z-8		FEED LINEAR INTERPOLATION TO Z-8 (DEPTH OF CUT)
N014	G17					SET X-Y PLANE FOR CIRCULAR INTERPOLATION
N015	G03	X50	Y60	I0 J-50	M09	ACW CIRCULAR INTERPOLATION TO POINT 5, COOLANT OFF
N016	G00			Z60	M05	RAPID UPWARDS MOVEMENT TO Z60 AT POINT 5, STOP SPINDLE
N017		X0	Y0		M02	RAPID MOVEMENT TO X0,Y0 AT Z60, END PROGRAM



Program statement							*Explanation*
N010	G90						Select absolute
N020	G71						Select metric
N030	G00	X0	Y0	Z300	T01	M06	Load centre drill
N040	G00	X100	Y100	Z25			Above 1st hole
N050	G01	Z17	F400.	S3000	M03		Centre drill
N060	G00	Z25					Retract
N070	G00	X150					2nd hole
N080	G01	Z17	F400.	S3000	M03		Centre drill
N090	G00	Z25					Retract
N100	G00	X0	Y0	Z300	T02	M06	Load 10 mm diameter drill
N110	G00	X100	Y100	Z25			Above 1st hole
N120	G01	Z3	F350.	S2000	M03		Drill
N130	G00	Z25					Retract
N140	G00	X150					2nd hole
N150	G01	Z3	F350.	S2000	M03		Drill
N160	G00	Z25					Retract
N170	M00						Program stop
N180	M30						Return to start

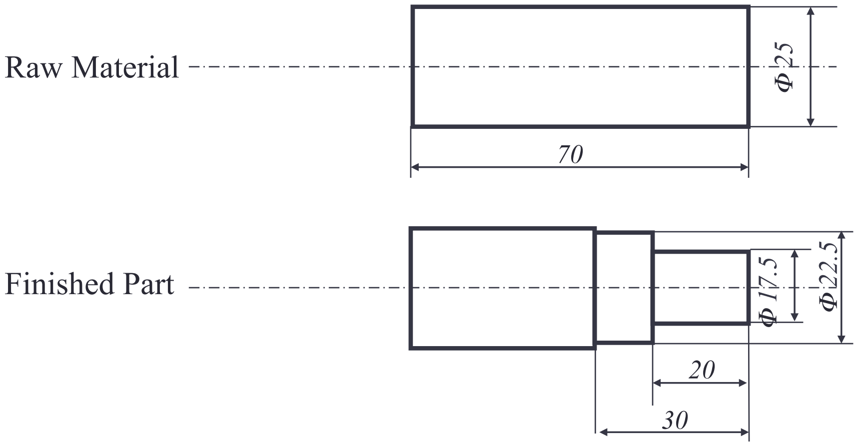
Simple example to drill two holes in a Plate

CNC Programming Example

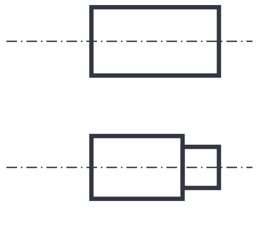


N0010G00M06T01X0.Y80.Z-20.
N0020M03M05S1500
N0030G01G41F010100Y40.
N0040X50
N0050G02X60.Y30.I0.J10.
N0060.G01Y-30
N0070G02X50.Y-40.I10.J10.
N0070G01X-50.
N0080G02X-60.Y-30.I0.J10.
N0090G01Y30.
N0100G02X-50.Y40.I10.J0.
N0110G01X0.
N0120G00G40Y80.Z0
N0130M05M09
N0140M02

Programming Example (*Cylindrical Part*)



Programming Example (*Cylindrical Part*)



```
O0013

N0005  G53
N0010  T0303
N0020  G57 G00 X26.00 Z0.0 S500 M04
N0030  G01 X-0.20 F100
N0040  G00 Z2.0
N0050  X50.0 Z50.0
N0060  T0404
N0070  G57 G00 X22.50 Z2.0 S500
N0080  G01 Z-30.0 F100
N0090  G00 X23.0 Z2.0 S500
N0100  G84 X17.5 Z-20.0 D0=200 D2=200 D3=650
N0110  G00 Z2.0
N0120  X50.0 Z50.0
N0130  M30
```

Program Interpretation

O0013

Program identification number

Program Interpretation

O0013

N0005 G53

To cancel any previous working zero point

Program Interpretation

O0013
N0005 G53
N0010 T0303

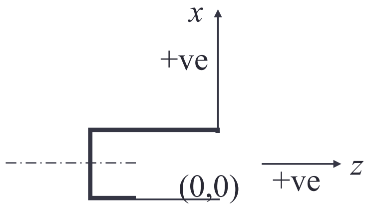
N0010 Sequence number
T0303 Select tool number 303

APT Program Interpretation

O0013

N0005 G53
N0010 T0404
N0020 G57 G00 X26.0 Z0.0 S500 M04

G57 To set the working zero point as saved
G00 Rapid movement (no cutting)
X26.0 *X* location (as a diameter; 13 form zero)
Z0.0 *Z* location
S500 Spindle speed is 500 rpm
M04 Rotate spindle counterclockwise

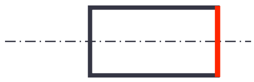


APT Program Interpretation

O0013

N0005 G53

N0010 T0404
N0020 G57 G00 X26.00 Z0.0 S500 M04
N0030 G01 X-0.20 F100



G01 Linear interpolation (cutting)
X-0.20 Move only in x direction until you pass the center by 0.1 mm (facing)
F100 Set feed rate to 100 mm/min.

Program Interpretation

O0013
N0005 G53

N0010 T0404
N0020 G57 G00 X26.00 Z0.0 S500 M04
N0030 G01 X-0.20 F100
N0040 G00 Z2.0

G00 Move rapidly away from work piece (no cutting)
Z2.0 the movement is 2 mm away from the face.

Program Interpretation

```
O0013
N0005 G53
N0010  T0404
N0020  G57 G00 X26.00 Z0.0 S500 M04
N0030  G01 X-0.20 F100
N0040  G00 Z2.0
N0050  X50.0 Z50.0
```

Go to a safe location away from the workpiece [x = 50 (25 from zero), z = 50] to change the tool.

APT Program Interpretation

```
O0013
N0005 G53
N0010  T0404
N0020  G57 G00 X26.00 Z0.0 S500 M04
N0030  G01 X-0.20 F100
N0040  G00 Z2.0
N0050  X50.0 Z50.0
N0060  T0404
```

T0404 Select tool number 404

APT Program Interpretation

O0013

N0005 G53

N0010 T0404
N0020 G57 G00 X26.00 Z0.0 S500 M04
N0030 G01 X-0.20 F100
N0040 G00 Z2.0
N0050 X50.0 Z50.0
N0060 T0404
N0070 G57 G00 X22.50 Z2.0 S500

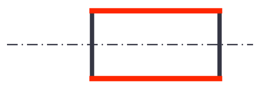
G57 PS0
G00 Rapid movement (no cutting)
X22.50 *X* location (as a diameter; 11.25 form zero)
Z2.0 *Z* location
S500 Spindle speed is 500 rpm

Program Interpretation

O0013

N0005 G53

N0010 T0404
N0020 G57 G00 X26.00 Z0.0 S500 M04
N0030 G01 X-0.20 F100
N0040 G00 Z2.0
N0050 X50.0 Z50.0
N0060 T0404
N0070 G57 G00 X25.00 Z2.0 S500 M04
N0080 G01 Z-30.0 F100



G01 Linear interpolation (cutting)
Z-30 Move only in z direction (external turning)
F100 Set feed rate to 100 mm/min.

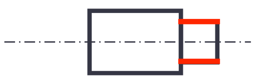
Program Interpretation

```
O0013

N0005 G53

N0010  T0404
N0020  G57 G00 X26.00 Z0.0 S500 M04
N0030  G01 X-0.20 F100
N0040  G00 Z2.0
N0050  X50.0 Z50.0
N0060  T0404
N0070  G57 G00 X25.00 Z2.0 S500 M04
N0080  G01 X22.5 Z-70.0 F100
N0090  G00 X23.0 Z2.0 S500
```

G00 Move rapidly away from work piece (no cutting) to location x= 23.0 (11.50 from zero) and z = 2.0.

Program Interpretation

```
O0013

N0005 G53
N0010  T0404
N0020  G57 G00 X26.00 Z0.0 S500 M04
N0030  G01 X-0.20 F100
N0040  G00 Z2.0
N0050  X50.0 Z50.0
N0060  T0404
N0070  G57 G00 X25.00 Z2.0 S500 M04
N0080  G01 X22.5 Z-70.0 F100
N0090  G00 X26.0 Z2.0 S500
N0100  G84 X17.5 Z-20.0 D0=200 D2=200 D3=650
```



G84 Turning cycle for machining the step
X17.5 final diameter
Z-20 length of step is 20 mm
D0=200 Finish allowance in X direction (0.2 mm) D2=200 Finish allowance in Z direction (0.2 mm)
D3=650 Depth of cut in each pass (0.65 mm)

Program Interpretation

O0013

N0005 G53
N0010 T0404
N0020 G57 G00 X26.00 Z0.0 S500 M04
N0030 G01 X-0.20 F100
N0040 G00 Z2.0
N0050 X50.0 Z50.0
N0060 T0404
N0070 G57 G00 X25.00 Z2.0 S500 M04
N0080 G01 X22.5 Z-70.0 F100
N0090 G00 X26.0 Z2.0 S500
N0100 G84 X17.5 Z-20.0 D_0=200 D_2=200 D_3=650
N0110 G00 Z2.0

G00 Move rapidly away from workpiece (no cutting)
Z2.0 the movement is 2 mm away from the face.

Program Interpretation

O0013

N0005 G53
N0010 T0404
N0020 G57 G00 X26.00 Z0.0 S500 M04
N0030 G01 X-0.20 F100
N0040 G00 Z2.0
N0050 X50.0 Z50.0
N0060 T0404
N0070 G57 G00 X25.00 Z2.0 S500 M04
N0080 G01 X22.5 Z-70.0 F100
N0090 G00 X26.0 Z2.0 S500
N0100 G84 X17.5 Z-20.0 D_0=200 D_2=200 D_3=650
N0110 G00 Z2.0
N0120 X50.0 Z50.0

X50.0 Z50.0 Move to the tool changing location

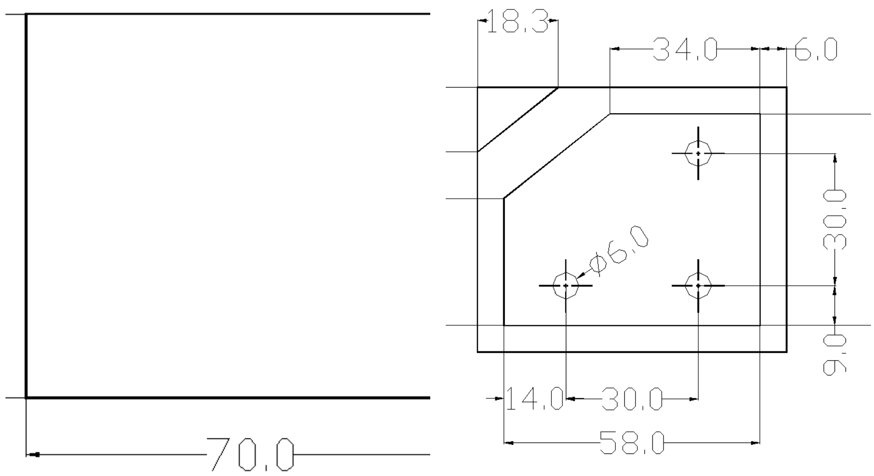
Program Interpretation

O0013

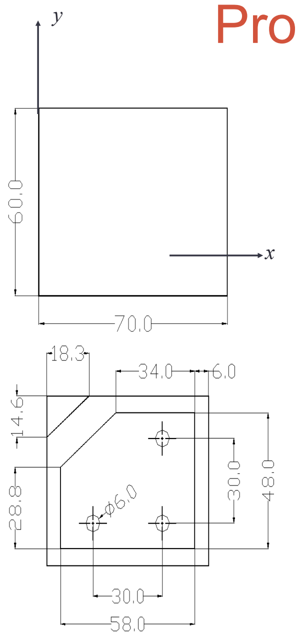
N0005 G53
N0010 T0404
N0020 G57 G00 X26.00 Z0.0 S500 M04
N0030 G01 X-0.20 F100
N0040 G00 Z2.0
N0050 X50.0 Z50.0
N0060 T0404
N0070 G57 G00 X25.00 Z2.0 S500 M04
N0080 G01 X22.5 Z-70.0 F100
N0090 G00 X26.0 Z2.0 S500
N0100 G84 X17.5 Z-20.0 D_0=200 D_2=200 D_3=650
N0110 G00 Z2.0
N0120 X50.0 Z50.0 T00
N0130 M30

M30 Program End

Programming Example

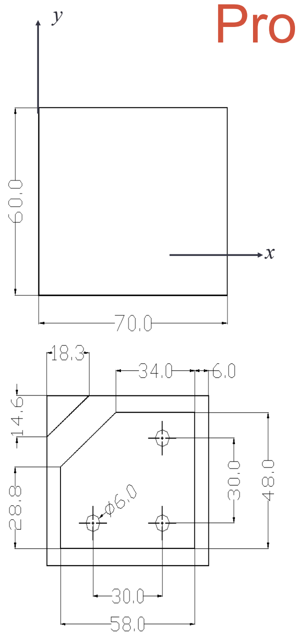


Programming Example



G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X-8 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75
N006 G01 X70 Y60 Z-0.5 XYFeed 75
N007 G01 X30 Y60 Z-0.5 XYFeed 75
N008 G01 X0 Y40 Z-0.5 XYFeed 75
N009 G01 X0 Y0 Z-0.5 XYFeed 75
N010 G81 R3 E9 N7 Z-0.5
N011 M05
N012 M02

Programming Example

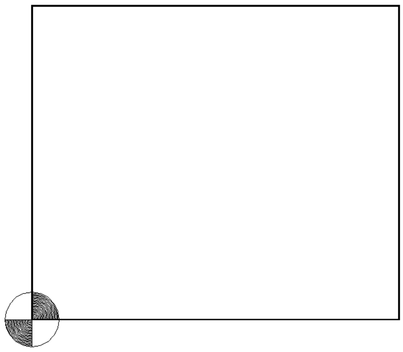


Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75
N009 G01 X50 Y15 Z10 ZFeed 150
N010 G01 X50 Y45 Z10 ZFeed 150
N011 G01 X50 Y45 Z-10 ZFeed 75
N012 G01 X50 Y45 Z10 ZFeed 150
N013 M05
N014 M02

Program Interpretation

G55 X200 Y80

Setting the datum to the lower left corner of the work piece



Program Interpretation

G55 X200 Y80
Program 1

Program Identification Number

Program Interpretation

G55 X200 Y80
Program 1
N001 M06 T1

N001 Sequence Number
M06   Tool Change (End Mill with
Diameter=12mm
T1   Tool Number

Program Interpretation

G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400

Start rotating the spindle clockwise with 400 rpm

Program Interpretation

G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150

Go to Safe Position with feed 150mm/min

Program Interpretation

G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X-8 Y0 Z-0.5 ZFeed 150

Lower the end mill to determine the depth of cut

Program Interpretation

G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X-8 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75

Move from the lower left corner of the work piece to the right lower one cutting with feed=75mm/min

Program Interpretation

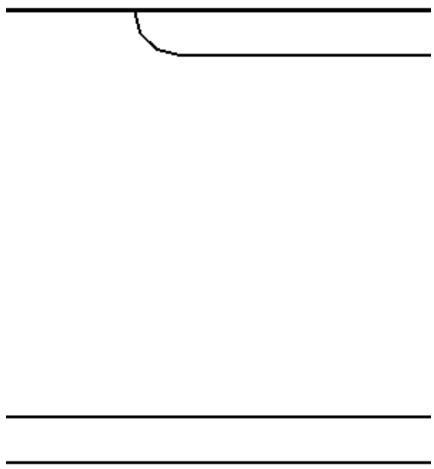
G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X-8 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75
N006 G01 X70 Y60 Z-0.5 XYFeed 75

Move from the lower left corner of the work piece to the right lower one cutting with feed=75mm/min

Program Interpretation

```
G55 X200 Y80
Program 1
N001  M06 T1
N002  M03 rpm 400
N003  G01 X-8 Y0 Z0 XYFeed 150
N004  G01 X-8 Y0 Z-0.5 ZFeed 150
N005  G01 X70 Y0 Z-0.5 XYFeed 75
N006  G01 X70 Y60 Z-0.5 XYFeed 75
N007  G01 X30 Y60 Z-0.5 XYFeed 75
```

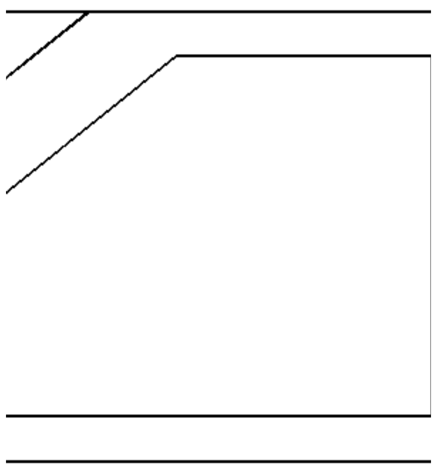
Cutting the horizontally up to X=30



Program Interpretation

```
G55 X200 Y80
Program 1
N001  M06 T1
N002  M03 rpm 400
N003  G01 X-8 Y0 Z0 XYFeed 150
N004  G01 X-8 Y0 Z-0.5 ZFeed 150
N005  G01 X70 Y0 Z-0.5 XYFeed 75
N006  G01 X70 Y60 Z-0.5 XYFeed 75
N007  G01 X30 Y60 Z-0.5 XYFeed 75
N008  G01 X0 Y40 Z-0.5 XYFeed 75
```

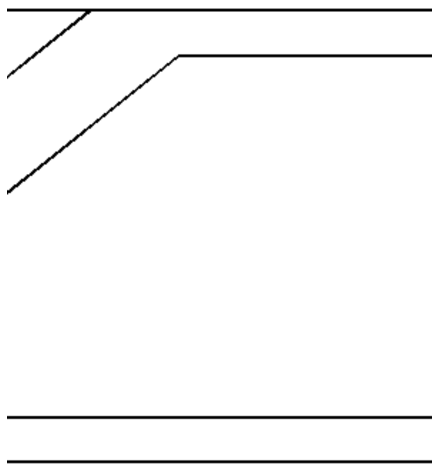
Cutting to X=0 & Y=40



Program Interpretation

G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X-8 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75
N006 G01 X70 Y60 Z-0.5 XYFeed 75
N007 G01 X30 Y60 Z-0.5 XYFeed 75
N008 G01 X0 Y40 Z-0.5 XYFeed 75
N009 G01 X0 Y0 Z-0.5 XYFeed 75

Complete the countering



Program Interpretation

G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X-8 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75
N006 G01 X70 Y60 Z-0.5 XYFeed 75
N007 G01 X30 Y60 Z-0.5 XYFeed 75
N008 G01 X0 Y40 Z-0.5 XYFeed 75
N009 G01 X0 Y0 Z-0.5 XYFeed 75
N010 G81 R3 E9 N7 Z-0.5

Repeat 7 times blocks from N003 to N009 with incremental offset of Z=-0.5

Program Interpretation

G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X-8 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75
N006 G01 X70 Y60 Z-0.5 XYFeed 75
N007 G01 X30 Y60 Z-0.5 XYFeed 75
N008 G01 X0 Y40 Z-0.5 XYFeed 75
N009 G01 X0 Y0 Z-0.5 XYFeed 75
N010 G81 R3 E9 N7 Z-0.5
N011 M05

Spindle Off

Program Interpretation

G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X-8 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75
N006 G01 X70 Y60 Z-0.5 XYFeed 75
N007 G01 X30 Y60 Z-0.5 XYFeed 75
N008 G01 X0 Y40 Z-0.5 XYFeed 75
N009 G01 X0 Y0 Z-0.5 XYFeed 75
N010 G81 R3 E9 N7 Z-0.5
N011 M05
N012 M02

End Program

Program Interpretation

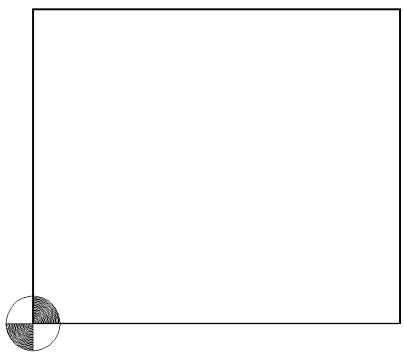
Tool Change

Changing the tool

Program Interpretation

Tool Change
G55 X200 Y80

Setting the datum to the lower left corner of the work piece



Program Interpretation

Tool Change
G55 X200 Y80
Program 2

Program Identification Number

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2

N001 Sequence Number
M06   Tool Change (Drill with Diameter=6mm
T2   Tool Number

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400

Start rotating the spindle clockwise with 400 rpm

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150

Go to Safe Position with feed 150mm/min

Program Interpretation

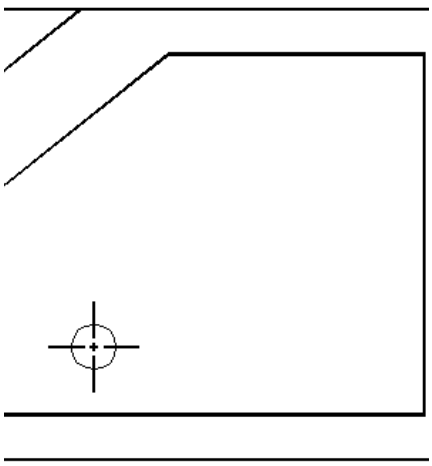
Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150

Stop above the center of the first hole

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75

Start Drill the first hole



Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150

Retract to a position above the hole

Program Interpretation

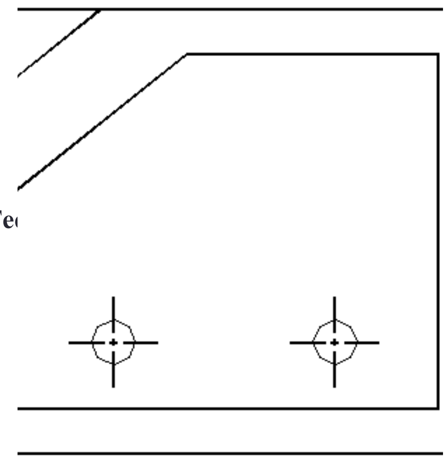
Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150

Stop above the center of the second hole

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFe
150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75

Drill the second hole



Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75
N009 G01 X50 Y15 Z10 ZFeed 150

Retract to a position above the second hole

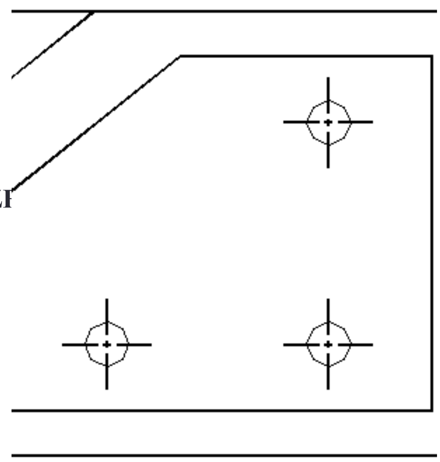
Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75
N009 G01 X50 Y15 Z10 ZFeed 150
N010 G01 X50 Y45 Z10 ZFeed 150

Stop above the center of the third hole

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZF
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75
N009 G01 X50 Y15 Z10 ZFeed 150
N010 G01 X50 Y45 Z10 ZFeed 150
N011 G01 X50 Y45 Z-10 ZFeed 75



Drill the third hole

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75
N009 G01 X50 Y15 Z10 ZFeed 150
N010 G01 X50 Y45 Z10 ZFeed 150
N011 G01 X50 Y45 Z-10 ZFeed 75
N012 G01 X50 Y45 Z10 ZFeed 150

Retract to a position above the third hole

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75
N009 G01 X50 Y15 Z10 ZFeed 150
N010 G01 X50 Y45 Z10 ZFeed 150
N011 G01 X50 Y45 Z-10 ZFeed 75
N012 G01 X50 Y45 Z10 ZFeed 150
N013 M05
Spindle off

Program Interpretation

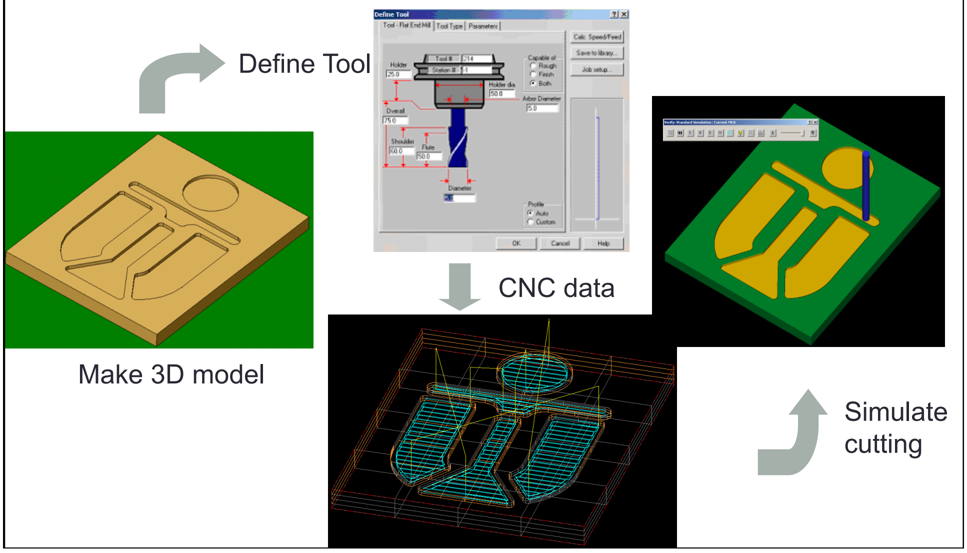
Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75
N009 G01 X50 Y15 Z10 ZFeed 150
N010 G01 X50 Y45 Z10 ZFeed 150
N011 G01 X50 Y45 Z-10 ZFeed 75
N012 G01 X50 Y45 Z10 ZFeed 150
N013 M05
N014 M02
End Program

APT

Automatically programmed tools

Automatic Part Programming

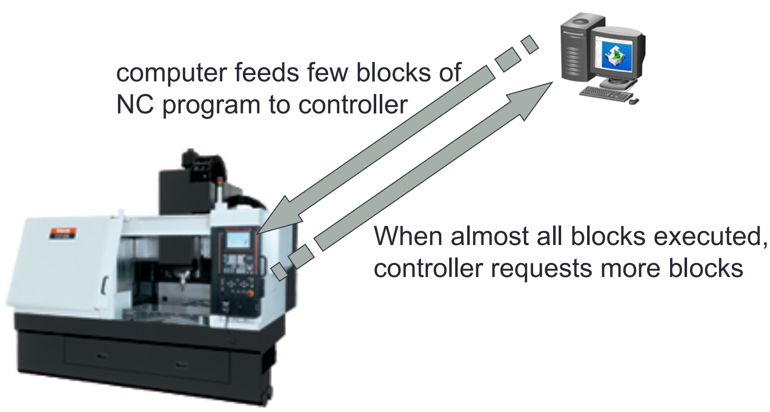
Software programs can automatic generation of CNC data



Automatic part programming

Very complex part shapes ➔ very large NC program

NC controller memory may not handle HUGE part program



Automatically Programmed Tools

This language allows tools to be programmed using geometrical shapes. This puts less burden on the programmer to do calculations in their heads.

APT programs must be converted into low level programs, such as G-codes.

Automatically Programmed Tools

Some samples of the geometrical and motion commands follow. These are not complete, but are a reasonable subset.

GEOMETRY: The simplest geometrical construction in APT is a point

p=POINT/x,y,z - a cartesian point

p=POINT/l1,l2 - intersection of two lines

p=POINT/c - the center of a circle

p=POINT/YLARGE,INTOF,l,c - the largest y intersection of a line and a circle

*Note: we can use YSMALL,XLARGE,XSMALL in place of YLARGE

Automatically Programmed Tools

GEOMETRY: Lines are one of the next simplest definitions,
l=LINE/x1,y1,z1,x2,y2,z2 - ***endpoint cartesian components***
l=LINE/p1,p2 - ***endpoints***
l=LINE/p,PARLEL,l - ***a line through a point and parallel to another line***
l=LINE/p,PERPTO,l - ***a line through a point and perpendicular to a line***
l=LINE/p,LEFT,TANTO,c - ***a line from a point, to a left tangency point on a circle***
l=LINE/p,RIGHT,TANTO,c - ***a line from a point, to a right tangency point on a circle***
l=LINE/LEFT,TANTO,c1,LEFT,TANTO,c2 - ***defined by tangents to two circles***
l=LINE/LEFT,TANTO,c1,RIGHT,TANTO,c2 - ***defined by tangents to two circles***
l=LINE/RIGHT,TANTO,c1,LEFT,TANTO,c2 - ***defined by tangents to two circles***
l=LINE/RIGHT,TANTO,c1,RIGHT,TANTO,c2 - ***defined by tangents to two circles***

Automatically Programmed Tools

GEOMETRY: Circles are very useful for constructing geometries

c=CIRCLE/x,y,z,r - ***a center and radius***
c=CIRCLE/CENTER,p,RADIUS,r - ***a center point and a radius***
c=CIRCLE/CENTER,p,TANTO,l - ***a center and a tangency to an outside line***
c=CIRCLE/p1,p2,p3 - ***defined by three points on the circumference***
c=CIRCLE/YLARGE,l1,YLARGE,l2,RADIUS,r - ***tangency to two lines and radius***

*Note: we can use YSMALL,XLARGE,XSMALL in place of YLARGE

Automatically Programmed Tools

GEOMETRY: More complex geometric constructions are possible

PLANE/ - ***defines a plane***
QUADRIC/a,b,c,d,e,f,g,h,i,j - ***define a polynomial using values***
GCONIC/a,b,c,d,e,f - ***define a conic by equation coefficients***
LCONIC/p1,p2,... - ***defines a conic by lofting (splining) points***
RLDSRF/ - ***a ruled surface made of two splines***
POLCON/ - ***define a surface using cross sections***
PATERN/ - ***will repeat a motion in a linear or circular array***

Automatically Programmed Tools

MOTION: We can use the basic commands to follow the specified geometry
FROM/p - ***specify a start point***
FROM/x,y,z - ***specify a start point***
GOTO/p - ***move to a final point***
GOTO/x,y,z - ***move to a final point***
GOTO/TO,p - ***move until the tool touches a point***
GOTO/TO,l - ***move until the tool touches a line***
GOTO/TO,c - ***move until the tool touches a circle***
GOLFT/l1,TO,l2 - ***go on the left of l1 until the tool touches l2***
GORGT/l1,TO,l2 - ***go on the right of l1 until the tool touches l2***
GOBACK/l1,TO,l2 - ***reverses direction along l1 to l2***
GOBACK/l1,TO,c1 - ***reverses direction along l1 to c1***
GOUP/l1,TO,l2 - ***goes up along l1 to l2***
GODOWN/1l,TO,l2 - ***goes down along l1 to l2***
GODLTA/x,y,z - ***does a relative move***
Note: TO can be replaced with PAST, ON to change whether the tool goes past the structure, or the center stops on the structure.

Automatically Programmed Tools

MOTION: The following commands will create complex motion of the tool
POCKET/ - ***will cut a pocket***
PSIS/ - ***will call for the part surface***

· As would be expected, we need to be able to issue commands to control the machine.

CONTROL: The following instructions will control the machine outside the expected cutting tool motion.
CUTTER/n1,n2 - ***defines diameter n1 and radius n2 of cutter***
MACHIN/n,m - ***uses a post processor for machine `n', and version `m'***
COOL/ANT/n - ***either MIST, FLOOD or OFF***
TURRET/n - ***sets tool turret to new position***
TOLER/n - ***sets a tolerance band for cutting***
FEDRAT/n - ***sets a feedrate n***
SPINDL/n,CW - ***specifies n rpm and direction of spindle***

Some program elements that are only used for programming can also be included.

PROGRAM: The following statements are programming support instructions
REMARK - ***starts a comment line that is not interpreted***
$$ - ***also allows comments, but after other statements***
NOPOST - ***turns off the post processor that would generate cutter paths***
CLPRNT - ***prints a sequential history of the cutter center location***
SQRTF(n) - ***calculates the floating point square root***
FINI - ***stop program***
PARTNO/n - ***allows the user to specify the part name***
LOOPST and LOOPND - ***loop instructions***
RESERV/n,m - ***defines an array of size `n' by `m'***
JUMPTO/n - ***jump to line number***

· **Note:** variables can also be defined and basic mathematical operations can be performed.

· **Note:** macro functions are also available.

Sample CL DATA file

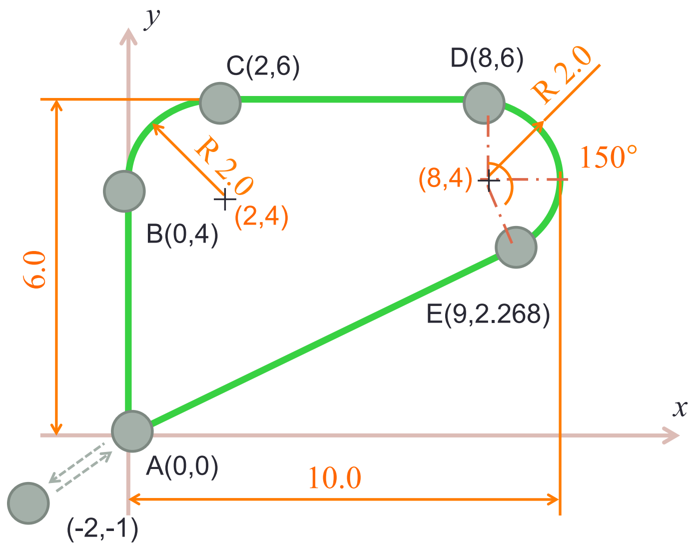
```
PARTNO/'mold'
UNITS/MM
PPRINT/'OPERATION CATEGORY & TYPE: Milling Copy Mill'
PPRINT/'OPERATION NUMBER & NAME:   Operation-2'
PPRINT/'TOOL IDENTIFIER:           25mm ball mill'
PPRINT/'POST TOOL ID:              0'
PPRINT/'TOOL DESCRIPTION:          '
PPRINT/'TOOL STATION NUMBER:       2'
MODE/MILL
MULTAX/OFF
LOADTL/0, IN, 2, LENGTH, 0.000000, OSETNO, 0
CUTTER/25.000000, 12.500000
LINTOL/0.050000
SPINDL/2200.000, RPM, CLW
FEDRAT/460.000000, MMPM
RAPID
GOTO/-17.482028, -26.300947, 175.888264
RAPID
GOTO/-17.482028, -26.300947, 90.500001
FEDRAT/460.000000, MMPM
COOLNT/FLOOD
GOTO/-11.799326, -28.654799, 87.500001
GOTO/-11.764249, -28.654799, 86.205800
GOTO/-11.653224, -28.654799, 84.876054
GOTO/-11.460460, -28.654799, 83.515977
GOTO/-11.179711, -28.654799, 82.131884
GOTO/-10.806236, -28.654799, 80.730904
```

description
feeds and speeds
tool definition

Cutter path data

COMPUTER AIDED PART PROGRAMMING

CNC Coding ?

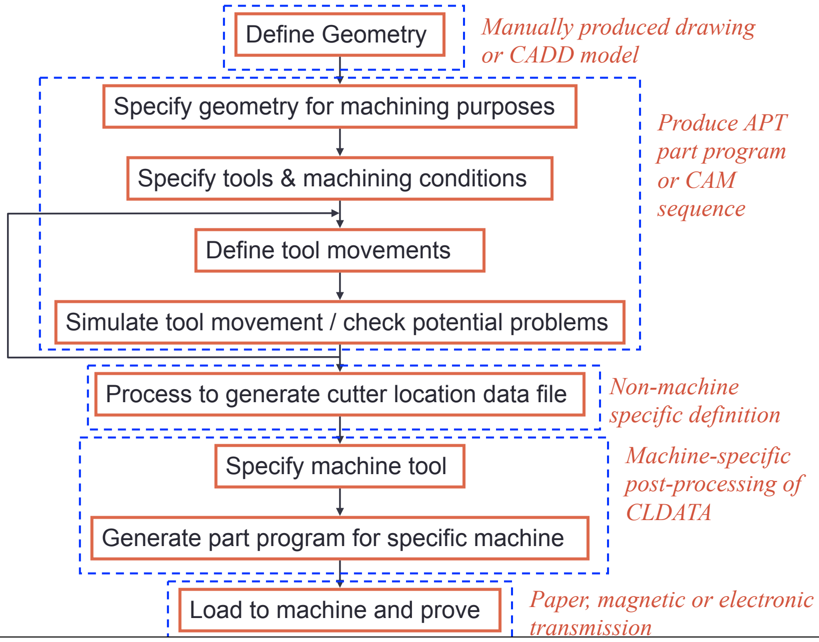


Computer Aided Part Programming

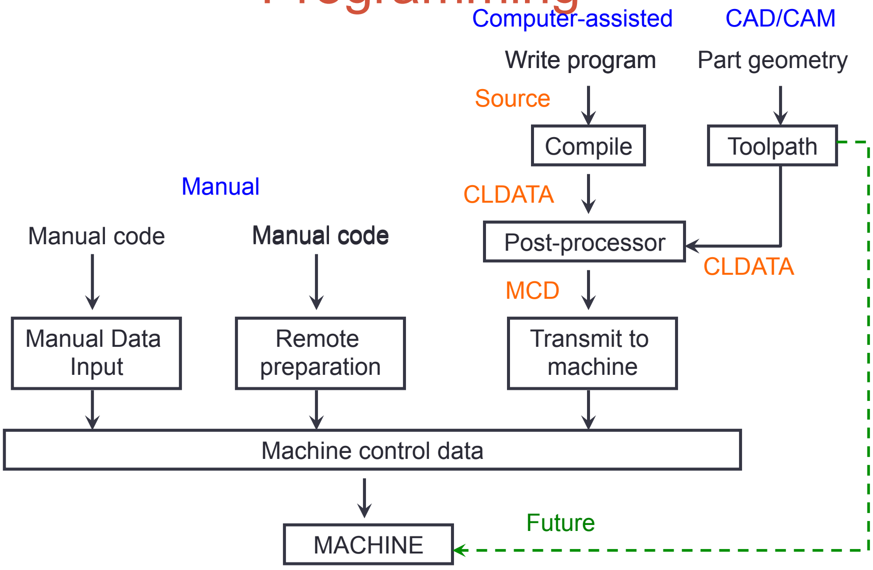
Programmes are increasingly created with

- Direct CADCAM link
- User friendly Computer Aided Part Programming packages
- **CAPP packages commonly employ**
  - simplified language forms
  - Graphical techniques, which provide
    - part programming
    - dynamic tool-path simulations (assists in debugging before any actual cutting is done)
- Production data such as cycle times
  - Combination of both
  - Provide additional facilities such as complex trigonometrical calculations
- **CAPP Format**
  - Depends upon company which writes software package
  - Most packages are based on APT (Automatically Programmed Tools) system
  - APT is compatible for a range of computers and CNC machines

Computer Aided Part Programming



Alternative Routes for Part Programming



CAPP Stages

Geometric Definition → **Manufacturing Processor** → **Post Processor**

Geometric Definition

- Identification of part geometry
- Breaking component shape into its primitive geometric elements
- In APT system primitives include points, lines, circles, planes, cylinders, cones and spheres

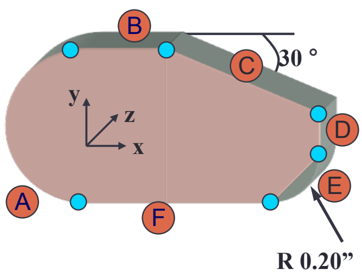
Manufacturing Processor

- Uses geometric definition to generate the date required to produce the component
- Includes tool selection, feeds and speeds
- Cutter path specification
  - interpolation
  - tool offset calculations
  - tool interference (gouge avoidance)

Post Processor

- Generation of NC instructions
- Conversion of APT to CNC machine code i.e. G & M-Code

Proprietary NC Codes




```
001 START INS 01        <- Start program in inches
001 TD = 0.125          <- Set tool diameter
002 FR   XY=10          } <- Set feed rates
003 FR   Z =4           }
004 SETUP > zcxyu       <- Set absolute zero position
005 GO   Y -.625        } <- Move to start position
006 GO   Z -.125        }
007 GR   a -180           (A)
008 ZERO AT             }
009        X.634        }
010        Y.5          }  (B)
011 GO   r.125          }
012        a 90         }
013 GR   a -30          }
014 > REF COODS         }
015 ZERO AT             }
016        X 1.50       }  (C)
017        Y 0          }
018 GO   r.125          }
019        a 60         }
020 GR   a -60          }
021 > REF COODS         }
022 ZERO AT             }
023        X 1.5        }
024        Y -0.3       }  (D)
025 GO    r.125         }
026        a 0          }
027 GR    a -90           (E)
028 GR    X -1.3          (F)
029 END                 <- End program
```

What Is Parametric Programming?

Parametric programming can be compared to any computer programming language like BASIC, C Language, and PASCAL.

However, this programming language can be accessed at G code level.

Combine manual programming techniques with parametric programming techniques.

Example of Parametric Programming

It will machine a mill a hole of any size at any location. Notice how similar this program is to a program written in BASIC.

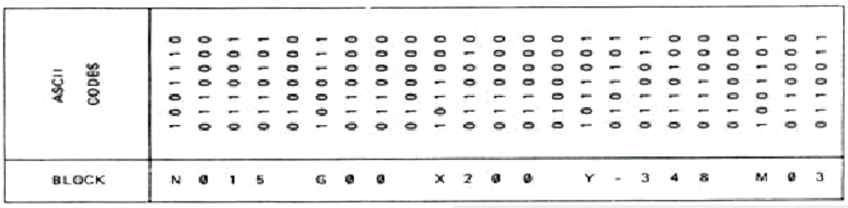
Program

```
O0001 (Program number)
#100=1. (Diameter of end mill)
#101=3.0 (X position of hole)
#102=1.5 (Y position of hole)
#103=.5 (Depth of counterbored hole)
#104=400 (Speed in RPM)
#105=3.5 (Feedrate in IPM)
#106=3. (Tool length offset number)
#107=2.0 (Diameter of counterbored hole)
G90 G54 S#104 M03 (Select abs mode, coordinate system, start spindle)
G00 X#101 Y#102 (Rapid to hole center)
G43 H#106 Z.1 (Instate tool length compensation, rapid to approach Z position)
G01 Z-#103 F[#105 / 2]
Y[#102 + #107 / 2 - #100 / 2] F#105
G02 J-[#107 / 2 - #100 / 2]
G01 Y#102
G00 Z.1
M30
```

Data Preparation for NC

- **Coding System**
  - **ASCII Code (American Standard Code for Information Exchange)**
    - Principle coding system for NC and CNC applications
    - Uses 7-bit binary numbers to represent all alpha-numeric data
    - Recommended by ISO
    - On paper tape, 1-represents hole, 0-represents no hole
  - **EIA Code (Electronic Industries Association)**
    - This code was popular before The present ISO code
    - Also based on 7-bit code
    - Adopted by American Standards Association

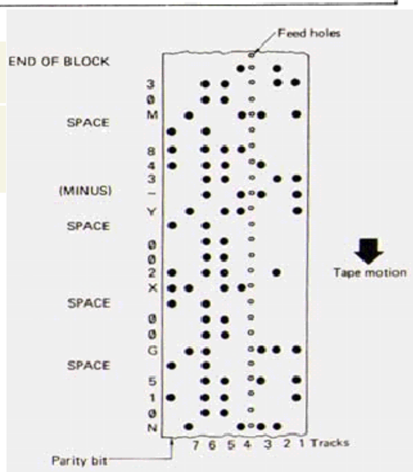
(Most Modern NC-CNC machines accommodate either of these two)



Typical Part-Program Instruction Block

Punched tape representation

Holes shown in Black



Example:
N015 G00 X200 Y-348 M03

Binary codes for
1 – 0110001
2 – 0110010
3 – 0110011
A – 1000001
B – 1000010 etc.

MOLD TOOL DESIGN

Mould Tool Design

Design in injection mould tool has always been an expensive process because of the need to build a prototype.

It is used for
- ❖ reducing the development time
- ❖ reducing the number of prototype modification

The functionality of a typical system

Initial mould design to generate mould parts and position split lines, gates and risers.

Mould fill simulation to evaluate flow uniformity throughout the mould and fill time.

Mould cooling simulation is based on finite element thermal calculations to evaluate heat transfer from the plastic throuugh the mould to the coolant.

RAPID PROTOTYPING

Rapid Prototyping

Rapid prototyping has the ability to convert a computer-generated model into a prototype model or final component more quickly and at a much lower cost than conventional production methods.

The fabrication of a physical, three dimensional part of arbitrary shape directly from a numerical description - typically a Computer Aided Design model, by a quick, highly automated and totally flexible manufacturing process.

Rapid Prototyping

- Several rapid prototyping technologies are available through the CAE team.
- CAD models can be directly converted from the computer screen to physical models in a variety of materials including plastic, investment casting wax, and wood.

Rapid Prototyping

The automatic construction of physical objects using additive manufacturing technology.

Prototyping technologies	Base materials
Selective laser sintering (SLS)	Thermoplastics, metals powders
Direct metal laser sintering (DMLS)	Almost any alloy metal
Fused deposition modeling (FDM)	Thermoplastics, eutectic metals
Stereolithography (SLA)	photopolymer
Laminated object manufacturing (LOM)	Paper
Electron beam melting (EBM)	Titanium alloys
3D printing (3DP)	Various materials

Rapid Prototyping

- **Laminated Object Modeling**

A laser beam is used to cut the shape of each layer from adhesive paper. These are then bonded together to build up a solid model.

- **Fused Deposition Modeling**

A tiny nozzle ejects pressurised molten plastic into the shape of the solid model, solidifying as expands and cools.

- **Three-dimensional Printing**

A 'binder' is directed onto either ceramic or metal powder which is laid down in thin layers by a roller. The 'binder' is cured at a temperature of around 150ºC.

Benefits of Rapid Prototyping

Some of the benefits and disadvantages of Rapid Prototyping over traditional production techniques include:

- **Time and Money Savings in Development**
- **Time and Money Savings in Production**
- **Poor Mechanical Properties**
- **Functional Testing**

Stereolithography

- the first Rapid Prototyping process.
- Charles Hull conceived the idea in 1982. Extensive research and production of prototype machines followed and he was granted a patent in 1986.

Charles W. Hull

Stereolithography, photochemical machining, laser sintering, and laminated-object manufacturing use 3D CAD data to produce models in hours.

Stereolithography

- Stereolithography, photochemical machining, laser sintering, and laminated-object manufacturing use 3D CAD data to produce models in hours.
- Most of these processes make parts from plastic.
- Models can be built from layers of liquid plastic, fused from plastic powders, or cut from partially cured polymer.

Stereolithography

Principles

Stereolithography uses the principle of photo-polymerisation, whereby a liquid plastic monomer (resin) is converted into a solid polymer by exposure to

Ultra-Violet (UV) light. The UV light is provided by a laser, which, by varying its power output can control the thickness of each layer.

Stereolithography

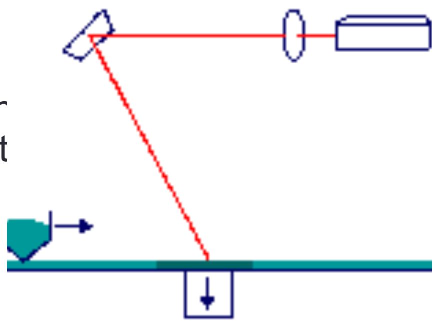
Preparation

The initial CAD model can be produced in a number of 3D CAD packages, but is converted into a file format known as STL via an 'STL Translator'. This translator slices the CAD model into layers, recording the coordinates of all boundaries.

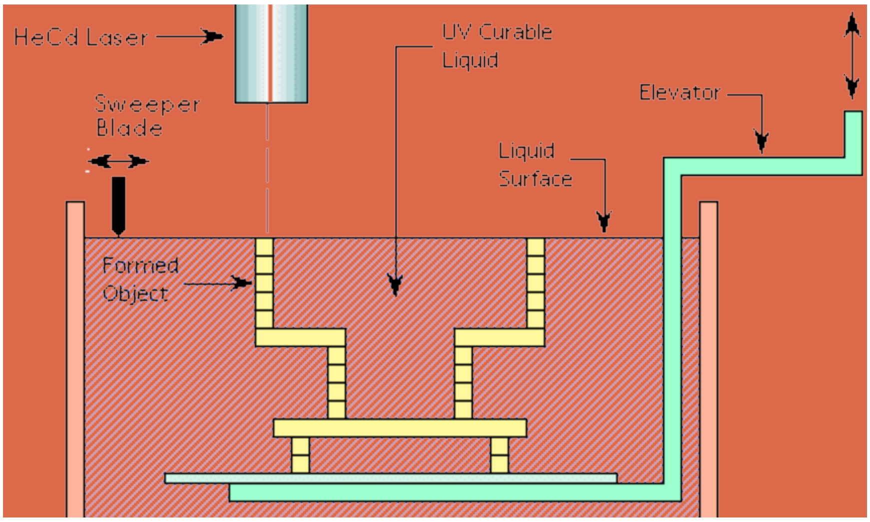
The STL also adds compensation factors for polymer shrinkage (the conversion from a monomer to a polymer creates about 3% shrinkage, so the layers are expanded by 3% in all axes).

Stereolithography

- The laser only performs about 60-80% cure.
- After the model has beer produced it is subjected t a further 2-3 hours (depending on size) low intensity UV light to cure fully.
- This causes a further 2% shrinkage, which is allowed for by the STL translator.



The Stereolithography Process



Selective Laser Sintering

Carl Deckard (left) and Joseph Beaman (right) invent the printing process SLS (Selective Laser Sintering) in 1987. It is incredible to think that this technology has been around for over 30 years and has only just entered the wider consumer market. However, today it's been said to be the Next Industrial Revolution, a revolution after all.

Selective Laser Sintering

Process

- A laser is used to fuse together tiny particles of wax, metal or plastic, which accumulates to form the solid model.
- Selective Laser Sintering (SLS) is the most diverse of all Rapid Prototyping techniques as it can be used to produce objects in plastics, metals, ceramics and sand.

Selective Laser Sintering

Benefits

SLS has the ability to produce objects with geometry or complexity that were previously impossible to form by any other manufacturing process. These include patterns for machine tools with internal cooling channels.

CMM INSPECTION

Coordinate Measuring Machine

Coordinate measuring machines offer one of the most efficient ways of measuring and capturing dimensional data because they can replace numerous surface plate tools and expensive fixed gages, and reduce complex measurement tasks from hours to minutes.

Coordinate Measuring Machines

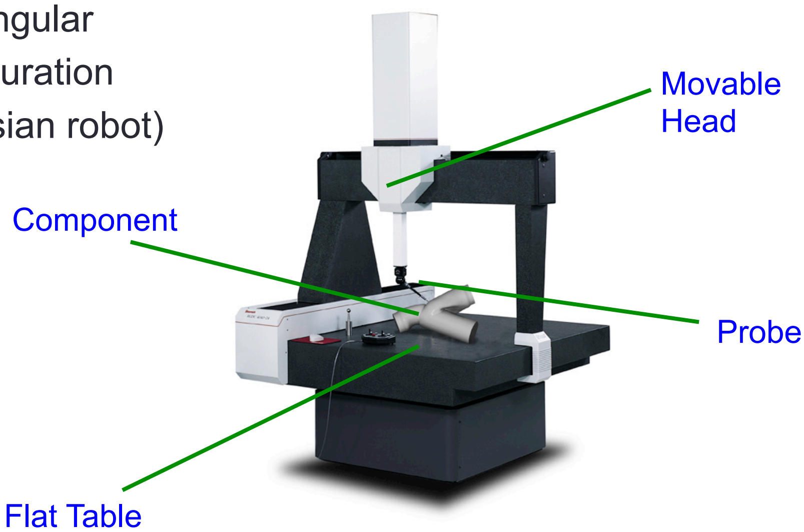
CMM is a computer controlled device which can be programmed to inspect

- the dimensions
- positions
- form of features

of a machined component.

Coordinate Measuring Machines

Rectangular Configuration (cartesian robot)



Coordinate Measuring Machines

Flat table : place to mounte the components

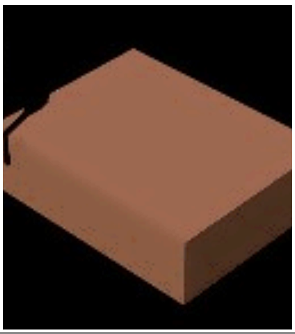
Movable head : holding a sensing probe

CMM can be programmed to touch a number of points (5 or more) along its length and a circular / cylindrical feature.

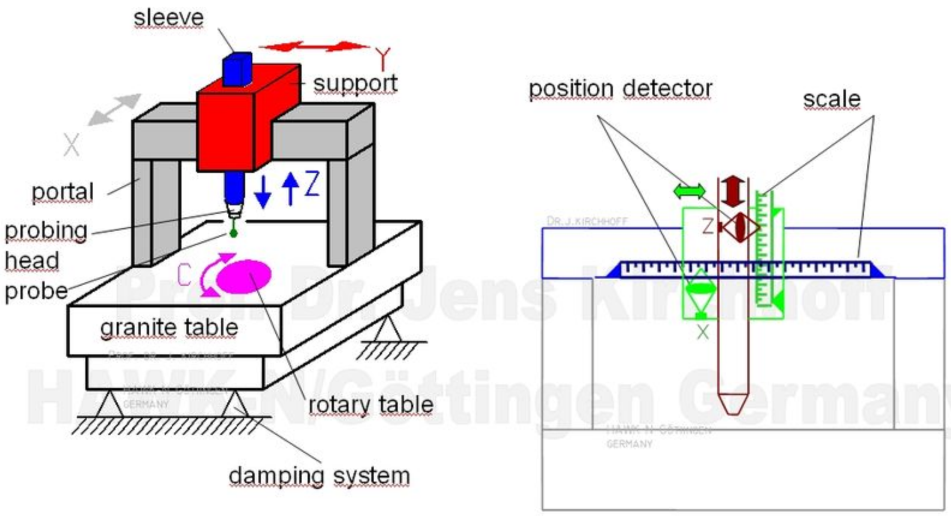
Coordinate Measuring Machine

- ✓allow to quickly and economically inspect large quantities of a given part or configuration.
- ✓inspect parts at precisely the same location on each part,
- ✓process variations be accurately measured.
- ✓statistical analysis can be applied to determine the control limits of any dimensional characteristics.

✓evaluate dimensional data
✓provide the operator with meaningful information concerning the condition of the manufacturing process

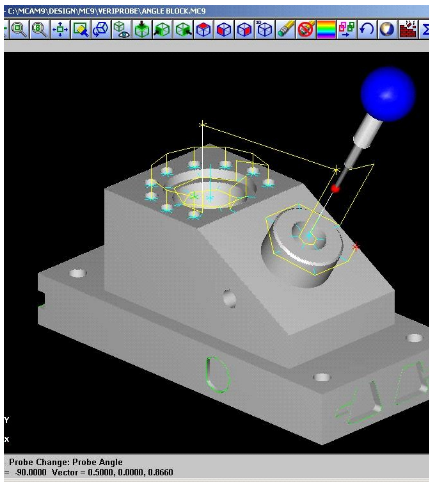


Coordinate Measuring Machine



Types of the CMM

- Gantry Type
- Cantilever Type
- Bridge Type
- Column Type



Selecting the Right Measurement Software

Data transfer standards have been developed and accepted by industry, such as Dimensional Measurement Interface Specification (DMIS) for communication from CMMs to CAD systems and Initial Graphics Exchange Specification (IGES) for CAD-to-CAD data exchange.

Selecting the Right Measurement Software

Such capability makes it possible to use CMMs in applications such as reverse engineering, where specifications are derived from the measurement of a model or broken part.