Lecture Notesb

In

Computer Numerical Control

Prepared by: Dr.Ibrahim Abed El-Maksoud Nasr

Mechanical Department , Higher Technological Institute

10th of Ramadan City

Contents

Chapter.1 : <u>Basic Terms used in CNC Machines</u>

The following terms are used in different types of machines such as conventional , NC machines and CNC machines

<u>1.Axis Relationships:</u>

When points are located on a work piece, two straight intersecting lines, one vertical and one horizontal, are used. These lines must be at right angles to each other, and the point where they cross is called the origin, or zero point (Fig. 1)
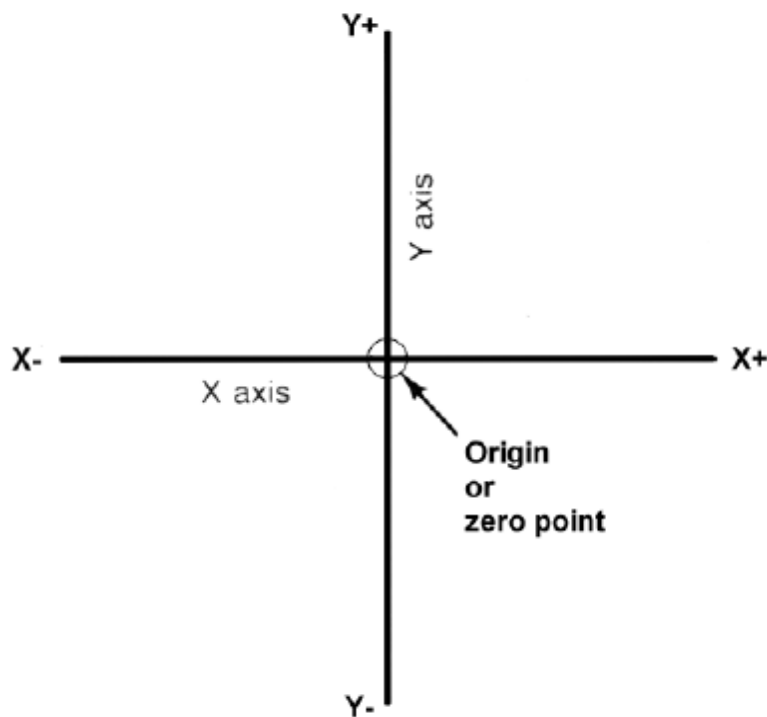
Fig. 1 Intersecting lines form right angles and establish the zero point (Allen-Bradley)
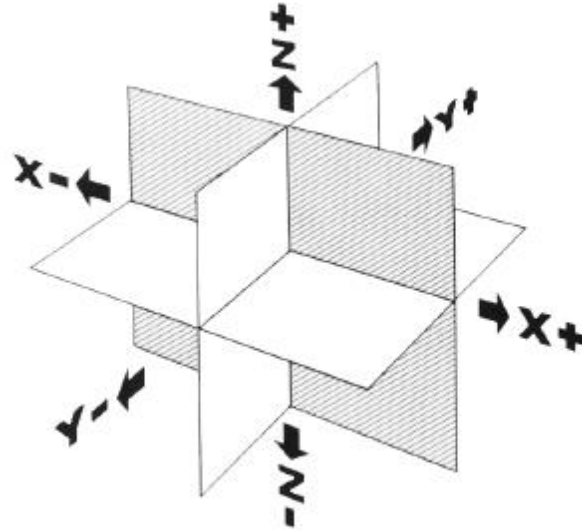
Fig. 2 The three-dimensional coordinate planes (axes) used in CNC. (The Superior Electric Company)

The three-dimensional coordinate planes are shown in Fig.2. The X and Y planes(axes) are horizontal and represent horizontal machine table motions. The Z plane or axis represents the vertical tool motion .The plus(+) and minus(-) signs indicate the direction from zero point (origin) along the axis motion .

1.1 In Milling Machines:

The milling machine has always been one of the most versatile machine tools used in industry Fig3 . Operations such as milling , contouring gear cutting , drilling , boring and reaming are only a few of the many operations which can be performed on milling machine. The milling machine can programmed on three axes:

1.The X axis controls the table movement left or right.

2.The Y axis controls the table movement towards or away from the column.

3.The Z axis controls the vertical (up or down) movement of spindle.

A typical 3- axis machines uses three controlled axes of motion . they are defined as the X axis , the Y axis , and the Z axis . the X axis is parallel to the longest dimension of the machine table , the Y axis is parallel to the shortest dimension of the table and the Z axis is the spindle movement Fig.4 on the vertical machining center . But in horizontal milling machine the X axis is the table longitudinal direction , the Y axis is the saddle cross direction and the Z axis is the spindle direction as in Fig.5
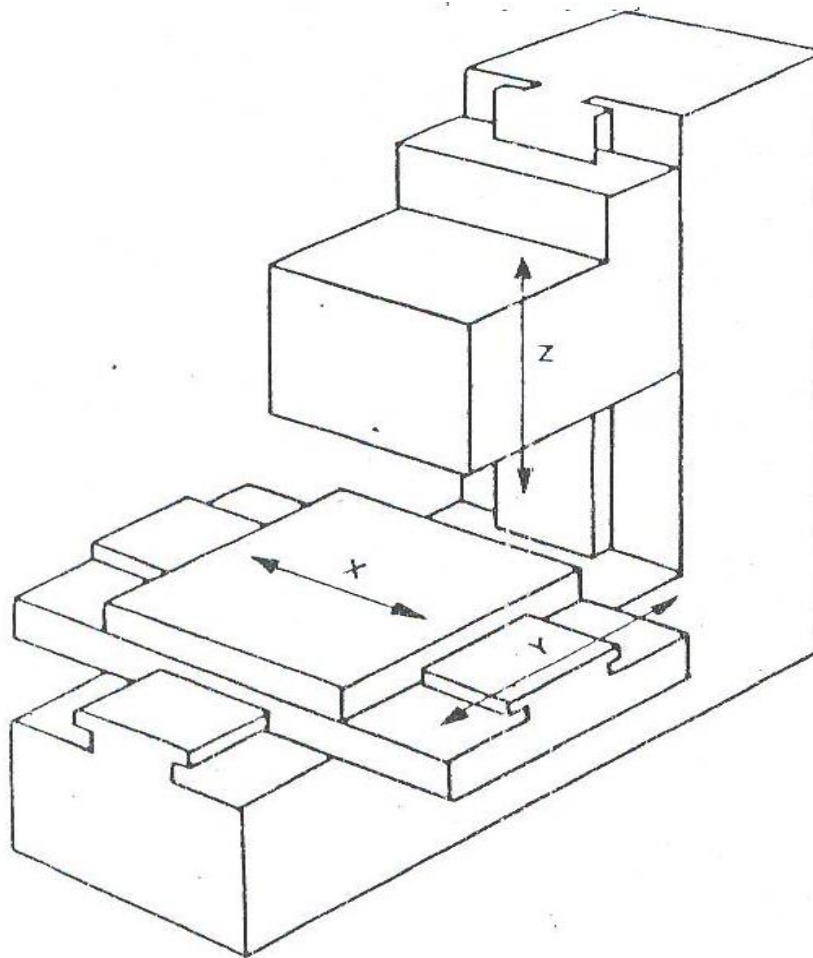
Fig.4 Vertical Milling Machine
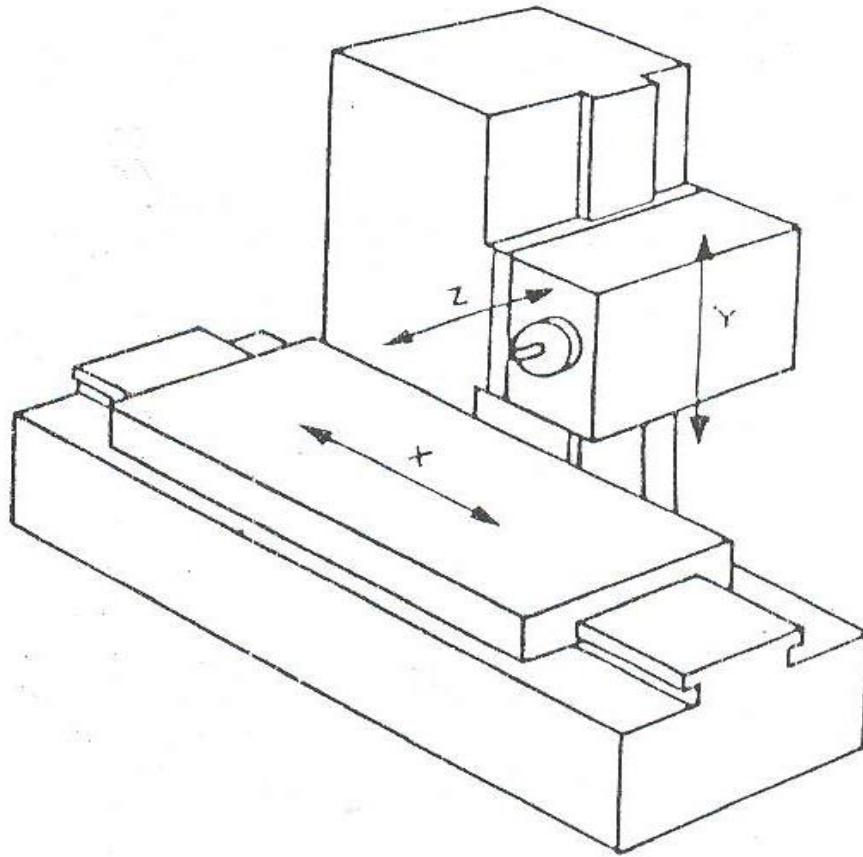
***Fig.5** Horizontal Milling Machine*

1.2 In Turning Machines

The engine lathe, one of the most productive machine tools, has always been an efficient means of producing round parts (Fig.6). Most lathes are programmed on two axes.

• The X axis controls the cross motion of the cutting tool. Negative X (X-) moves the tool towards the spindle centerline; positive X moves the tool away from the spindle centerline.

• The Z axis controls the carriage travel toward or away from the headstock.
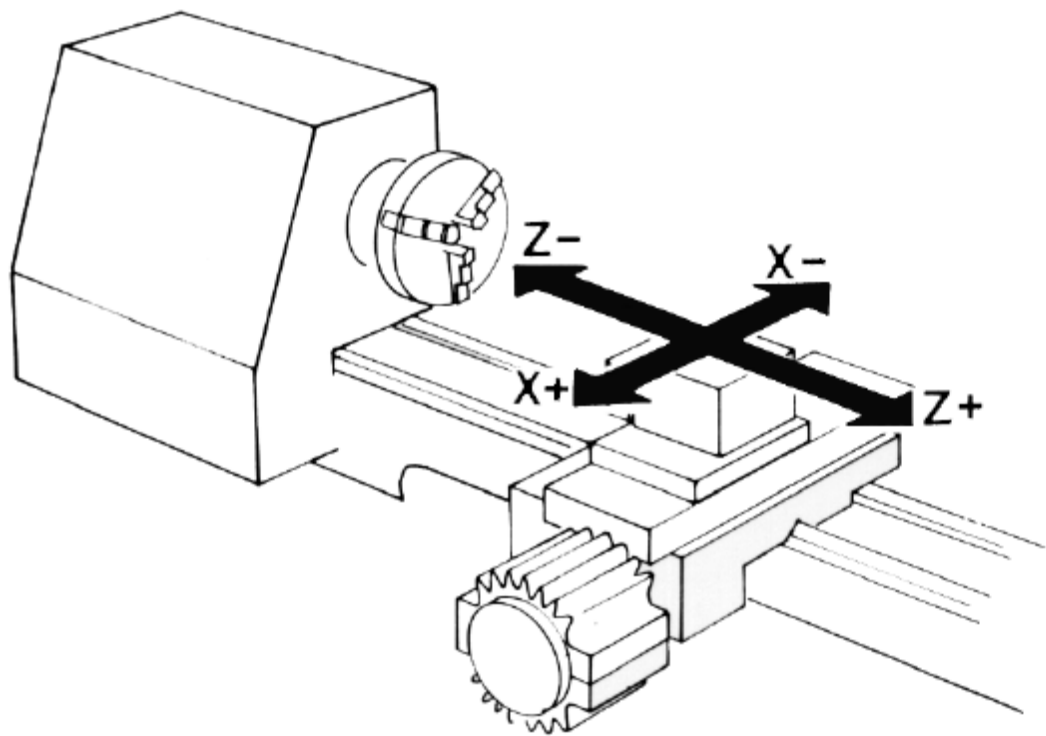
Fig.6 **The main axes of a lathe or Turning center**

2.Programming Systems

Two types of programming modes, the incremental system and the absolute system, are used for CNC. Both systems have applications in CNC programming, and no system is either right or wrong all the time. Most controls on machine tools today are capable of handling either incremental or absolute programming.

Positioning Reference Point Systems

Incremental   **Absolute**
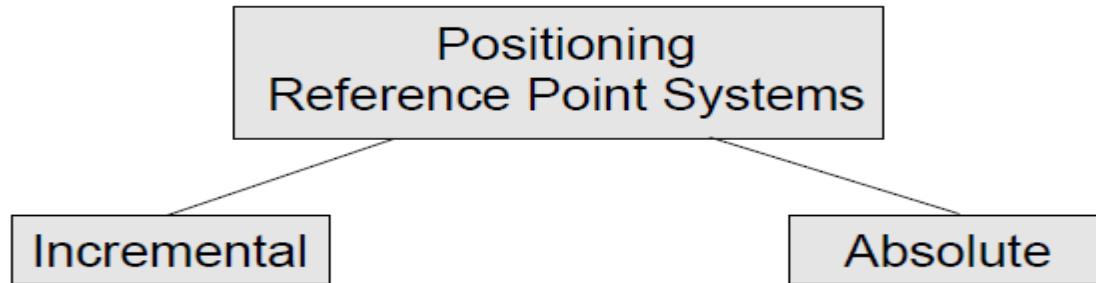
Incremental program locations are always given as the distance and direction from the immediately preceding point (Fig. 7). Command codes which tell the machine to move the table, spindle, and knee are explained here using a vertical milling machine as an example:
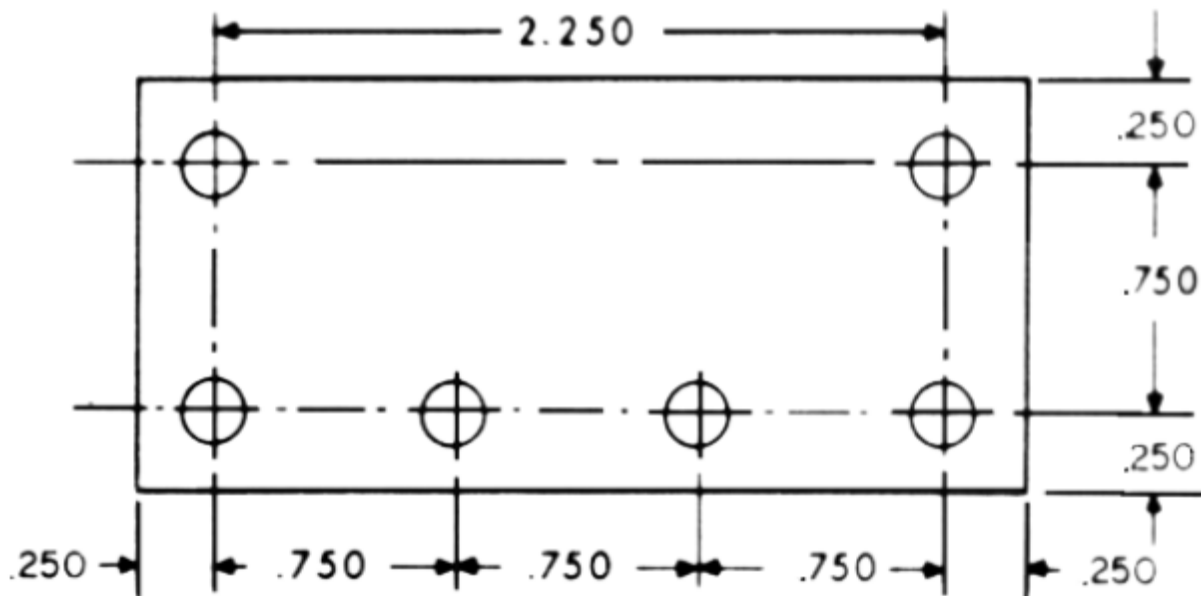
Fig.(7) A Work piece dimensioned in the incremental system mode

• A "X plus" (X+) command will cause the cutting tool to be
located to the right of the last point.

• A "X minus" (X-) command will cause the cutting tool to be located
to the left of the last point.

• A "Y plus" (Y+) command will cause the cutting tool to be located toward the
column.

• A "Y minus" (Y-) will cause the cutting tool to be located away
from the column.

• A "Z plus" (Z+) command will cause the cutting tool or spindle

to move up or away from the work piece.

• A "Z minus" (Z-) moves the cutting tool down or into the work piece

In incremental programming, the G91 command indicates to the computer and MCU (Machine Control Unit) that programming is in the incremental mode.

Absolute program locations are always given from a single fixed zero or origin point (Fig. 8). The zero or origin point may be a position on the machine table, such as the corner of the worktable or at any specific point on the workpiece. In absolute dimensioning and programming, each point or location on the workpiece is given as a certain distance from the zero or reference point.
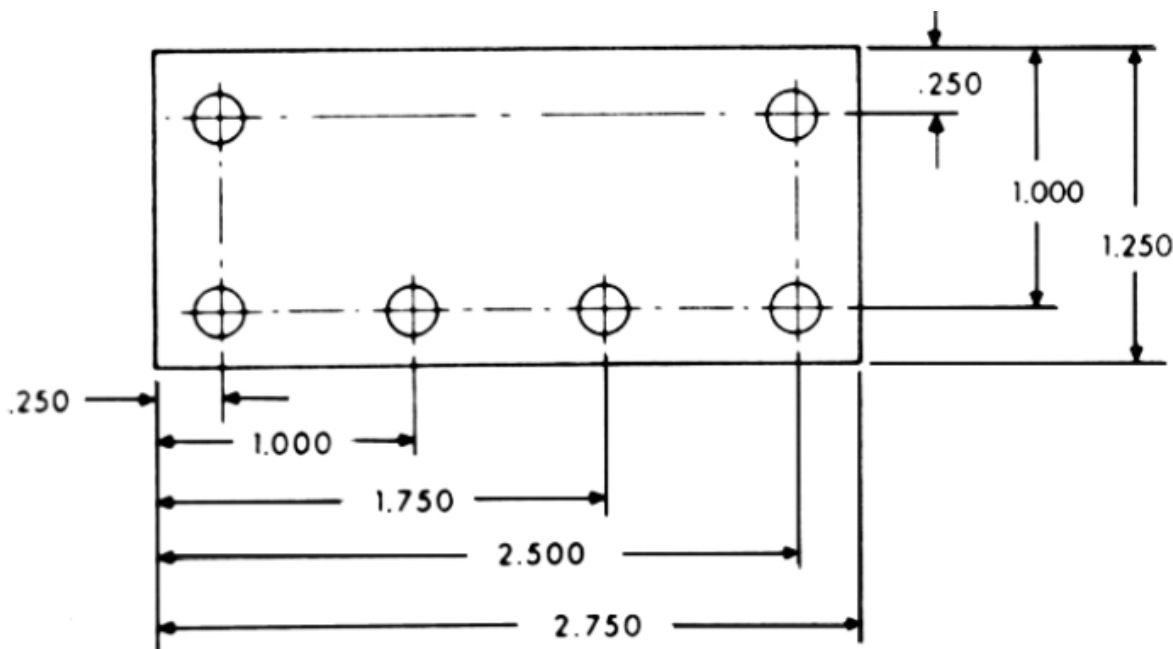
Fig.(8)A Work piece dimensioned in the absolute system mode

• A "X plus" (X+) command will cause the cutting tool to be located to the right of the zero or origin point.

• A "X minus" (X-) command will cause the cutting tool to be located to the left of the zero or origin point.

• A "Y plus" (Y+) command will cause the cutting tool to be

located toward the column.

• A "Y minus" (Y-) command will cause the cutting tool to be located away from the column.

In absolute programming, the G90 command indicates to the computer and MCU that the programming is in the absolute mode.

3.Types of Motion Control :

3.1 Point to Point Motion (PTP):
- Tool moves to the defined points with only feedback at the Specific point
- Used in drilling operations.
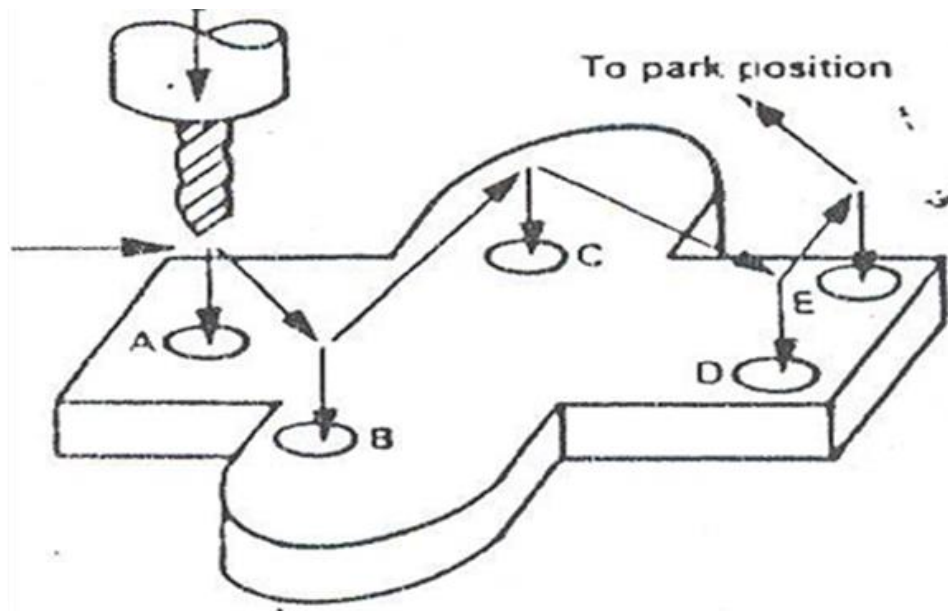
3.2. Contouring Motion:

- X,Y,Z motion are always controlled and they have constant feedback
- Used in milling or turning operations.
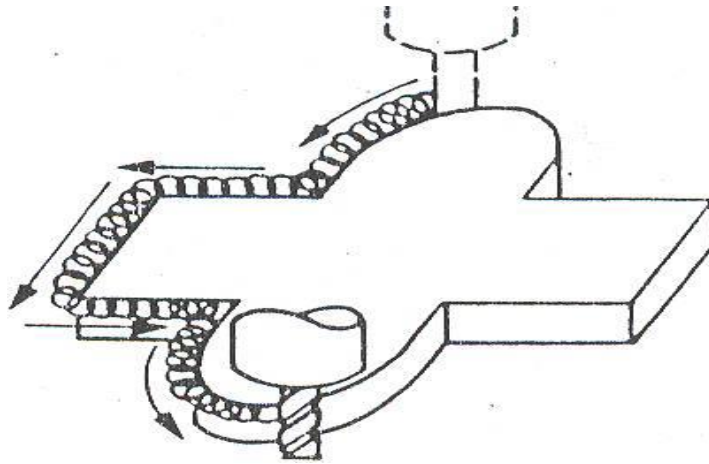
3.3. Straight Cut:

Only one coordinated is monitored at a time.
Used in simple straight cutting operations.
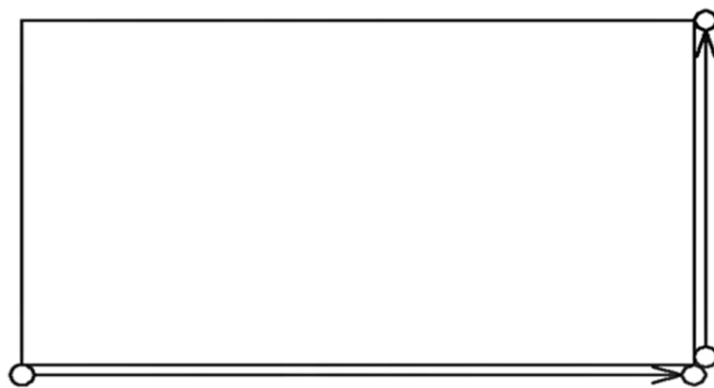
4. *Zero Shift System :*

They define the origin of the coordinate axis. They have three types:

4.1. *Fixed Zero (machine zero):*

It is where the machine manufacturers define the origin; normally at the corner of the table, (X & Y are always positive).
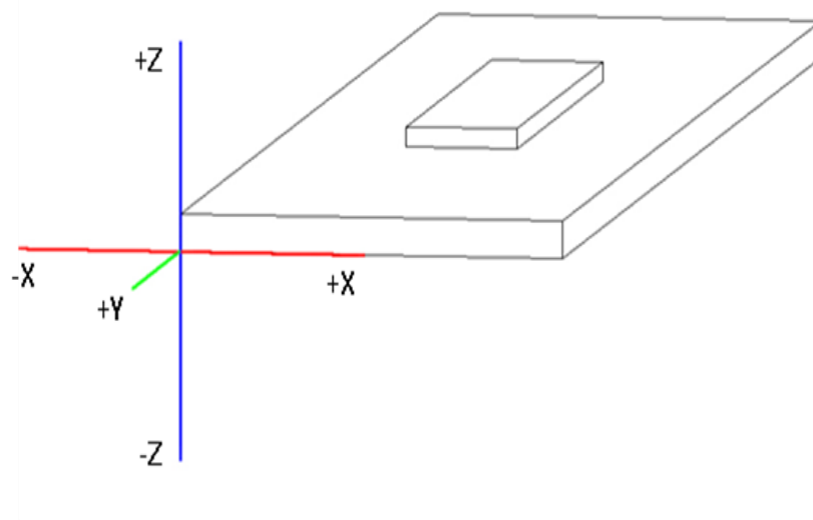
4.2. *Floating Zero (operator zero):* *The origin point is left for the operator to define using a special tool, which is lowered until it touches the desired point with no restrictions.(X & Y can take positive or negative values)*
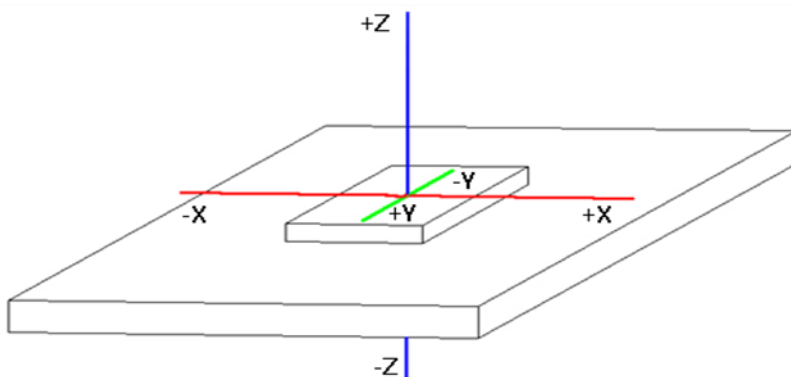
Revision Questions

1. Explain using graphs the axis relationships in milling, drilling and turning operations?

2. What are the positioning reference point systems?

3. Explain with sketch the different motion control systems?

4. With sketches, explain the different zeros in machining processes? (milling, drilling and turning operations)?

Chapter.2 : Computer Numerical Control Machines

[CNC Machines]

2.1 Flow Diagram of the Steps in Processing NC Program:
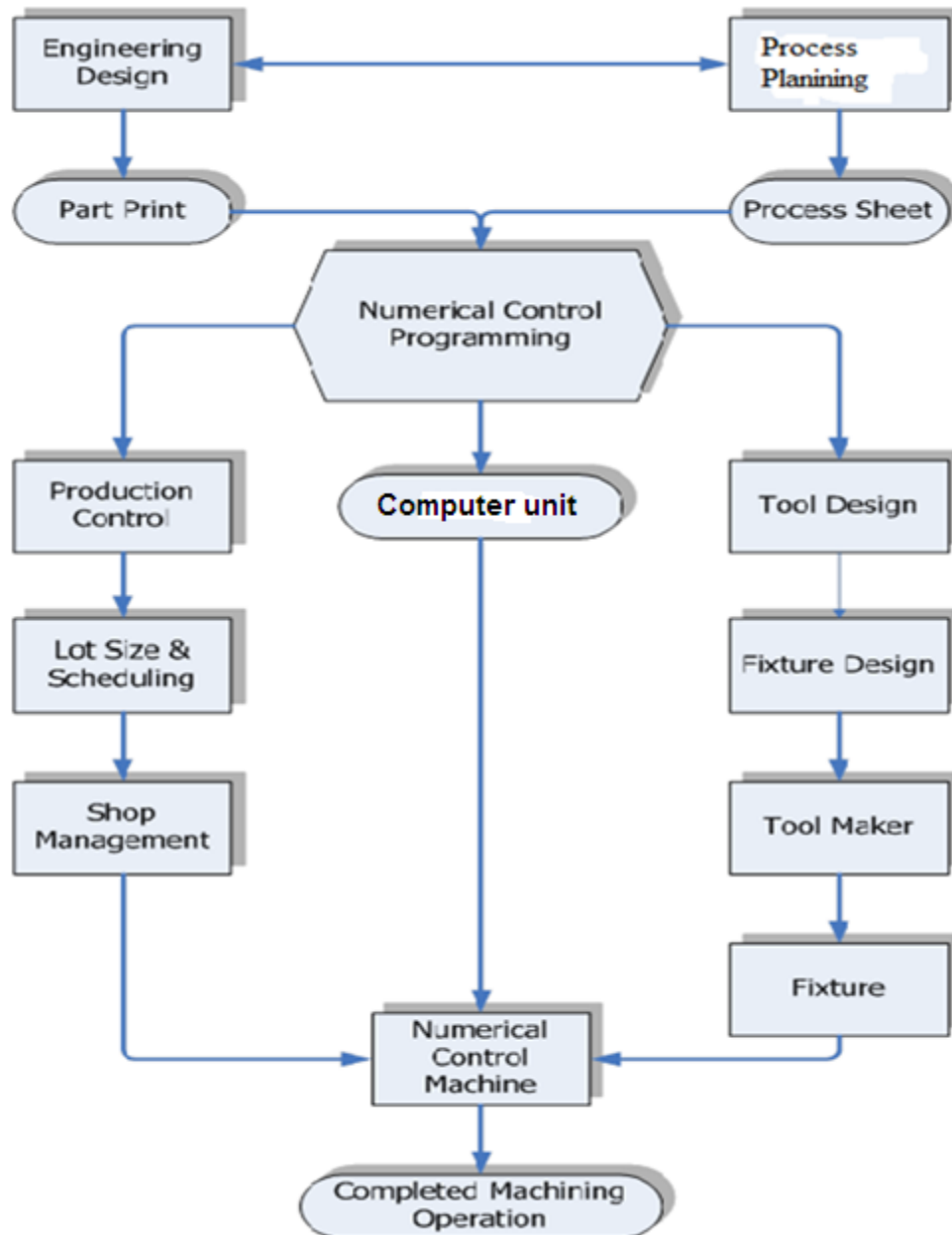
2.2 Features of CNC Machines:

1.   Storage of more than one part program.
2.   Various form of program input
•   Manual data input
•   Floppy diskette
•   Communication with external computer
3.   Program editing at the machine tool
4.   Fixed cycles & programming subroutines
5.   Interpolation
6.   Cutter length and size compensation
7.   Acceleration and deceleration calculation
8.   Communication interface
9.   Diagnostics

Program performs
interpolation
calculations
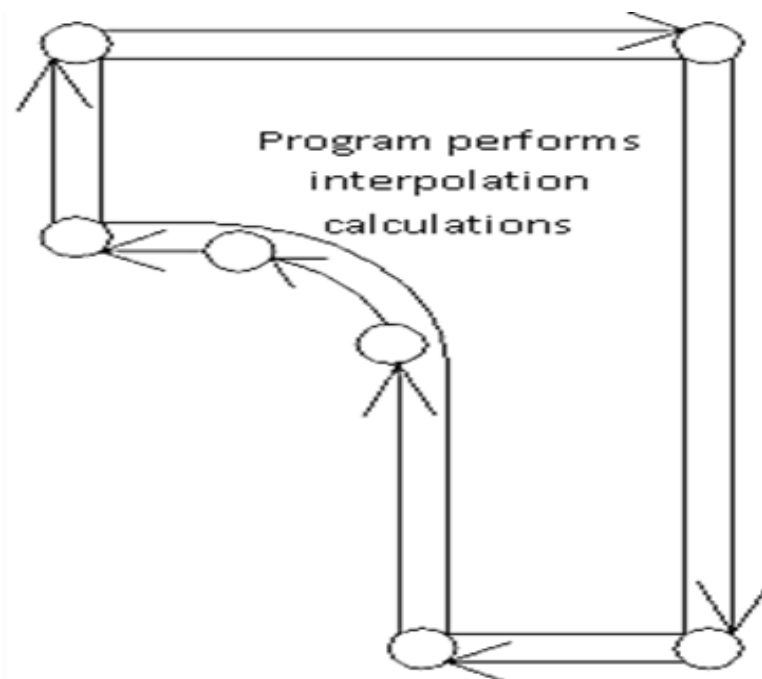
2.3. Machine Control Unit of CNC:

2.3.1 Hardware:

Memory	CPU	I/O Interface
1.Main Memory:	1.Control Section	1.Operator Panel (Keyboard, CRT,…)
.ROM: Operating Sys.	2.Arithmetic Logic Unit (ALU).	
RAM: Part Program		1. 2.Tape Reader

System Bus

Machine Tool Controls

• Position Control (X,Y,Z)
• Spindle Speed Control

Sequence Controls

• Coolant
• Fixture Clamping
• Tool Changer
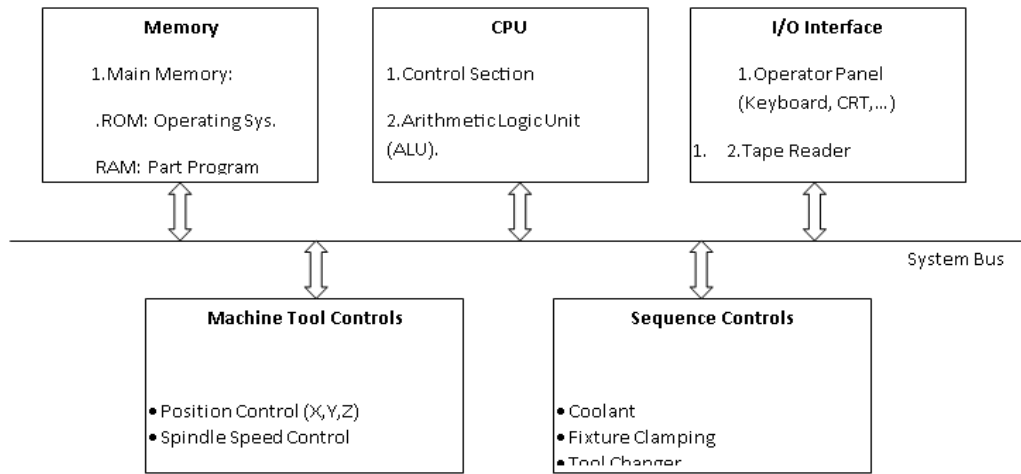
2.3.2 Software :   1. Operating system software: is to interpret the NC part program and generate the corresponding control signals to derive the machine tool axes ,it is installed by the controller manufacturer and is stored in ROM in the MCU. The operating system software consists of the following :• Editor: which permits the machine operator to input and edit NC part programs• Control program: which decodes the part program instructions, perform interpolation and acceleration/deceleration calculations ,and accomplishes other related functions to produce the coordinate control signals for each axis.• Executive program: which manage the execution of the CNC software as well as the I / O operations of the MCU.

2. Machine interface software: is used to operate the communication link between the CPU and the machine tool to accomplish the CNC auxiliary functions (coolant control ,tool changer ,fixture clamping device ,timers …),as previously indicated ,the I / O signals associated with the auxiliary functions are sometimes implemented by means of a programmable logic controller interfaced to the MCU ,and so the machine interface software is often written in the form of ladder logic diagram
3. Application software: consists of the NC part programs that are written for machining

2.4.Positioning system:

The NC positioning system converts the coordinate axis values in the NC part program into relative positions of the tool and work part during the processing .Let us consider the simple positioning system shown in figure .The system consists of A cutting tool and worktable on which a work part is fixed .The table is designed to move the part relative to the tool . The worktable moves linearly by means of a rotating lead screw ,which is driven by stepping motor or servomotor .For simplicity,

We show only one axis in our sketch .To provide x-y capability ,the lead screw has a certain pitch P (in/thread , mm/thread). Thus the table moves a distance equal to the pitch for one revolution. The velocity of the work table ,which corresponds to the feed rate in a machining operation is determined by the rotational speed of the lead screw
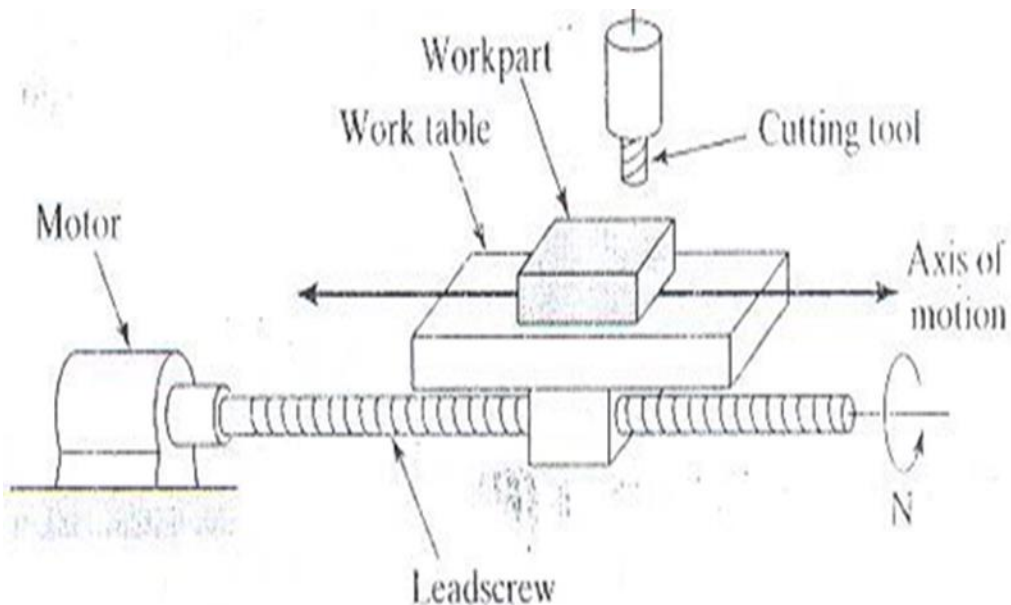
Motor and leadscrew arrangement in an NC position

There are two types of positioning systems used in NC system : shown in figure

a. open loop system operates without verifying that the actual position achieved in the move is the same as the desired position.

b. Closed loop control system uses feedback measurements to confirm that the final position of the worktable is the location specified in the program .

Open loop cost less than closed loop systems and are appropriate when the force resisting the actuating motion is minimal , closed loop systems are normally specified for machines that perform continuous path operations such as milling or turning in which there are significant forces resisting the forward motion of the cutting tool.
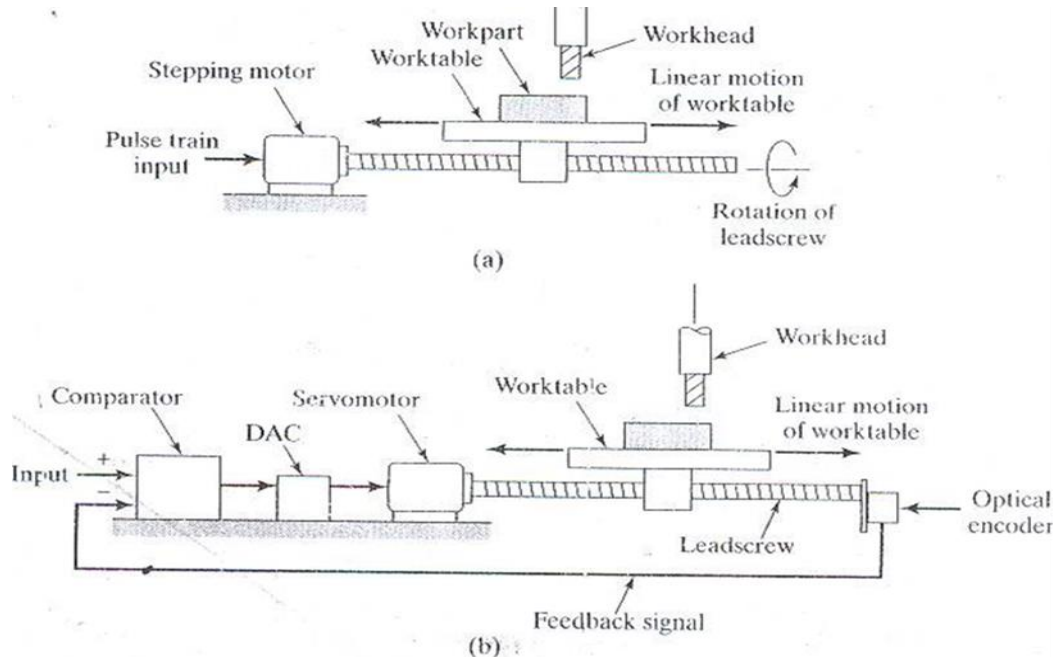
Figure Two types of motion control in NC: (a) open loop and (b) closed loop.

2.5 Applications of NC & CNC:

1..Machine tool application :

- NC lathe
- NC boring mill
- NC drill press
- NC milling machine
- NC cylindrical grinding

2. Non-machine tool application:

- Welding machining
- Thermal cutting machine

2.6 Part characteristics to be Suited to Applications of NC & CNC:

1. Batch production of small to medium lot size (1-100 products)
2. Repeated order
3. Complex part geometry
4. Much metal needs to be removed
5. Part is expensive

2.7 Advantages of using NC,CNC:

1. Greater accuracy
2. Lower scrap rates
3. More complex part geometry
4. Less floor space is required
5. Operator skill level is reduced
6. Inspection requirement is reduced

2.8 Disadvantages of using NC, CNC:

1. Higher investment cost
2. Higher maintenance effort

<u>Revision Questions:</u>

1. Draw and explain the flow diagram of CNC process ?
2. What are the main features of CNC machining processes ?
3. Explain with sketch the machine control unit of CNC machines
4. Explain with sketch the different positioning system ?
5. What are the different applications of NC , CNC machining?
6. What are the part characteristics to be suited to applications of NC and CNC applications ?
7. What are the advantages disadvantages   of using NC and CNC   machining?

Chapter.3: NC Coded Program :

3.1.Steps in program Planning :

The steps required in program planning are decided by the nature of the work . There is no useful formula for all the jobs , but some basic steps should be considered :

1. initial information /machine tool features
 2.part complexity
 3.manual programming / computerized programming
4. typical programming   procedure
5.part drawing / engineering data
6.Machinning sequence
7.part setup
8.technical decisions
9.work sketch and calculations
10.quility consideration in CNC programming

3.2 . Structure of Part Program :

There are four basic terms used in CNC programming , these words are the key to understand the general CNC terminology :

> Character ------- work -------- Block ---------- Program

each term is very common and important in CNC programming

3.2.a . Character

 A character is the smallest unit of CNC program ,it can have one of the three forms: Digits - Letters - Symbol
Characters are combined into words ,this combination is called alpha-numerical Program input.

3.2.b Word

A program word is a combination of   alpha-numerical characters , creating a single instruction to the control system . Normally each word begins with capital letter that is followed by a number representing a program code or actual value . typical words indicate the axis position , feed rate , speed , preparatory commands , miscellaneous functions and many other definitions.

3.2.C. Block

Just like the word is used as a single instruction to the CNC system ,the block is used as multiple instruction . A program entered into the control system consists of individual lines of instructions , sequenced in a logical order . each line called a sequence block is composed of one or several words and each word is composed of two or more characters.

In the control system , each block must be separated from all others . To separate blocks in manual data input (MDI) mode at the control , each block has to end with a special End – Of – Block code (symbol). this code is marked by EOB on the computer panel .When preparing the program on a computer , the Enter key on the keyboard will terminate the block .

3.2.d Program

The part program structure varies for different controls , but the logical approach does not change from one control to another . a CNC program usually begins with a program number or similar identification , followed by the blocks of instructions in a logical order . the program ends with stop code or a program termination symbol .

3.3 . _Format of Part Program :_

Since the early days of numerical control , three formats had become significant in their time . they are listed in the order of original introduction:

  1.Tap Sequential Format   NC only - no decimal point

  2.Fixed Format   NC only - no decimal point

  3. Word Address Format   NC or CNC - decimal point

only the very early control systems use the tab sequential or fixed formats. both of them disappeared in the early 1970 and now obsolete. they have replaced by the much more convenient word address format.

Characters

| 6 | F | . | - |

Words

| F 2 7 5 | . | 0 | G 0 1 | N 5 |

Block

| N 5 | G 0 1 | Y | - 6 | . | 4 8 | F 2 7 5 | . | 0 |
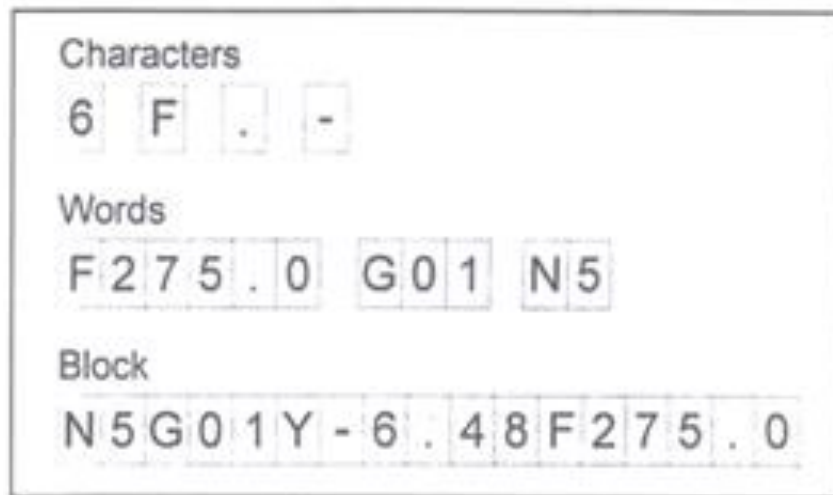
      Typical word address programming format

The address – the letter – in the block defines the meaning of the word and must always be written first . for example X5.75 is correct ,5.75X is incorrect , no spaces are allowed within a word but only allowed before the word .

Data - indicate the word numerical assignment . this value varies greatly and depend on the preceding address. it may represent :

a sequence number (N) , a preparatory command (G) ,a miscellaneous function (M) , a coordinate word (X,Y,Z) , feed rate function (F) , spindle function (S) , and tool function (T), etc and the above example of typical words that had explain the meaning in CNC program .

4. *Preparatory Commands* :

G Code	Description
G00	Rapid positioning
G01	Linear interpolation
G02	Circular interpolation close wise
G03	Circular interpolation counter clock wise
G04	Dwell (as a separate block)
G09	Exact stop check – one block only
G10	Programmable data input (data Setting)
G11	Data setting mode cancel
G15	Polar coordinate command cancel
G16	Polar coordinate command
G17	XY plane designation
G18	ZX plane designation
G19	YZ plane designation
G20	English units of input
G21	Metric units of input
G22	Store stroke check ON
G23	Store stroke check OFF
G25	Spindle speed fluctuation detection ON
G26	Spindle speed fluctuation detection ON
G27	Machine zero position check
G28	Machine zero return (reference point 1)
G29	Return from machine zero
G30	Machine zero return (reference point 2)
G31	Skip function
G40	Cutter radius compensation cancel
G41	Cutter radius compensation – left
G42	Cutter radius compensation – right
G43	Cutter radius compensation – positive
G44	Cutter radius compensation – negative
G45	Position compensation – single increase
G46	Position compensation – single decrease
G47	Position compensation – double increase
G48	Position compensation – double decrease
G49	Tool length offset cancel
G50	Scaling function cancel
G51	Scaling function

G Code	Description
G52	Local coordinate system setting
G53	Machine coordinate system
G54	Work coordinate offset 1
G55	Work coordinate offset 2
G56	Work coordinate offset 3
G57	Work coordinate offset 4
G58	Work coordinate offset 5
G59	Work coordinate offset 6
G60	Single direction position
G61	Exact stop mode
G62	Automatic corner override mode
G63	Tapping mode
G64	Cutting mode
G65	Custom macro call
G66	Custom macro modal call
G67	Custom macro modal call cancel
G68	Coordinate system rotation
G69	Coordinate system rotation cancel
G73	High speed peck drilling cycle (deep hole)
G74	Left hand threading cycle
G76	Fine boring cycle
G80	Fixed cycle cancel
G81	Drilling cycle
G82	Spot drilling cycle
G83	Peck- drilling cycle (deep hole drilling cycle)
G84	Right hand threading cycle
G85	Boring cycle
G86	Boring cycle
G87	Back Boring cycle
G88	Boring cycle
G89	Boring cycle
G90	Absolute dimensioning mode
G91	Incremental dimensioning mode
G92	Tool position register
G98	Return to initial level in fixed cycle
G99	Return to R level in fixed cycle

3.4.1. Examples of Preparatory Commands in Milling and Drilling:

G00 Rapid Traverse: When the tool being positioned at a point

preparatory to a cutting motion , to save time it is moved along straight line at rapid traverse , at a fixed traverse rate which is pre-programmed into the machine control system : Format N_ _G00 __ X_Y_Z_
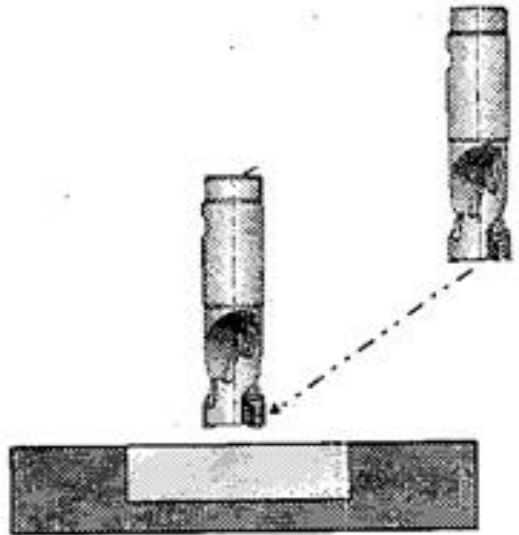
G01 Linear Interpolation(feed traverse) :

The tool moves along a straight line in one or two axis simultaneously at a programmed linear speed , feed rate

Format N_ _G01 __ X_Y_Z__ F__
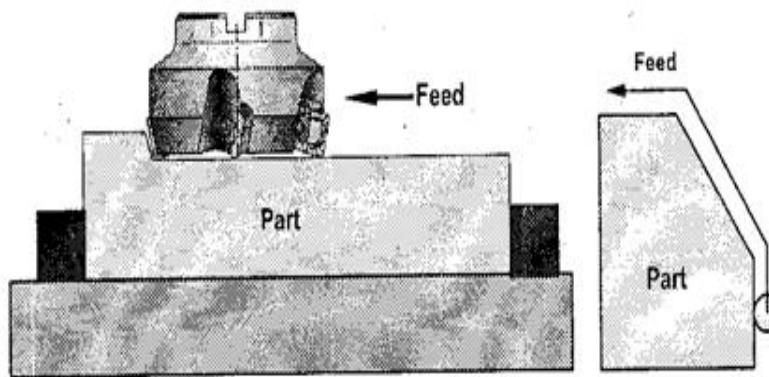
G02/03 Circular Interpolation :
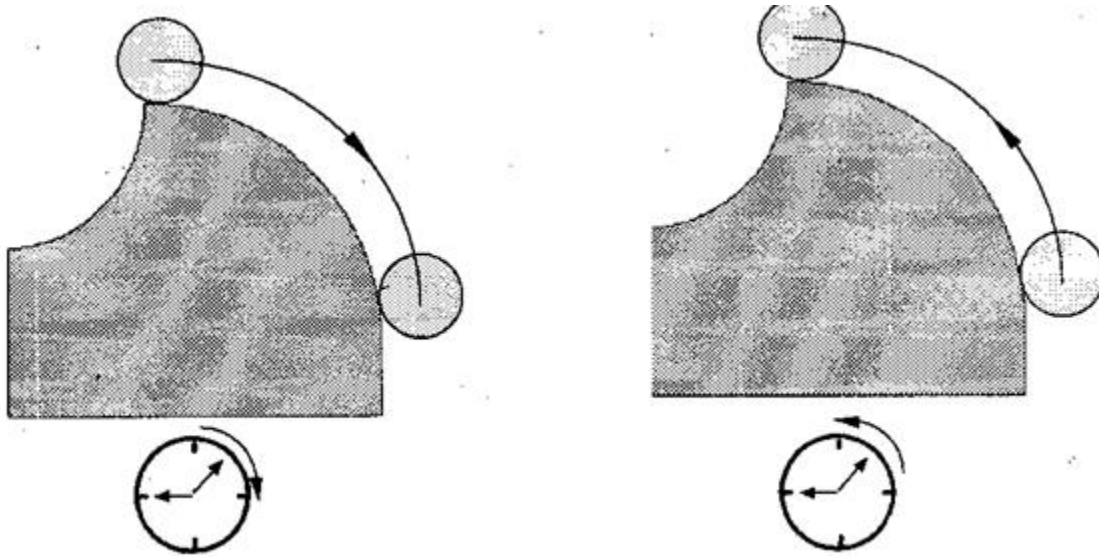
Format   **N_ _G02/03   X_Y_Z _ I_J_K_F_**   _using arc center_

   _or_   **N_ _G02/03   X_Y_Z_R_F_**   _using arc radius_

G02 moves along a CW arc , and   G03   moves along a CCW arc

Arc Center

The arc center is specified by address I,J,K are the X,Y,Z coordinates of the center with reference to the arc start point.
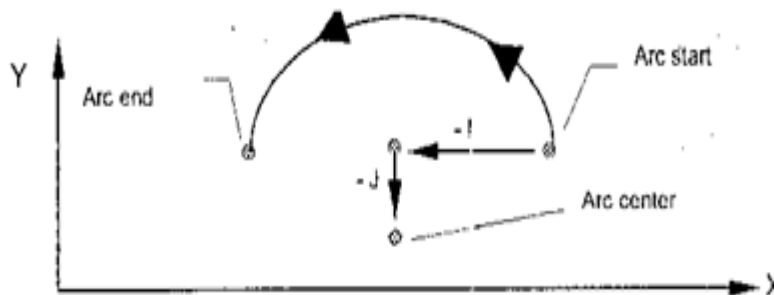
Arc Radius

The radius is specified by Address   R
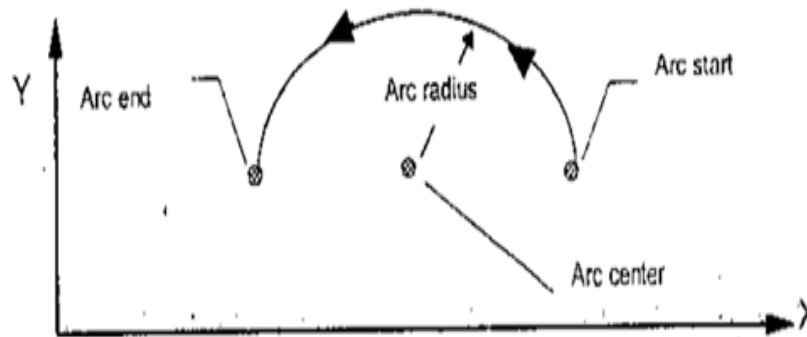
Block Format   **N__G02 X_Y_Z_ R_F_**

                           N__G03 X_Y_Z_R_F_

3.4.2 Preparatory function for Canned Cycle   in milling and drilling:
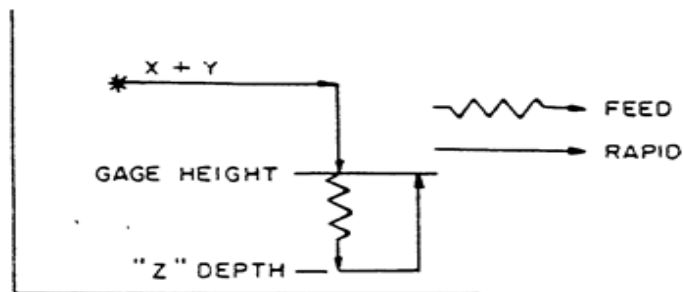
FIGURE
G81 — Drill cycle

Drill Cycle — G81. Figure   illustrates the G81 drill cycle. When a G81
cycle is programmed, the tool will:
    1) rapid in X and/or Y.
    2) rapid in the Z axis to *gage height.* (Gage height is the rapid dis-
        tance the tool advances prior to contacting the part surface or the
        rapid distance the tool retracts after completing the cycle.)
    3) feed in the Z axis to the Z depth.
    4) rapid retract to gage height.
These four steps will occur in the same order every time a G81 cycle is
called.

Dwell Cycle — G82. The G82 dwell cycle is illustrated in figure   . When G82 is programmed, the tool will:

    1) rapid in X and/or Y.
    2) rapid in the Z axis to gage height.
    3) feed in the Z axis to the Z depth.
    4) dwell for the amount of time selected (usually .1 to 6 seconds).
    5) rapid retract to gage height.

These five steps will occur in the same order every time a G82 cycle is called.

FIGURE
G82 — Dwell cycle
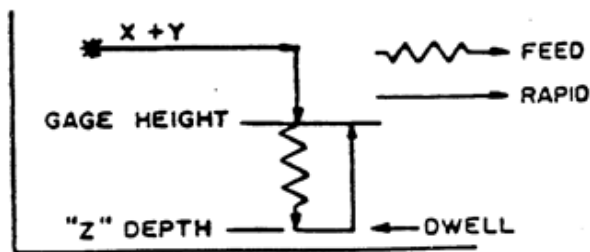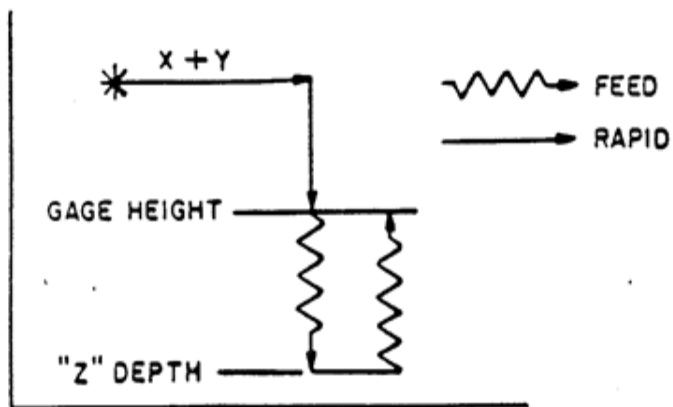
Bore Cycle — G85. Figure   shows the G85 basic bore cycle. When the G85 is programmed, the tool will:

    1) rapid in X and/or Y.
    2) rapid in the Z axis to gage height.
    3) feed in the Z axis to the Z depth.
    4) feed retract to gage height.

These four steps will occur in the same order every time a G85 cycle is programmed.

FIGURE
G85 — Bore cycle

Tap Cycle — G84. Figure   demonstrates the G84 tap cycle. When the G84 is programmed, the tool will:

1) rapid in X and/or Y.
2) rapid in the Z axis to gage height.

FEED
RAPID
SPINDLE DIRECTION

X + Y
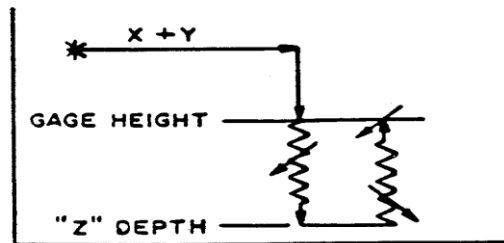
GAGE HEIGHT

"Z" DEPTH

FIGURE
G84 — Tap cycle

3) feed in the Z axis to the Z depth.
4) reverse spindle direction and feed retract to gage height.
5) reverse spindle direction again at gage height.

These steps will occur in the same order every time a G84 cycle is called.

X + Y

GAGE HEIGHT

"Z" DEPTH   FEED TO FOLLOWING
                      POSITIONS IF PROGRAMMED
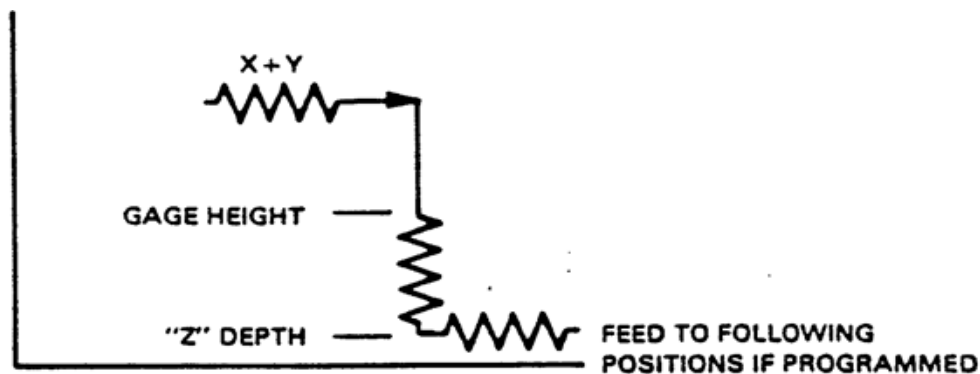
FIGURE
G79 — Basic mill cycle

Basic Mill Cycle — G79. The G79 basic milling cycle is shown in figure
When the G79 is programmed, the tool will:

1) feed in X and/or Y.
2) rapid in the Z axis to gage height.
3) feed in the Z axis to the Z depth.
4) feed to following positions.

These four steps will occur in the same order whenever a G79 is programmed.

Cancel Cycle — G80. Figure   shows the G80 cancel cycle. When a G80
is programmed, the tool will:

     1) rapid in X and/or Y.

     2) cancel any Z motion.

These steps will occur in the same order every time a G80 cycle is called.

FIGURE
G80 — Cancel cycle

RAPID
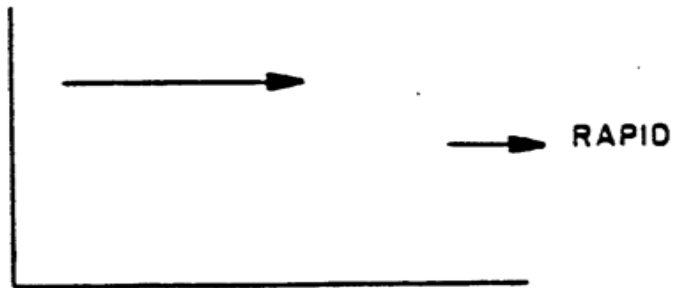
3.4.2.Examples of G-Codes used in Turning:

In any inconsistency between the listed codes in this handbook and the control system manual, the G codes listed by the control manufacturer must be selected.

APPLICATIONS FOR TURNING

Fanuc lathe controls use three G code group types - A, B and C. The *Type A* is the most common; in this handbook, all examples and explanations are Type A group, including the table below. Only one type can be set at a time. Types A and B can be set by a control system parameter, but type C is optional. Generally, most G codes are identical, only a few are different in the A and B types. More details on the subject of G code groups is listed at the end of this chapter.

G code	Description
G00	Rapid positioning
G01	Linear interpolation
G02	Circular interpolation clockwise
G03	Circular interpolation counterclockwise
G04	Dwell (as a separate block)
G09	Exact stop check - one block only
G10	Programmable data input (Data Setting)
G11	Data Setting mode cancel
G20	English units of input
G21	Metric units of input
G22	Stored stroke check ON
G23	Stored stroke check OFF
G25	Spindle speed fluctuation detection ON
G26	Spindle speed fluctuation detection OFF
G27	Machine zero position check
G28	Machine zero return (reference point 1)
G29	Return from machine zero
G30	Machine zero return (reference point 2)
G31	Skip function
G32	Threading - constant lead
G35	Circular threading CW
G36	Circular threading CCW
G40	Tool nose radius offset cancel
G41	Tool nose radius offset left
G42	Tool nose radius compensation right

| G code	Description	
G50	Tool position register / Maximum r/min preset	
G52	Local coordinate system setting	
G53	Machine coordinate system setting	
G54	Work coordinate offset 1	
G55	Work coordinate offset 2	
G56	Work coordinate offset 3	
G57	Work coordinate offset 4	
G58	Work coordinate offset 5	
G59	Work coordinate offset 6	
G61	Exact stop mode	
G62	Automatic corner override mode	
G64	Cutting mode	
G65	Custom macro call	
G66	Custom macro modal call	
G67	Custom macro modal call cancel	
G68	Mirror image for double turrets	
G69	Mirror image for double turrets cancel	
G70	Profile finishing cycle	
G71	Profile roughing cycle - Z axis direction	
G72	Profile roughing cycle - X axis direction	
G73	Pattern repetition cycle	
G74	Drilling cycle	
G75	Grooving cycle	
G76	Threading cycle	
G90	Cutting cycle A	(Group type A)
G90	Absolute command	(Group type B)
G91	Incremental command	(Group type B)
G92	Thread cutting cycle	(Group type A)
G92	Tool position register	(Group type B)
G94	Cutting cycle B	(Group type A)
G94	Feedrate per minute	(Group type B)
G95	Feedrate per revolution	(Group type B)
G96	Constant surface speed mode	(CSS)
G97	Direct r/min input	(CSS mode cancel)
G98	Feedrate per minute	(Group type A)
G99	Feedrate per revolution	(Group type A)

G00 rapid traverse

When the tool being positioned at a point preparatory to cutting motion ,to save time it is moved along a straight line at a rapid traverse , at a fixed traverse rate which is preprogrammed into the machine control system typical traverse rate are 50 m/min ,but can be as high as 80 m/min.

*Format   N__G00 X_Z_*
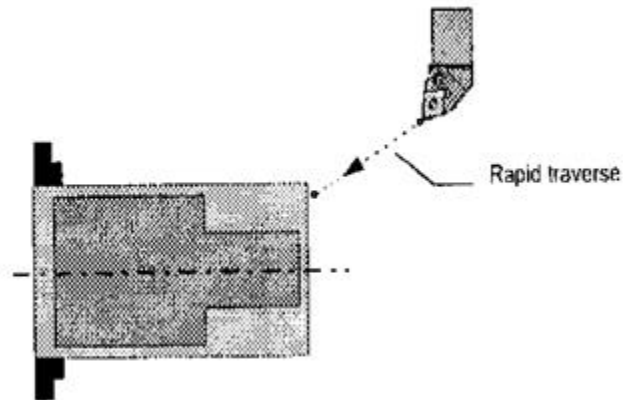
Rapid traverse

G01 Linear Interpolation (feed traverse)

The tool moves along a straight line in one or two axis simultaneously at a programmed linear speed ,the feed rate

*Format   N__ G01 X_Z_F_*

Feed motion
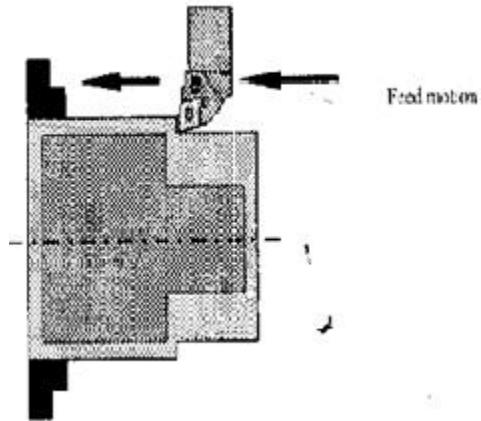
G02/03 Circular Interpolation
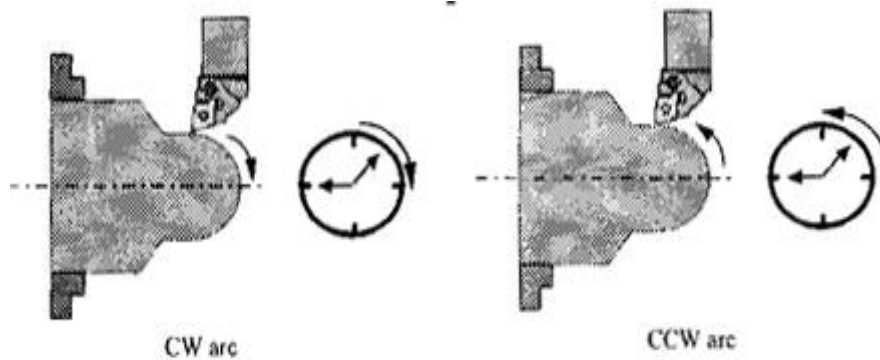
CW arc   CCW arc

Format   N_ _G02/03   X_ Z _ I_ K_F_   using arc center

or   N_ _G02/03   X_ Z_R_F_   using arc radius

*G02 moves along a CW arc , and   G03   moves along a CCW arc*

Arc center

The arc center is specified by addresses **I** and **K**. I and K are the X and Z co-ordinates of the arc center with reference to the arc start point.
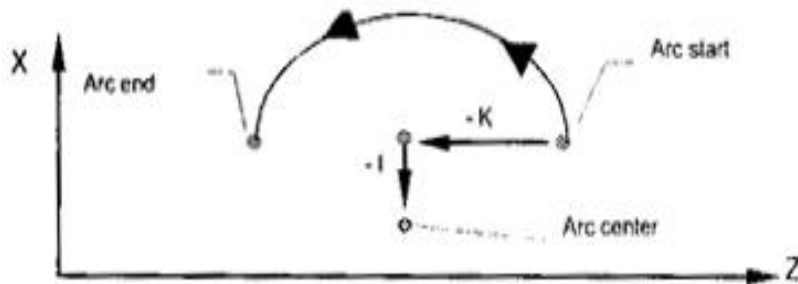
$I = (X$ coord. of center - X coord. of start point$)/2$
$K = Z$ coord. of center - Z coord. of start point
I and K must be written with their signs.

Arc   Radius

The radius is specified with address R.
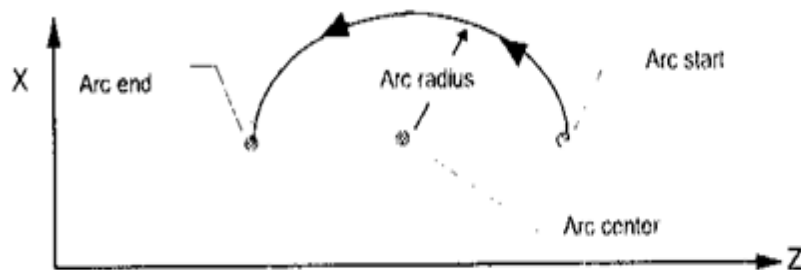
N__ G02 X__ Z__ R__ F__
N__ G03 X__ Z__ R__ F__

If the radius is used, only arcs of less than 180 deg. can be programmed in a block. An arc with included angle greater than 180 deg. must be specified in two blocks.

3.5   Miscellaneous   Functions

Typical application

  Before learning the M function, note the type of activity these function do, regardless of whether such activity relates to the machine of the program. Also note the abundance of two way toggle modes, such as ON and OFF, IN

M Code	Description
M00	Compulsory program stop
M01	Optional program stop
M02	End of program (usually with reset)
M03	Spindle rotation normal
M04	Spindle rotation reverse
M05	Spindle stop
M06	Automatic tool change ATC
M07	Coolant mist ON
M08	Coolant ON (coolant pump motor ON)
M09	Coolant OFF (coolant pump motor OFF)
M19	Spindle orientation
M30	Program end (always with reset and rewind)
M48	Feed rate override cancel OFF
M49	Feed rate override cancel ON
M60	Automatic pallet change
M78	B axis clamp
M79	B axis unclamp
M98	Subprogram call
M99	Subprogram end

- **Application for turning**

M Code	Description
M00	Compulsory program stop
M01	Optional program stop
M02	End of program (usually with reset)
M03	Spindle rotation normal
M04	Spindle rotation reverse
M05	Spindle stop
M07	Coolant mist ON
M08	Coolant ON (coolant pump motor ON)
M09	Coolant OFF (coolant pump motor OFF)
M10	Chuck open
M11	Chuck close
M12	Tailstock quill IN
M13	Tailstock quill OUT
M17	Turret indexing forward
M18	Turret indexing inverse
M19	Spindle orientation (optional)
M21	Tailstock forward
M22	Tailstock backward
M23	Thread gradual pull-out ON
M24	Thread gradual pull-out OFF
M30	Program End (always with reset and rewind)
M41	Low gear selection
M42	Medium gear selection 1
M43	Medium gear selection 2
M44	High gear selection
M48	Feed rate override cancel OFF
M49	Feed rate override cancel ON
M98	Subprogram call
M99	Subprogram end

Revision   Questions

1. What are the steps in program planning?

2.Explain the basic terms used in CNC Programming

3.With example ,express the word address programming format ?

4.With sketch explain the preparatory functions in milling, drilling operations   used in:

   interpolation -   cutting cycles

5. With sketch explain the preparatory functions in turning operations used in:

   interpolation -   cutting cycles

6. State only five of miscellaneous commands in milling ,drilling operations?

Chapter.4   NC Part Program :

NC part programming consists of planning and documenting the sequence of processing Steps to be performed on an NC machines, the part programmer must have a knowledge Of machining as well as geometry . The documentation portion of the part programming involves the input medium used to transmit the program of instructions to the NC machine

Control unit (MCU).The traditional input medium (1950) to the NC machines is 1-inch wide punched tape . more recently ,the use of magnetic tape and floppy disks have been growing in popularity as storage technologies for NC .Part programming can be accomplished using a variety of procedures ranging from highly

Manual to highly automated methods. These methods are :

1.   Manual Part Programming

2.   Computer Assisted Part Programming

3.   Part Programming Using CAD/CAM

4.   Manual Data Input

The following are explanations of the different methods of part programming with some Examples .

1. Manual part programming:

In manual part programming ,the programmer prepares the NC code using the low level machine language .the program is either written by hand on a form from which a punched tape or other storage media is subsequently coded ,or it is entered directly into A computer equipped with NC part programming software ,which writes the program onto the storage media .In any case ,the part program is a block by block listing of the machining instructions for the given job ,formatted for the particular machine tool to be used. Manual part programming can be used for both point-to-point and contouring jobs. It is Most suited for point –to-point machining operations such as drilling . It can also be used for simple contouring jobs, such as milling and turning when two axes are involved. However ,for complex three dimensional machining operations ,there is an advantage in using computer assisted part programming

2. NC Part Program using CAD/CAM:

A CAD/CAM system is a computer interactive graphics system equipped with software in accomplish certain tasks in design and manufacturing . one of the important tasks performed on a CAD/CAM system is NC part programming . in this method of part programming portions of procedure usually done by the part programmer are instead done by the computer. the main two tasks of part programing are :

1.define the part geometry   2.Specifying the tool path

2.1. Geometry Definition using CAD/CAM :

A Fundamental objective of CAD/CAM is to integrate the design engineering and manufacturing engineering functions . a computer model of each part is developed by the designer and stored in CAD/CAM data base .that model contains all of the geometric , dimensional and material specifications of the part.

When the same CAD/CAM system or a CAM system that has access to the same CAD data base in which the part model resides , is used to perform NC part program it makes little sense to recreate the geometry of the part during the programming procedure . in stead the programmer has the capability to retrieve the part geometry model from the storage and to use that model to construct the appropriate cutter path .

2.2. Tool Path Generation using CAD/CAM :

The second task of the NC programmer is tool path specification. the first step in specify the tool path is to select the cutting tool for the operation. most CAD/CAM systems have tool libraries that can be called by the programmer to identify what tools are available in the tool path. the programmer must describe which of the available tools is most appropriate for the operation under the considerations and specify it for the tool path . the second step is tool path definition. there are differences in capabilities of the variations CAD/CAM systems ,which result in different approaches for generating the tool path, the most basic approach involves the use of interactive graphics system to enter the motion commands one by one similar to computer assisted part programming(APT languages) , but the more advanced approach for generating

tool path commands is to use one of the automatic software modules available on the CAD/CAM system, these modules have been developed to accomplish a number of common machining cycles for milling ,drilling and turning..

when the complete part program has been prepared , the CAD/CAM system can provide an animated simulation of the program for validation purposes.

Computer – Automated Part Programming

In CAD/CAM approach to NC part programming several aspects of the procedure are automated ,it should be possible to automate the complete NC programming procedure. Given the geometric model of the part that has been defined during product design .the computer automated system would possess sufficient logic and decision making capability to accomplish NC part programming for the entire part without human assistance.

3. Manual Data Input (MDI)(conversational programming):

It is a potential method of simplifying the procedure is to have the machine operator perform the part programming task at the machine tool . this is called manual data input(MDI) because the operator manually enters the part geometry data and motion commands directly into MCU prior to running the job.it is also known as a conversational programming .

Communication between the machine operator programmer and MDI system is accomplished using a display monitor and alphanumeric keyboard . entering the programming commands into the controller is typically done using a menu-driven procedure in which the operator responds to prompts and questions posed by the NC system about the job to be machined .

4. Computer Assisted Part Program:

In computer assisted part programming the machining instructions are written in English – like statements that are subsequently translated by the computer into the low level machine code that can be interpreted and executed by the machine tool controller . when using one of the part programming languages the two main tasks of the programmer are : 1. defining of the part geometry   2. specifying the tool path and operation sequence.

1.Defining the part geometry

No matter how complicated the work part may appear ,it is composed of the basic geometric elements and mathematically defined surfaces , consider our sample part in figure , although its appearance is somewhat irregular ,the outline of the part consists of intersecting straight lines and a partial circles. the hole locations in the part can be defined in terms of the X-and Y- coordinates of there centers nearly any component that can be conceived by the a designer can described by points , straight lines , planes , circles , cylinders and other mathematically defined surfaces.

Let us begin with the simplest geometric elements ,a point. The simplest way to define the point is by means of its coordinates , for example

P4 = POINT / 35,90,0

Where the point is identified by a symbol (P4), and its coordinates are given in the order x ,y , z In mm (x=35 mm , y = 90 mm and z=0) . A line can be define by the two points as in the following

L1 = LINE / P1 , P2
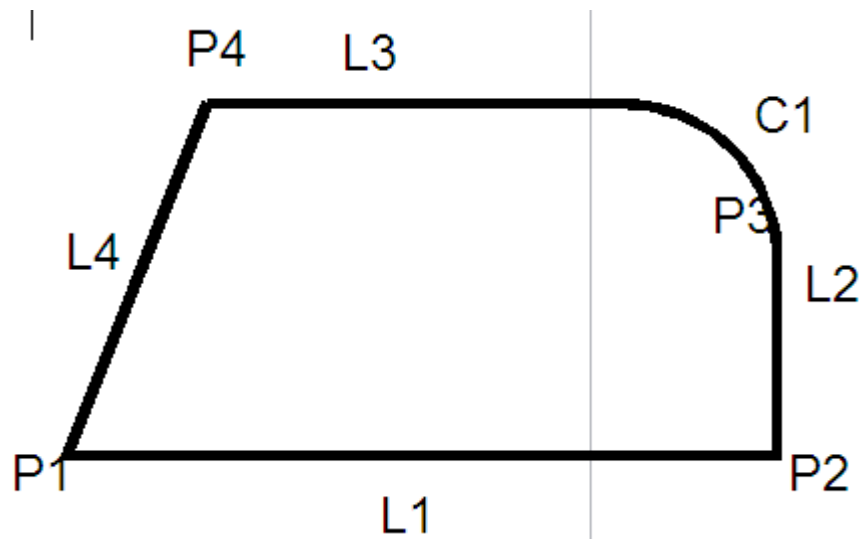
Figure. Sample part with geometry elements (points, lines and circle) labeled for computer assisted part program

where L1 is the line defined in the statement and P1 and P2 are two previously defined points and finally a circle can be defined by its center location and radius

C1 = CIRCLE / CENTER , P8 , RADIUS ,30

Where C1 is the newly defined circle with center at previously defined point P8 and radius = 30 mm , our example based on the APT language.

2. Specifying Tool Path and Operation Sequence:

After the part geometry has been defined the part programmer must next specify the tool path that the cutter will follow to machine the part .the tool path consists of a sequence of connected line and arc segments using the previously defined geometry elements to guide the cutter. For example suppose we are machining the outline of our sample part in previous figure , in profile milling operation (contouring), we have just finished cutting along surface L1 in a counter clock wise direction around the part and the tool is presently located at the intersection of surfaces L1 and L2 . The following APT statement could be used to command the tool to a left turn from L1 onto L2 AND TO CUT ALONG l2

GOLFT / L2 , TANTO , C1

The tool proceeds along surface L2 until it is tangent to (TANTO) circle C1 . This is a continuous path motion command.

a variety of contouring and point to point motion commands are available in the APT language.

3.Other Functions

In addition to defining part geometry and specifying tool path the program must also accomplish various other programming functions such as :

1. naming the program

2.Identify the machine tool on which the job will be performed

3.Specifying cutting speed and feed rates

4.Designing the cutter size (cutter radius , length of tool etc.)

5. specifying tolerances in circular interpolation

Computer Tasks in Computer assisted part programming

The tasks performed by the computer in computer assisted part programming are the following :

1. Input translation

2. arithmetic and cutter offset computations

3. editing

4. post processing

the first three tasks are carried out under the supervision of the language processing program . for example , the APT language uses processor designed to

interpret and process the words, symbols and numbers written in APT . other languages require their own processors .

The forth task post processing require a separate computer program . the sequence and relationship of the tasks of part programmer and the computer are shown in the following figure
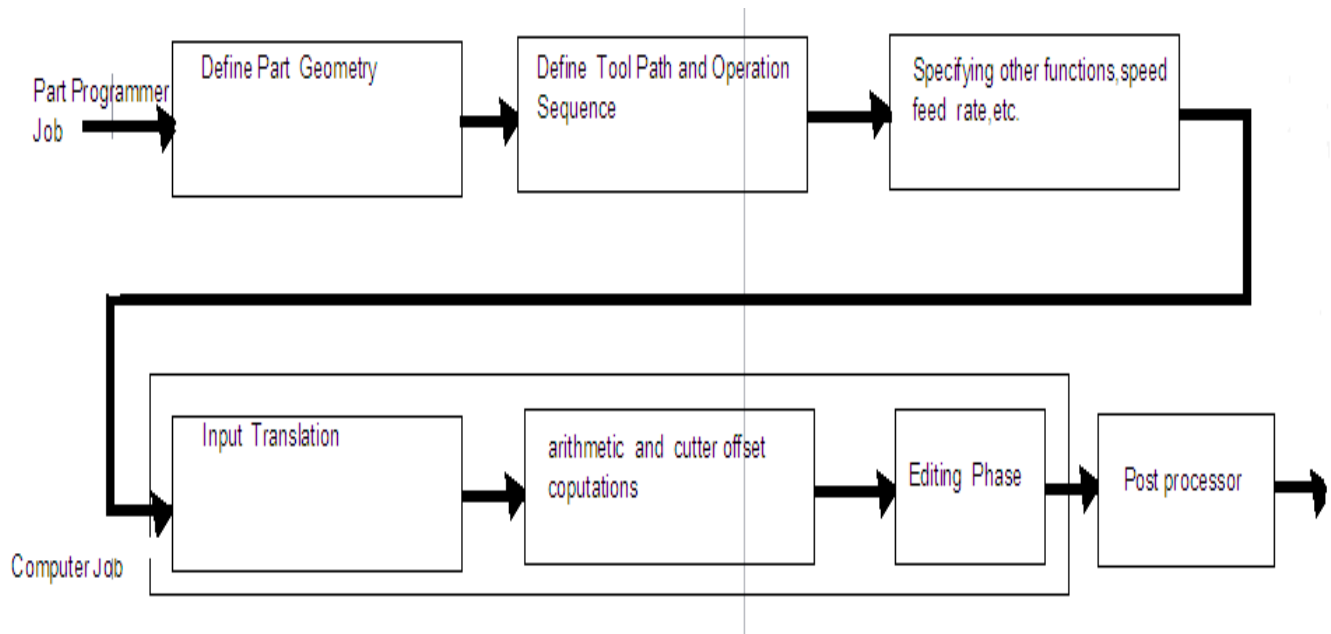
Figure Tasks in computer assisted part programming

Revision   Questions:

1.What are the different methods of NC part program?

2.Briefly , explain the method of manual data input in NC program?

3. Briefly , explain the method part program using CAD /CAM method?

4.Briefly , explain the method of part program using conversation method?

5.Briefly , explain the computer assisted part program method using for NC part program?

6.What are the computer tasks in computer assisted part program method?

Chapter.5 CNC Application Programs:

5.1 Milling and Drilling Application Programs:

Programming Sheet

Programming Sheet

Drg No.	PROGRAMMING SHEET										MATERIAL:				
	TITLE:					WRITTEN BY				SHEET No. OF					
Description		Prep Code	Misc Code	Axis Coordinates						Feed Rate	Spindle Speed				Tool No.
	N	G	M	X	Y	Z	I	J	K	F	S	P	Q	R	T

Program Format

A program is a series of "blocks" or lines each showing a set of functions and/or coordinates.

A typical format for a mill or router is:

N	G	M	X	Y	Z	I	J	K	F	S	P	Q	R	T

These are the headings used on the programming sheet, where:

N	=	block or sequence number.
G	=	preparatory function.
M	=	miscellaneous function.
X	=	X co-ordinate (absolute or incremental depending on programming mode).
Y	=	Y co-ordinate (absolute or incremental depending on programming mode).
Z	=	Y co-ordinate (absolute or incremental depending on programming mode).
I	=	incremental move in X from start of arc to arc centre when circular interpolating.
J	=	incremental move in Y from start of arc to arc centre when circular interpolating.
K	=	incremental move in Z from start of arc to arc centre when circular interpolating.
F	=	feed rate in mm/min.
S	=	spindle Speed in RPM
P	=	Time dwell (in milli seconds) or a Canned cycle parameter.
Q	=	Canned cycle parameter.
R	=	Radius value when used with circular interpolation or a canned cycle parameter.
T	=	Tool Change.

Note: F and S values are modal throughout a program. I.e. they remain the active value until a new value is specified.

Tutorial 2 - Linear Interpolation

This tutorial illustrates the use of the G01 code; cutting takes place in a straight line at a controlled feed rate.

Consider the component illustrated below; it is required to mill the two L shaped slots.


```
Material:          15mm thick Plastic
Grooves:           4 wide x 2 deep
Tool change at:    X 0, Y 0, Z 25
Tool used:         4mm Slot Drill
```
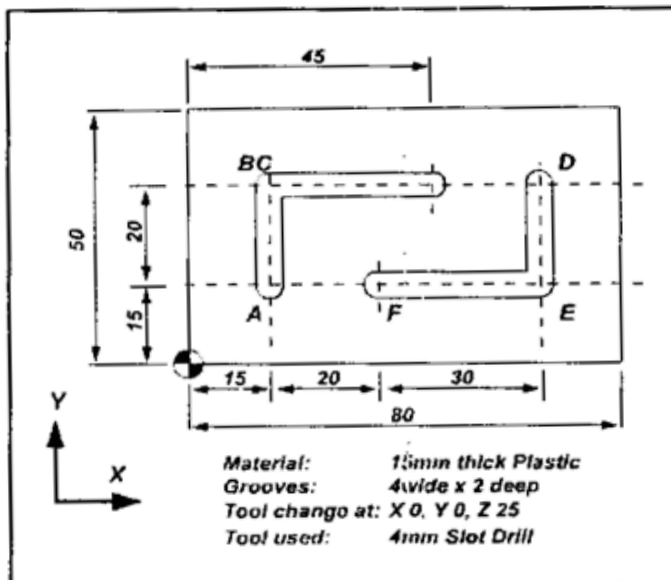
Absolute Co-ordinates

The absolute co-ordinates of the points on the profile are tabulated below.

Point	A	B	C	D	E	F	T.C.
X	15	15	45	65	65	35	0
Y	15	35	35	35	15	15	0

Tools used : 4mm Ø Slot Drill

The program is written as follows:
Notes:
G90 and G21 can be specified in the same block because they are from different groups.
G00, G01, Feed Rates (F) and Axis Co-ordinates (X, Y, Z) are modal therefore only changes are specified.

Drg No.			PROGRAMMING SHEET							MATERIAL: PLASTIC					
			TITLE: TUT2-1			WRITTEN BY				SHEET No. 1 OF 1					
Description		Prep Code	Misc Code	Axis Co-ordinates						Feed Rate	Spindle Speed				Tool No.
	N	G	M	X	Y	Z	I	J	K	F	S	P	Q	R	T
Abs Cords Metric Units	10	90 21													
1st Tool	20		06												01
Park Position	30	00		0	0	25									
Spindle on	40		03								2000				
Above A	50			15	15	2									
Feed to Depth	60	01				-2				200					

A to B	70				35										
B to C	80			45											
Pull out	90	00				2									
Above D	100			65											
Feed to Depth	110	01				-2				200					
D to E	120				15										
E to F	130			35											
Pull out	140	00				2									
Park Position	150		05	0	0	25									
End of Program	160		30												

ncremental Co-ordinates

he incremental co-ordinates of the points on the profile are tabulated below.

Point	Tool Change	A	B	C	D	E	F	T.C. from F
X	0	35	0	30	20	0	-30	-35
Y	0	35	20	0	0	-20	0	-15

Ising these co-ordinates the program can be written as follows:

Drg No.	PROGRAMMING SHEET									MATERIAL: PLASTIC					
	TITLE: TUT2-2				WRITTEN BY					SHEET No. 1 OF 1					
Description	Prep Code	Misc Code	Axis Co-ordinates						Feed Rate	Spindle Speed				Tool No.
	N	G	M	X	Y	Z	I	J	K	F	S	P	Q	R	T
Abs Cords Metric Units	10	90 21													
1st Tool	20		06												01
Park Position	30	00		0	0	25									
Spindle on	40		03								2000				
Inc Cords Above A	50	91		15	15	-23									
Feed to Depth	60	01				-4				200					
A to B	70				20										
B to C	80			30											
Pull out	90	00				4									
Above D	100			20											
Feed to Depth	110	01				-4				200					
D to E	120				-20										
E to F	130			-30											
Pull out	140	00				4									
Park Position	150		05	-35	-15	23									
End of Program	160		30												

1. Examine the program listed

2. Enter the program into the computer

Tutorial 3 - Canned Cycles and Macros - Hole Drilling and Pocket Milling

Canned cycles and Macros are used to shorten and simplify the CNC program.
This tutorial gives examples of the G81 and G83 Hole Drilling Canned Cycles and G65 - P1088 Pocket Milling cycle.

G81 - Drilling & Spot Drilling

This code drills a hole at the X, Y position specified on in the G81 block. If no X, Y position is specified, the hole is drilled in the current position.
The hole is drilled to the depth Z.

A retraction plane can be specified under the R address. If specified, the tool rapids to the R point, Feeds to Z, retracts to R and rapids to the initial point.

A hole repeat value can be specified under the K address. If a K value is used, the G81 block must include an incremental X, Y value or the repeats will be drilled in the same position. This means the current modal G code from Group 1 must be G91 or G91 must be included on the cycle line.

G83 - Drilling Cycle with Peck

This code is identical in its operation to G81 described above with the addition of a Q address. The Q value represents the depth of cut for each peck and is always specified as an incremental +ve value (-ve values are ignored). The R address is non optional.
In between pecks, the tool rapids to the retraction plane to clear the swarf then rapids back to a position just above where the last cut was performed before feeding to the next peck depth.

For the component shown below, holes A and B are to be drilled using G81 and C and D drilled using G83.

Tool change at: X 0, Y 0, Z25
Tool used: 4mm Slot Drill
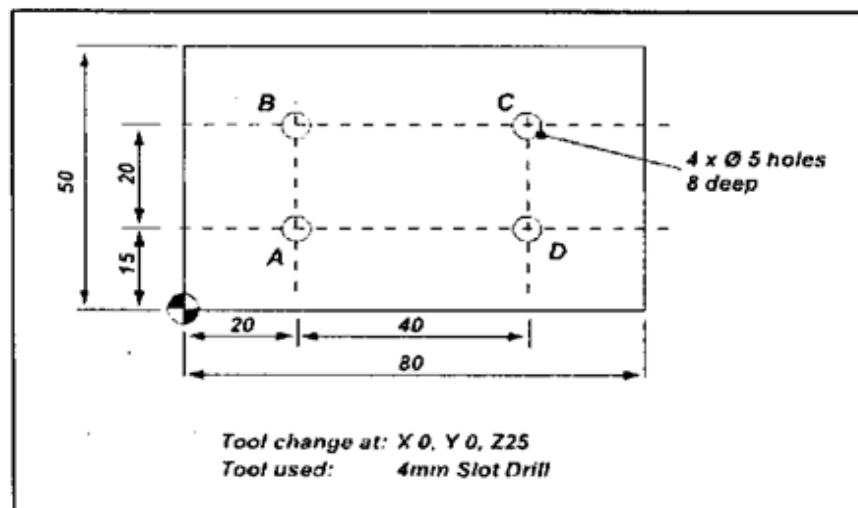
Absolute Co-ordinates

Point	A	B	C	D
X	20	20	60	60
Y	15	35	35	15

The program is written as follows:
Notes:
Canned cycles are modal therefore to produce hole D after hole C, only the change in axis position needs to be specified.

Drg No.	PROGRAMMING SHEET			MATERIAL: PLASTIC			
	TITLE: TUT3-1		WRITTEN BY	SHEET No. 1 OF 1			
	Prep	Misc	Axis Co-		Feed	Spindle	Tool

Description	N	Code G	Code M	ordinates X	Y	Z	I	J	K	Rate F	Speed S	P	Q	R	No. T
Abs Cords Metric Units	10	90 21													
1st Tool	20		06												01
Park Position	30	00		0	0	25									
Spindle on	40		03								2000				
Start Position	50			20	-5	10									
Inc Cords Drill Hole A & B	60	91 81			20	-18			2	200				-8	
Cancel Cycle	70	80													
Abs Cords Drill Hole C	80	90 83		60	35	-8				200			5	2	
Drill Hole D	90				15										
Park Position	100	00	05	0	0	25									
End of Program	110		30												

G65 - Macro Call

This code allows custom macros to be executed from the X, Y position included in the G65 block or from the current position if not specified.

The G65 code is not modal.

Macros are stored under a P address number.

Rectangular Pocket Macro

The P address for a rectangular pocket macro used in this tutorial is P1088.

X,Y = Centre of Pocket.

Z = Absolute depth

I = pocket X length

J = Pocket Y length

F = Feedrate

P = Macro address number.

Q = Number of equal passes to reach depth Z.

R = Retraction plane as used with drilling cycles described above.

The tool returns to its start point after the macro is complete.

Material: 15mm thick Plastic
Pocket: 8 deep
Tool change at: X 0, Y 0, Z 25
Tool used: 4mm Slot Drill
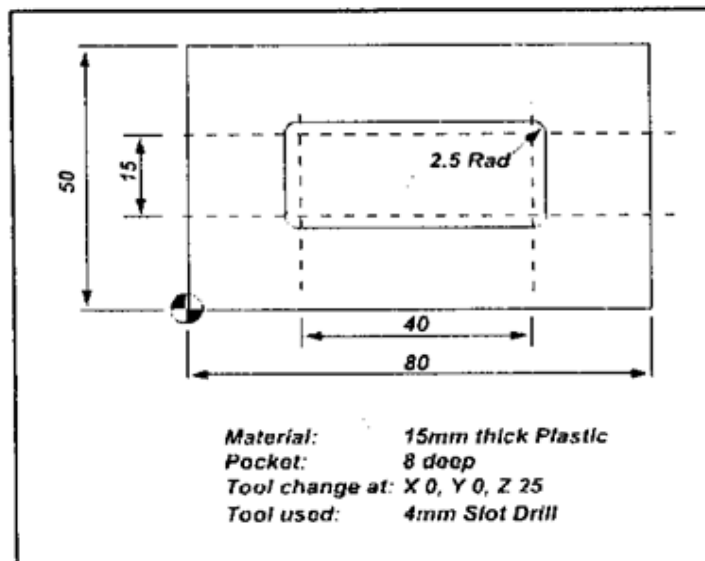
The program to produce the component shown above is as follows:

Drg No.		PROGRAMMING SHEET								MATERIAL: PLASTIC					
		TITLE: TUT3-2			WRITTEN BY					SHEET No. 1 OF 1					
Description		Prep Code	Misc Code	Axis Co-ordinates						Feed Rate	Spindle Speed				Tool No.
	N	G	M	X	Y	Z	I	J	K	F	S	P	Q	R	T
Abs Cords Metric Units	10	90 21													
1st Tool	20		06												01
Park Position	30	00		0	0	25									
Spindle on	40		03								2000				
Macro call	50	65		40	25	-8	40	15		200		1088	4	2	
Park Position	60	00	05	0	0	25									
End of Program	70		30												

G66 - Modal Macro Call

If a number of identical macros are required, the G66 code can be used.
All the macro parameters should be defined within the G66 block excluding axis positional commands.

On each subsequent line which includes a move along an axis, the macro is executed after the axis move is completed.

G66 is cancelled with a G67 - macro modal call cancel command.

An example program is shown below.

Drg No.		PROGRAMMING SHEET								MATERIAL: PLASTIC					
		TITLE: TUT3-3			WRITTEN BY					SHEET No. 1 OF 1					
Description		Prep Code	Misc Code	Axis Co-ordinates						Feed Rate	Spindle Speed				Tool No.
	N	G	M	X	Y	Z	I	J	K	F	S	P	Q	R	T
Abs Cords Metric Units	10	90 21													
1st Tool	20		06												01
Park Position	30	00		0	0	25									
Spindle on	40		03								2000				
Modal Macro call	50	66				-8	25	20		200		1088	4	2	
Execute Macro	60			20	20										
Execute Macro	70			60	30										
Cancel Modal Macro	80	67													
Park Position	90	00	05	0	0	25									
End of Program	100		30												

1. Examine the three programs

2. Enter each program into the computer.

3. Simulate the machining cycle for each program and compare the movement of the cutter with the program.

1. Tabulate the co-ordinates

2. Use a programming sheet and write a program to
   l drill the holes
   l mill the square pocket

   (For tool changing, refer to the section 'MDI - Programming a Tool change' ?)

3. Enter the program into the computer and simulate it.

4. Edit your program as necessary.

30

4 x Ø 2 x 5 deep

50

30

30

10

20

60

80

Material: 15mm thick Plastic
Square Pocket: 5mm deep
Tool change at: X 0, Y 0, Z 25
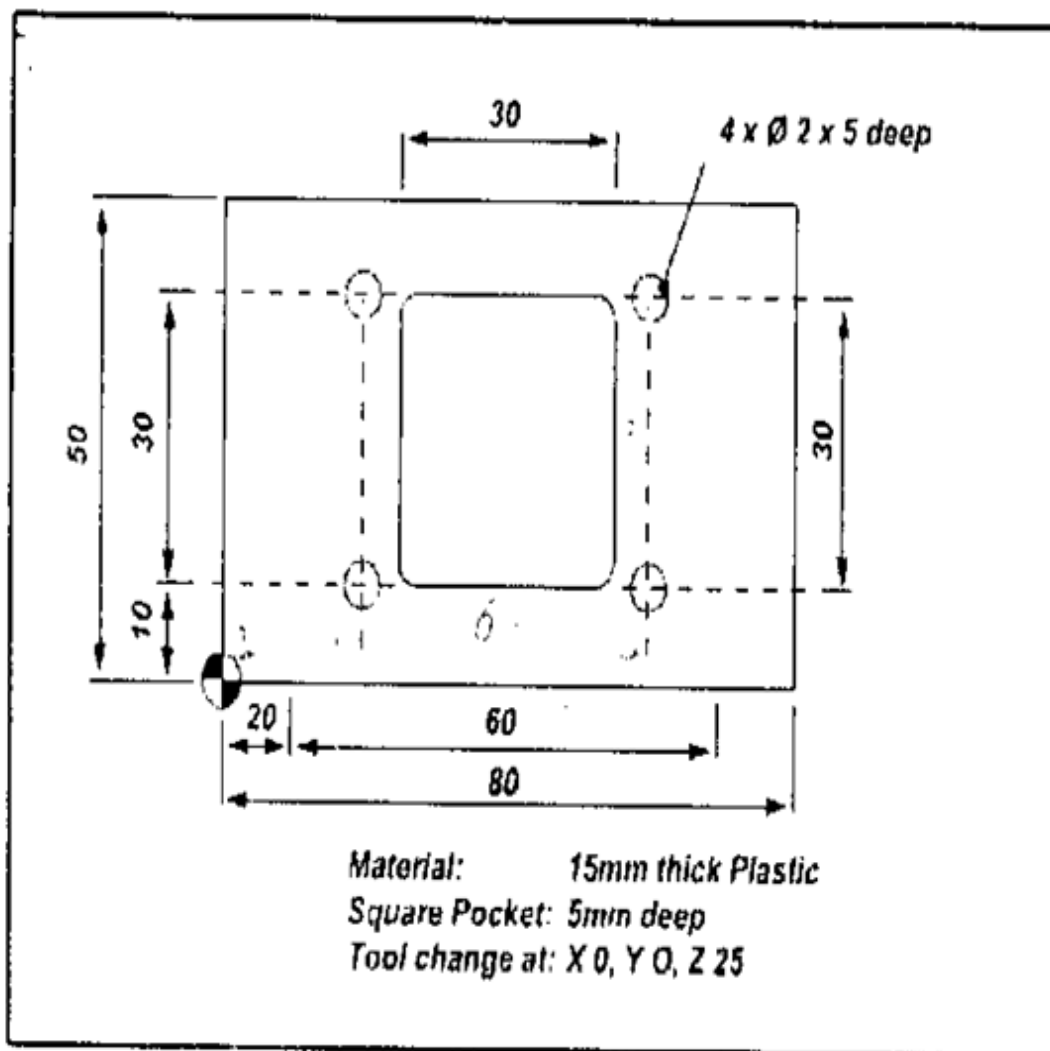
Tutorial 5 - Combining Operations and Tool Changing

This tutorial uses some cutters not supplied with the machine. It illustrates how tool changes can be made and a number of operation can be carried out on one component.
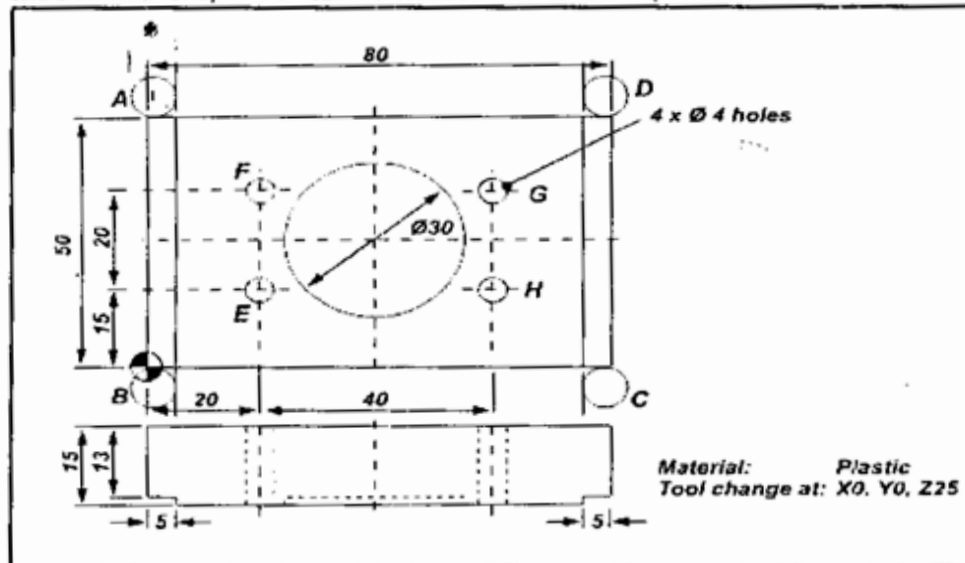
Sequence of Operations

1. Mill the steps (8mm diameter slot drill)
2. Drill the holes (4mm diameter slot drill)
3. Mill the 30mm diameter pocket (8mm diameter slot drill)

Circular Pocket Macro
The P address for a circular pocket macro used in this tutorial is P1089.
X,Y = Centre of Pocket.
Z = Absolute depth
I = Pocket outside diameter
J = Pocket inside (island) diameter. Enter a value of 0.000 for no island.
F = Feedrate
P = Macro address number.
Q = Number of equal passes to reach depth Z.
R = Retraction plane as used with drilling cycles described above.
The tool returns to its start point after the macro is complete.

Programming a Tool Change
The first tool is specified by entering an M06 T01 command, selecting the tool from the library list and entering a park position on the next line as the above tutorials show.
The second and subsequent tools are entered into the program with two lines of information. The first line stops the spindle and moves the tool away from the work and the second line selects the next tool. Entering a T number already used in the program will change back to that tool.
A typical example of this is:-

Line	G	M	X	Y	Z	I	J	K	F	S	P	Q	R	T
120	00	05	0.000	0.000	25.000									
130		06												02

The program is as follows:

PROGRAMMING SHEET	MATERIAL: PLASTIC

Description	N	G	M	X	Y	Z	I	J	K	F	S	P	Q	R	T
		Prep Code	Misc Code	Axis Co-ordinates						Feed Rate	Spindle Speed				Tool No.
Abs Cords Metric Units	10	90 21													
1st Tool-8mmØ	20		06												01
Park Position	30	00		0	0	25									
Spindle on	40		03								2000				
Above A	50			1	54	2									
Feed to Depth	60	01				-2				200					
A to B	70				-5										
Above C	80	00		79		2									
Feed to Depth	90	01				-2									
C to D	100				55										
Pull out	110	00				2									
Park Position	120		05	0	0	25									
2nd Tool-4mmØ	130		06												02
Above E	140	00	05	20	15	10									
Spindle on	150		03								2000				
Drill hole E	160	81				-8				200				2	
Drill hole F	170				35										
Drill hole G	180			60											
Drill hole H	190				15										
Park Position	200	00	05	0	0	25									
1st Tool	210		06												01
Spindle on	220		03								2000				
Circular pocket	230	65		40	25	-5	30	0		200		1089	3	2	
Park Position	240	00	05	0	0	25									
End of program	250		30												

1 Examine the program

2. Enter the program into the computer.

3. Simulate the machining cycle for each program and compare the movement of the cutter with the program.

Tutorial 6-1 - Subroutines

A subroutine can be regarded as a personalised canned cycle for use in a program which has repetitive shapes.
A subroutine is a program within a program which can be called at any time from the main program.

Subroutines are generally programmed in incremental co-ordinates so they can be called at any position.

M98 = Subroutine Call
P = Subroutine Number = 0001

The illustration below shows such a component; the milled slots can be programmed as a subroutine.

Material: 15mm thick Plastic
Grooves: 4 wide x 3 deep
Tool change at: X 0, Y 0, Z 25
A,B : start positions for subroutines

The program is as follows:

Drg No.		PROGRAMMING SHEET								MATERIAL: PLASTIC					
		TITLE: TUT6-1				WRITTEN BY				SHEET No. 1 OF 1					
Description		Prep Code	Misc Code	Axis Co-ordinates						Feed Rate	Spindle Speed				Tool No.
	N	G	M	X	Y	Z	I	J	K	F	S	P	Q	R	T
Abs Cords Metric Units	10	90 21													
1st Tool	20		06												01
Park Position	30	00		0	0	25									
Spindle on	40		03								2000				
Above A	50			5	5	2									
Subroutine call	60		98									0001			
Above B	70			55	30										
Subroutine call	80		98									0001			
Park Position	90		05	0	0	25									
End Program	100		30												
Subroutine 0001															
Inc Cords	10	91													

Description	N	G	M	X	Y	Z	I	J	K	F	S	P	Q	R	T
Feed to depth	20	01				-5				200					
Slot 1	30				15						2000				
Slot 2	40			20											
Slot 3	50				-15										
Slot 4	60			-20											
Pull out	70	00				5									
Abs Cords	80	90													
End Subroutine	90		99												

1. Examine the program

2. Enter the program into the computer.

3. Simulate the machining cycle for each program and compare the movement of the cutter with the program.

Tutorial 6-2 - Nested Subroutines
In this tutorial, a subroutine and a nested subroutine are used to create the component shown below.

Subroutines can call other subroutines. This is known as nesting.

Material: 15mm thick Plastic
Groovos: 4 wide x 3 deep
Tool change at: X 0, Y 0, Z 25
A,B :   start positions for subroutines
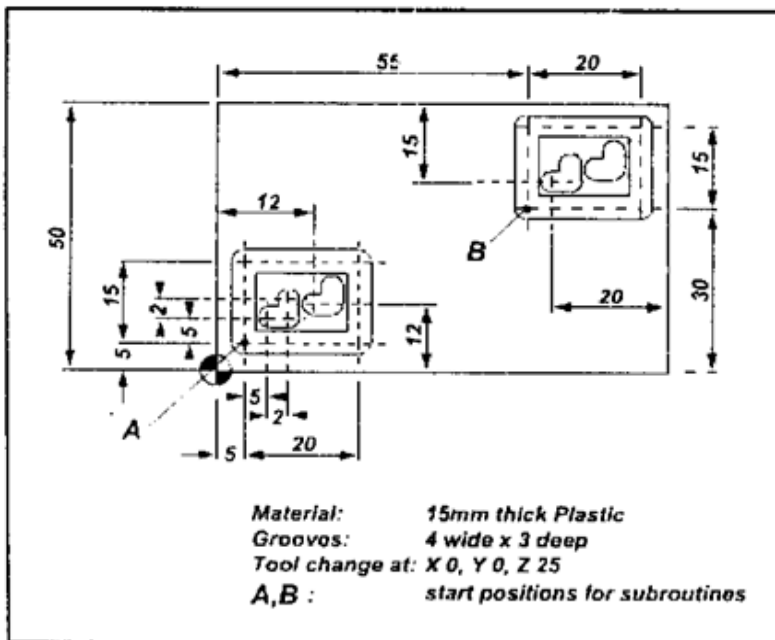
The program is as follows:

Drg No.		PROGRAMMING SHEET								MATERIAL: PLASTIC					
		TITLE: TUT6-2			WRITTEN BY					SHEET No. 1 OF 1					
Description		Prep Code	Misc Code	Axis Co-ordinates						Feed Rate	Spindle Speed				Tool No.
	N	G	M	X	Y	Z	I	J	K	F	S	P	Q	R	T
Abs Cords Metric Units	10	90 21													

	N			X	Y	Z				Sub	T
1st Tool	20		06								01
Park Position	30	00		0	0	25					
Spindle on	40		03					2000			
Above A	50			5	5	2					
Subroutine call	60		98							0001	
Above B	70			55	30						
Subroutine call	80		98							0001	
Park Position	90		05	0	0	25					
End Program	100		30								
Subroutine 0001											
Inc Cords	10	91									
Feed to depth	20	01				-5	200				
Slot 1	30				15			2000			
Slot 2	40			20							
Slot 3	50				-15						
Slot 4	60			-20							
Pull out	70	00				5					
Subroutine Call	80		98							0002	
Inc Cords	90	91		1	-4						
Subroutine Call	100		98							0002	
Abs Cords	110	90									
End Subroutine	120		99								
Subroutine 0002											
Inc Cords	10	91									
Move to start	20	00		5	5						
Feed to depth	30	01				-3	200				
Slot 1	40			2							
Slot 2	50				2						
Pull Out	60					3					
Abs Cords	70	90									
End Subroutine	80		99								

1. Examine the program

2. Enter the program into the computer.

3. Simulate the machining cycle for each program and compare the movement of the cutter with the program.

Left and Right Compensation

Compensation is defined as left or right in relation to the direction of cutter movement, and the cutter is displaced to left or right of the required profile by a distance equal to the cutter radius:
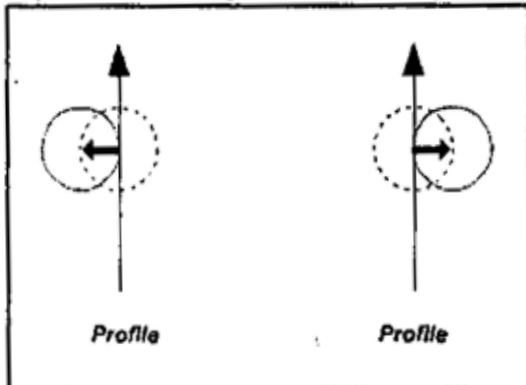
The selection of left or right compensation therefore depends on whether the profile is to be machined in a clockwise or anticlockwise direction, and whether an external or an internal profile is required, as shown below.
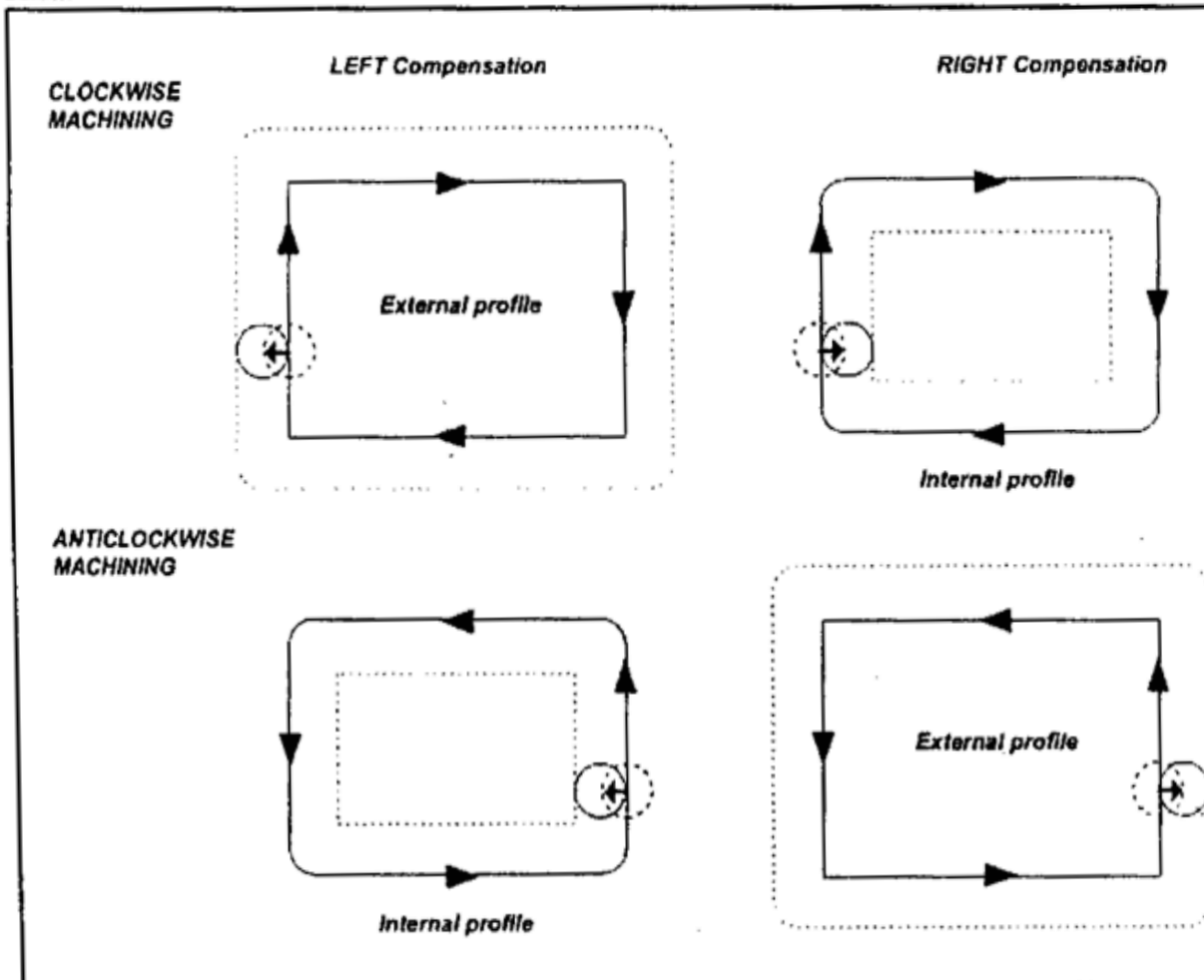
Start and End Point

When planning the machining of a profile, whenever possible, select a start and end point on a vertical or horizontal edge:

Tool parked position
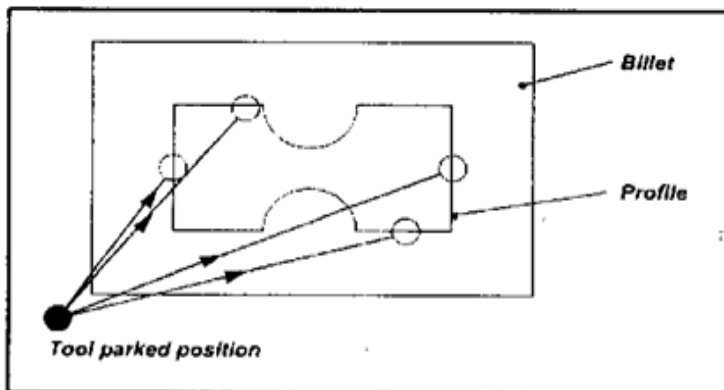
Machining will then begin at the start and end point, traverse the profile in a clockwise or anticlockwise direction as required, and finish at the start and end point.

Using a corner as a start and end point is not recommended as it may result in an incomplete profile or may cause the cutter to overrun the corner.

Example
In the following example, the start and end point has been placed on the left hand side and machining will be carried out in a clockwise direction:

Tool: 4mm Ø slot drill
Co-ordinates shown in the format X, Y
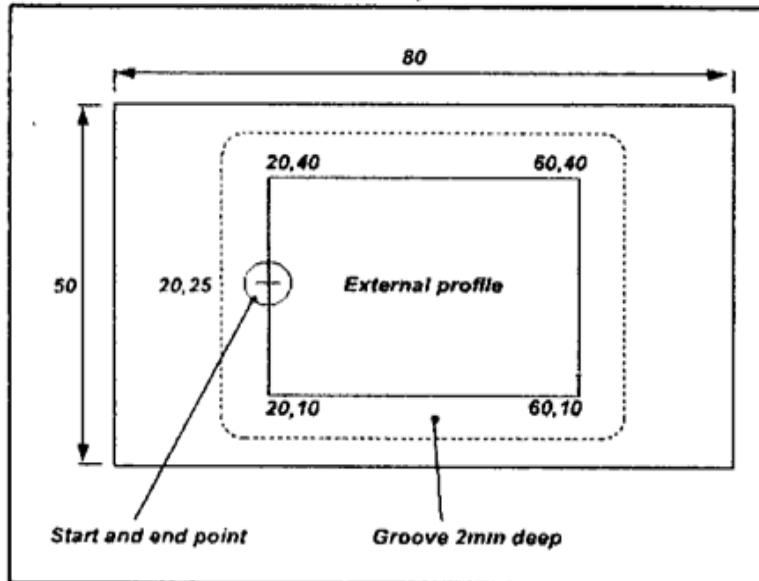
The program is written as follows:

Drg No.		PROGRAMMING SHEET					MATERIAL: PLASTIC					
		TITLE: TUT11			WRITTEN BY		SHEET No. 1 OF 1					
Description		Prep Code	Misc Code	Axis Co-ordinates						Feed Rate	Spindle Speed				Tool No.
	N	G	M	X	Y	Z	I	J	K	F	S	P	Q	R	T
Abs Cords Metric Units	10	90 21													

				X	Y	Z									
1st Tool	20		06												01
Park Position	30	00		0	0	25									
Start Spindle	40		03							2000					
Start Point	50			20	24	2									
Move in Direction of Move 1	60				25										
Compensation Left - 2 Radius	70	41												2	
Feed to depth	80	01				-2				200					
Move 1	90				40										
Move 2	100			60											
Move 3	110				10										
Move 4	120			20											
Move 5	130				25										
Pull Out	140					2									
Cancel compensation	150	40													
Park Position	160	00	05	0	0	25									
End of program	170		30												

1. Study the program.

2. Key in the program and run the simulation.

3. Compare the cutter movements with the program.

Exercise 11 - Cutter Compensation

Start and end point

Groove 2mm deep

5.2 Turning Application Programs:

Programming Sheet

Programming Sheet

Drg No.		PROGRAMMING SHEET								MATERIAL:					
		TITLE:				WRITTEN BY				SHEET No. OF					
Description		Prep Code	Misc Code	Axis Coordinates				Feed Rate	Spindle Speed						Tool No.
	N	G	M	X	Z	I	K	F	S	U	W	P	Q	R	T

Program Format

A program is a series of "blocks" or lines each showing a set of functions and/or co-ordinates.

A typical format for a lathe is:

N	G	M	X	Z	I	K	F	S	U	W	P	Q	R	T

These are the headings used on the programming sheet, where:

N = block or sequence number.

G = preparatory function.

M = miscellaneous function.

X = absolute X co-ordinate (Ø value).

Z = absolute Z co-ordinate.

I = incremental move in X (Ø value) from start of arc to arc centre when circular interpolating.

K = incremental move in Z from start of arc to arc centre when circular interpolating.

F = Feed rate in mm/min or mm/rev depending on current feed mode.

S = Surface speed in m/min or constant spindle speed in RPM depending on current speed mode.

U = incremental X co-ordinate (Ø value).

W = incremental Z co-ordinate.

P = Time dwell (in milli seconds) or a Canned cycle parameter.

Q = Canned cycle parameter.

R = Radius value when used with circular interpolation or a canned cycle parameter.

T = Tool Change.

Note: F and S values are modal throughout a program. I.e. they remain the active value until a new value is specified.

Tutorial 1 - Linear Interpolation and Rapid Traverse

This tutorial introduces simple part programming using the G01 preparatory code (linear interpolation) and also the G00 code (rapid movement).

The program is for the profile from A to E.
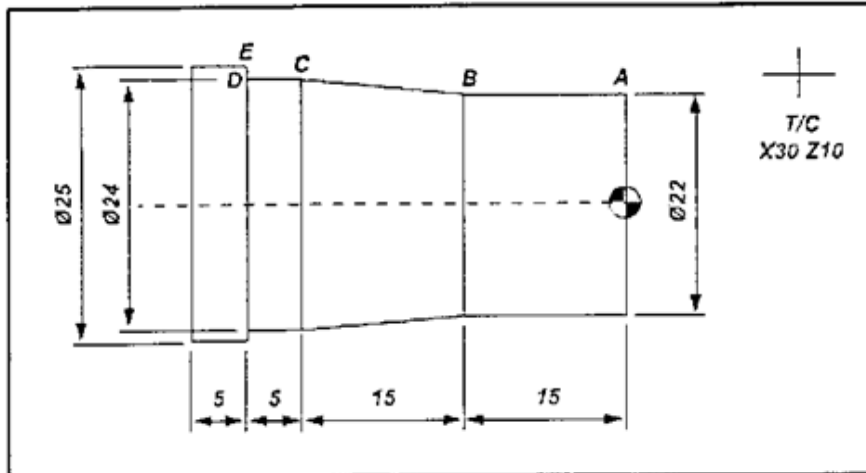
The absolute co-ordinates of the points on the profile are tabulated below.

Point	Datum	A	B	C	D	E
X	0	22	22	24	24	25
Z	0	0	-15	-30	-35	-35

Tools used : LH Turning Tool

The Program is written as follows:-

Drg No.		PROGRAMMING SHEET								MATERIAL: PLASTIC						
		TITLE: TUT1			WRITTEN BY					SHEET No. 1 OF 1						
Description		Prep Code	Misc Code	Axis Co-ordinates				Feed Rate	Spindle Speed							Tool No.
	N	G	M	X	Z	I	K	F	S	U	W	P	Q	R		T
Metric	10	21														
1st Tool	20															01
Park Position	30	00		30	10											
Move to Start	40		04	22	2				200							
A to B	50	01		22	-15			0.05								
B to C	60			24	-30											
C to D	70				-35											
D to E	80			25												
Spindle Stop	90	00	05	30	10											
End Program	100		30													

Tutorial 2 - Incremental Co-ordinate Programming

The program is for the profile from A to E.
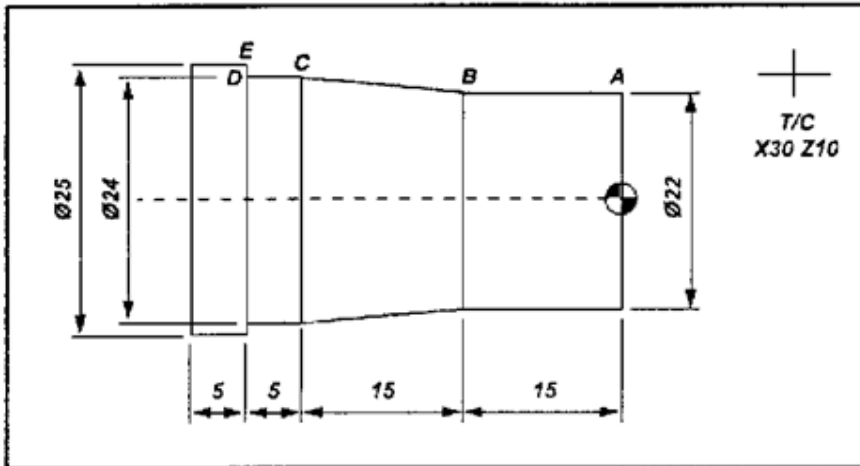
The incremental co-ordinates of the points on the profile are tabulated below.

Point	Datum	A	B	C	D	E
X	0	20	0	2	0	1
Z	0	0	-15	-15	-5	0

The program is written as follows:

Drg No.		PROGRAMMING SHEET							MATERIAL: PLASTIC						
		TITLE: TUT2			WRITTEN BY				SHEET No. 1 OF 1						
Description		Prep Code	Misc Code	Axis Co-ordinates				Feed Rate	Spindle Speed						Tool No.
	N	G	M	X	Z	I	K	F	S	U	W	P	Q	R	T
Metric	10	21													
1st Tool	20														01
Park Position	30	00		30	10										
Move to Start	40		04	22	2				200						
Move to A	50	01			0			0.05							
A to B	60										-15				
B to C	70									2	-15				
C to D	80										-5				
D to E	90									1					
Spindle Stop	100	00	05	30	10										
End Program	110		30												

Exercise 2 - Simple Programming - Incremental Co-ordinates

Using the component shown in exercise 1:

Tutorial 3 - Turning and Facing a Stepped Shaft

This tutorial introduces the student to the completion of operation and programming sheets and the writing of a program for a component requiring roughing and finishing cuts.

The operations sheet lists the sequence of machining operations, feed, spindle speed and the tool required. It also gives details of the material and its position in the chuck.

A roughing and a finishing cut have been asked for on each diameter. The programming sheet lists the program to machine the component; an explanation of each line has been given in the left hand column. The datum is at the right hand end of the component.

OPERATIONS SHEET - TURNING

DRAWING NUMBER					
BILLET	MATERIAL ALUMINIUM			TYPE NO. 1	TITLE
	25 O/DIA. 0 I/DIA. 40 STICKOUT				TUT 3

OPERATION NUMBER	OPERATION DESCRIPTION	SPINDLE SPEED	TOOL FEED	TOOL NO.
1	FACE	2500	70	1
2	TURN Ø 23.5 x 35	''	''	
3	'' Ø 22 x 35	''	''	
4	'' Ø 20.5 x 31	''	''	
5	'' Ø 19 x 30	''	''	
6	CHAMFER 1.5 x 45° TO Ø 23	''	''	
7	TURN Ø 17 x 10	''	''	
8	'' Ø 15 x 10	''	''	
9	TOOL PARK POSITION		RAPID	
10	END PROGRAM			

Drg No.		PROGRAMMING SHEET							MATERIAL: PLASTIC					
		TITLE: TUT3			WRITTEN BY				SHEET No. 1 OF 1					
Description		Prep Code	Misc Code	Axis Co-ordinates				Feed Rate	Spindle Speed					Tool No.
	N	G	M	X	Z	I	K	F	S	U	W	P	Q	R	T
Metric	10	21													
1st Tool	20														01
Park Position	30	00		30	10										
Spindle On	40		04						200						
Move to Start	50	00		26	0										

Operation	N	G	M	X	Z		F
Face Off	60	01		-1			0.05
Rapid Clear	70	00			1		
Move to 1st Ø Cut	80	00		23.5			
Cut Ø 23.5 x 35 Long	90	01			-35		0.05
Rapid Clear	100	00		24.5	1		
Move to Next Cut	110	00		22			
Cut Ø 22 x 35 long	120	01			-35		0.05
Move Clear	130	01		26			0.05
Rapid Clear	140	00			1		
Move to Next Cut	150	00		20.5			
Cut Ø 20.5 x 30 Long	160	01			-30		0.05
Rapid Clear	170	00		21.5	1		
Move to Next Cut	180	00		19			
Cut Ø 19 x 30 Long	190	01			-30		0.05
Cut Chamfer x 45°	200	01		23	-32		0.05
Rapid Clear	210	00			1		
Move to Next Cut	220	00		17			
Cut Ø 17 x 10 Long	230	01			-10		0.05
Rapid Clear	240	00		18	1		
Move to Next Cut	250	00		15			
Cut Ø 15 x 10 Long	260	01			-10		0.05
Move Clear	270	01		20			0.05
Park Position/Spindle Off	280	00	05	30	10		
End Program	290		30				

Exercise 3- Turning and Facing a Stepped Shaft
The component for this exercise is similar to that in Tutorial 3.

1. Write an operations sheet for this component.

2. Write a program to machine the component.

3. Test run your program and edit if necessary.
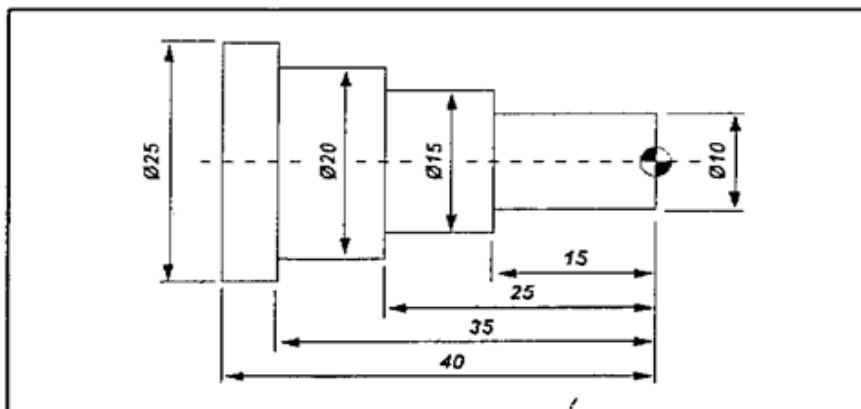
Tutorial 4 - Circular Interpolation

This gives an explanation of the G02 and G03 codes for clockwise and anticlockwise circular interpolation respectively.

The programs are for the profile only and do not include any roughing cuts. Do not use these programs to machine a component.

Circular Interpolation (G02 and G03)

To program an arc it is necessary to define a FINISH point by an X (U) and Z (W) dimension and, in addition either(a) I and K values which give the co-ordinate of the arc centre from the start point in the X and Z directions respectively; (these are unsigned) or (b) the radius of the arc as an R value.

The start and finish point of an arc must be in the same quadrant. A semicircle requires two blocks to be programmed.

G02 Clockwise Circular Interpolation
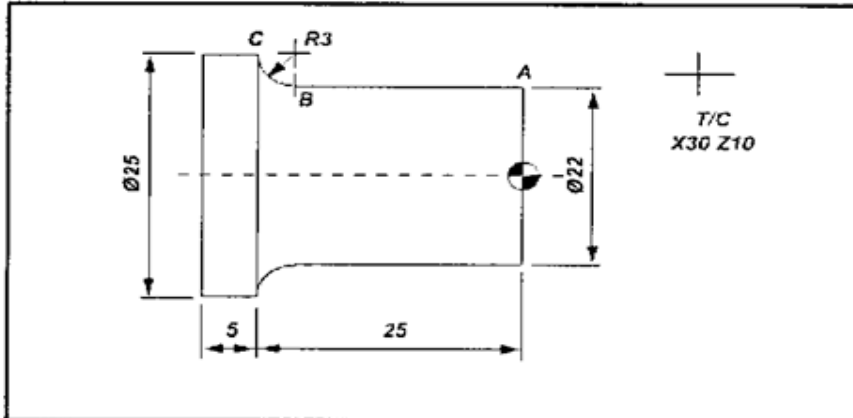
The absolute co-ordinates of the points on the profile are tabulated below:

Point	Datum	A	B	C
X	0	17	17	25
Z	0	0	-22	-25

Tool Used: LH Turning Tool

The program is written as follows:

Drg No.		PROGRAMMING SHEET							MATERIAL: PLASTIC						
		TITLE: TUT4-1			WRITTEN BY				SHEET No. 1 OF 1						
Description		Prep Code	Misc Code	Axis Co-ordinates				Feed Rate	Spindle Speed						Tool No.
	N	G	M	X	Z	I	K	F	S	U	W	P	Q	R	T
Metric	10	21													
1st Tool	20														01
Park Position	30	00		30	10										
Move to Start	40		04	22	2				200						
A to B	50	01			-23.5			0.05							
B to C	60	02		25	-25	1.5	0	0.05							
Spindle Stop	70	00	05	30	10										
End Program	80		30												

Note: In block N60 I and K are the co-ordinates of the arc centre measured from the start of the arc.

G03 Counter Clockwise Circular Interpolation
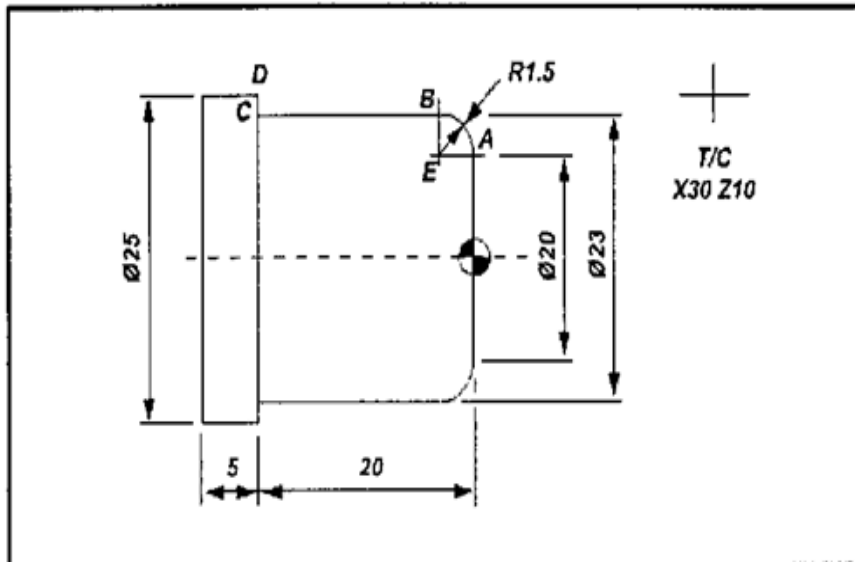
The absolute co-ordinates of the points on the profile are tabulated below:

Point	Datum	A	B	C	D	E
X	0	20	23	23	25	17
Z	0	0	-1.5	-20	-20	-3

Tool Used: LH Turning Tool

The Program is written as follows:

Drg No.	PROGRAMMING SHEET							MATERIAL: PLASTIC							
	TITLE: TUT4-2				WRITTEN BY			SHEET No. 1 OF 1							
Description	Prep Code	Misc Code	Axis Co-ordinates				Feed Rate	Spindle Speed							Tool No.
	N	G	M	X	Z	I	K	F	S	U	W	P	Q	R	T
Metric	10	21													
1st Tool	20														01
Park Position	30	00		30	10										
Move to Start	40		04	20	2				200						
Point A	50	01			0			0.05							
A to B	60	03		23	-1.5			0.05						1.5	
B to C	70	01		23	-20			0.05							
C to D	80			25				0.05							
Spindle Stop	90	00	05	30	10										
End Program	100		30												

Tutorial 5 - Circular Interpolation and Roughing Cuts with a Tool Change

This tutorial gives a further example of circular interpolation and also illustrates how to rough surplus material from the billet so that the maximum depth of cut of 3mm on the diameter is not exceeded.

Suppose that it is required to take roughing cuts before turning an arc of 4mm radius on a steel component. A scale drawing of the arc section will reveal the variations between the lengths of roughing cut. In the drawing below, the horizontal lines represent the depths of cut and the vertical lines the lengths of roughing cuts.
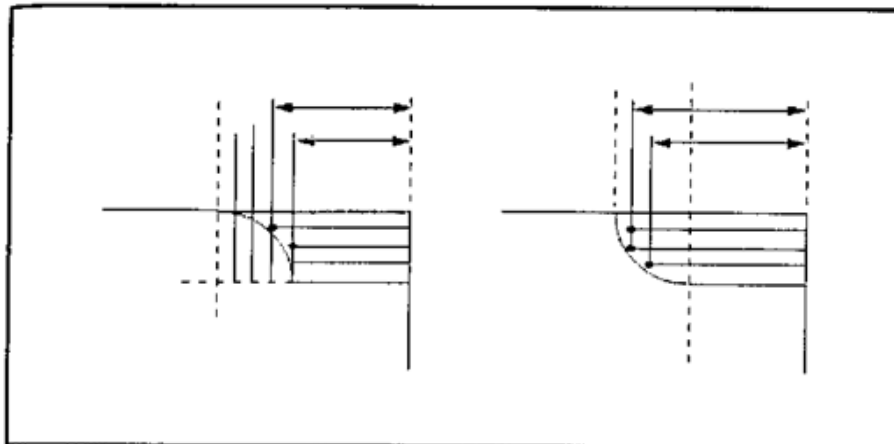
An alternative method is to take chamfering cuts on the surplus material. This is generally more difficult to program because the X co-ordinates are measured on the diameter and so are double the corresponding Z co-ordinates to give a 45 degree chamfer.

The drawing on the next page is used to determine the length of the roughing cuts; 2mm is left on the diameter before the final finishing cut is taken.

The operations sheet shows the details of the roughing cuts and a program for the component is given. Test run the program and examine its operation.
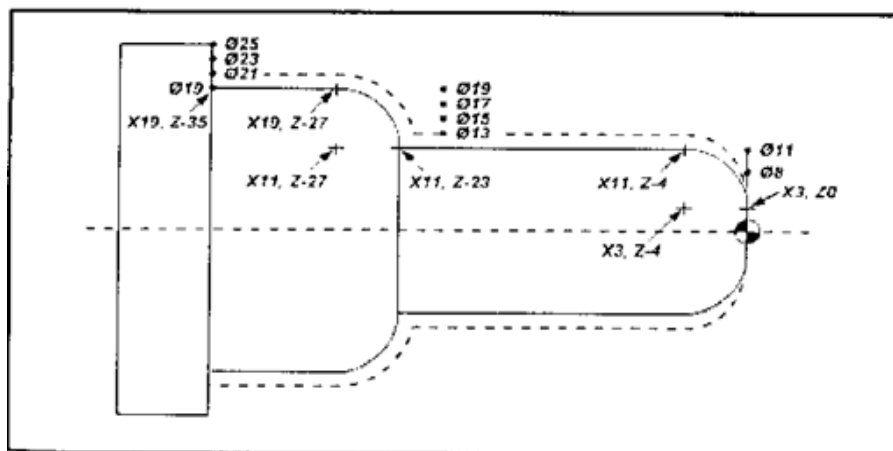
Programming a Tool Change

The first tool is specified by entering a T01 command, selecting the tool from the library list and entering a park position on the next line as the above tutorials show. The second and subsequent tools are entered into the program with two lines of information.

The first line stops the spindle and moves the tool away from the work and the second line selects the next tool. Entering a T number already used in the program will change back to that tool.

A typical example of this is:-

Line	G	M	X	Z	I	K	F	S	U	W	P	Q	R	T
70		05	30.000	10.000										
80														02

Modal Programming

When programming G01, G02 and G03, Axis positions (X, Z), G, F and S values are modal.

Therefore if the mode or value is not required to change it need not be re-entered.

ᛒ OPERATIONS SHEET - TURNING

DRAWING NUMBER								

BILLET MATERIAL PLASTIC
_ _ _ 25 _ _ _ O/DIA. _ _ _ 0 _ _ _ I/DIA. _ _ 40 _ _ STICKOUT

TYPE NO. 1
TITLE
TUT 5

OPERATION NUMBER	OPERATION DESCRIPTION		SPINDLE SPEED m/min	TOOL FEED	TOOL NO.
1	FACE		200	0.05	1
2	TURN	Ø 23 x 35	"	"	
3	"	Ø 21 x 35	"	"	
4	"	Ø 19 x 24	"	"	
5	"	Ø 17 x 23	"	"	
6	"	Ø 15 x 22	"	"	
7	"	Ø 13 x 22	"	"	
8	"	Ø 11 x 2	"	"	
9	"	Ø 8 x 1	"	"	
10	TAKE CONTINUOUS FINISHING CUT		225	0.07	
11	TOOL PARK POSITION			RAPID	
12	CHANGE TOOLS				
13	PART OFF TO LENGTH		200	0.05	2
14	TOOL PARK POSITION			RAPID	
15	END PROGRAM				

Drg No.		PROGRAMMING SHEET						MATERIAL: PLASTIC					
		TITLE: TUT5			WRITTEN BY			SHEET No. 1 OF 1					
Description		Prep Code	Misc Code	Axis Co-ordinates			Feed Rate	Spindle Speed					Tool No.
	N	G	M	X	Z	I	K	F	S	U	W	P	Q	R	T
Metric	10	21													
Select LH Turning Tool	20														01
Park Position/Spindle On	30	00		30	10										
Move to Start	40		04	26	0				200						
Face Off	50	01		-1				0.05							
Rapid Clear	60	00			1										
Move to 1st Ø Cut	70			23											
Cut Ø 23 x 35 Long	80	01			-35			0.05							
Rapid Clear	90	00		24	1										
Move to Next Cut	100			21											

Operation	N	G	M	X	Z	F	S	R	T
Cut Ø 21 x 35 Long	110	01			-35	0.05			
Rapid Clear	120	00		22	1	0.05			
Move to Next Cut	130			19					
Cut Ø 19 x 24 Long	140	01			-24	0.05			
Rapid Clear	150	00		20	1	0.05			
Move to Next Cut	160			17					
Cut Ø 17 x 23 Long	170	01			-23	0.05			
Rapid Clear	180	00		18	1				
Move to Next Cut	190			15					
Cut Ø 15 x 22 Long	200	01			-22	0.05			
Rapid Clear	210	00		16	1				
Move to Next Cut	220			13					
Cut Ø 13 x 22 Long	230	01			-22	0.05			
Rapid Clear	240	00		14	1				
Move to Next Cut	250			11					
Cut Ø 11 x 2 Long	260	01			-2	0.05			
Rapid Clear	270	00		13	1				
Move to Next Cut	280			8					
Cut Ø 8 x 0.8 Long	290	01			-0.8	0.05			
Rapid Clear	300	00		9	1				
Move to Finish Cut	310			3	0				
Increase Spindle Speed	320		04				225		
Finish Cut 4mm Radius	330	03		11	-4	0.07		4	
Finish Turn Ø 11	340	01			-23	0.07			
Finish 4mm Radius	350	03		19	-27	0.07		4	
Finish Turn Ø 19	360	01			-35	0.07			
Finish Turn Face	370			26					
Tool Change Position	380	00	05	30	10				
Select Parting Tool	390								02
Move to Part Off Position	400		04	27	-38.6		200		
Part Off	410	01		-1		0.05			
Rapid Clear	420	00		30					
Park Position	430		05		10				
End Program	440		30						

Tutorial 7 - Canned Cycles - Grooving

Examine the operations sheet and drawing.

The G75 grooving cycle is carried out using the parting off tool. This tool is 1.6mm wide and when setting offsets, its reference face is taken as the side nearest the chuck.

The tool is positioned clear of the diameter to be cut in the X axis (2mm is suggested) and is aligned to one side of the finished slot in the Z (side nearest billet front face is suggested).

Start Point=X22 Z-11.6

The G75 cycle is defined on two lines.

Line 1
R = Pull out amount when pecking (in microns) = 500

Line 2
X (U) = Finished diameter of groove = X15
Z (W) = Groove Width (less width of cutter) = W-10
F = Feedrate = 70mm/min
P = Depth of Peck (in microns) = 2000
Q = Movement in Z axis after each cut (in microns) = 1500

Test it and examine its operation.

OPERATIONS SHEET - TURNING

DRAWING NUMBER	BILLET	MATERIAL PLASTIC			TITLE TUT 7
		25 O/DIA. 0 I/DIA. 40 STICKOUT			

OPERATION NUMBER	OPERATION DESCRIPTION	SPINDLE SPEED	TOOL FEED	TOOL NO.
1	FACE	200	0.05	1
2	START POSITION	"	RAPID	
3	TURN Ø20 x 35 USING G90 CYCLE WITH 3 EQUAL CUTS	"		
4	CHAMFER 1mm x 45° TO Ø20	"	"	
5	TOOL PARK POSITION		RAPID	
6	CHANGE TOOLS			2
7	GROOVE START POSITION	200	RAPID	
8	CUT GROOVE 5 DEEP x 10 WIDE USING G75 CANNED CYCLE	"	0.05	
9	TOOL PARK POSITION		RAPID	
10	END PROGRAM			

Description	N	G	M	X	Z	I	K	F	S	U	W	P	Q	R	T
		Prep Code	Misc Code	Axis Coordinates				Feed Rate	Spindle Speed						Tool No.
Metric Units	10	21													
LH Turning Tool	20														01
Park Position	30	00		30	10										
Move to Start	40		04	26	0				200						
Face Off	50	01		-1				0.05							
Rapid Clear	60	00			1										
Move to Start	70			27	0.5										
Axial Cut 1	80	90						0.05		-3.667	-35.5				
Axial Cut 2	90									-5.333					
Axial Cut 3	100									-7					
Chamfer Start	110	00		18	0										
Cut Chamfer	120	01		20	-1			0.05							
Park Position	130	00	05	30	10										
Parting Tool	140														02
Groove Start	150	00	04	22	-11.6				200						
Grooving Cycle 1	160	75												0.5	
Grooving Cycle 2	170	75		15				0.05			-10	2000	1500	0	
Park Position	180	00	05	30	10										
End Program	190		30												

Additional header info:
- Drg No.
- PROGRAMMING SHEET
- MATERIAL: PLASTIC
- TITLE: TUT7
- WRITTEN BY
- SHEET No. 1 OF 1

Exercise 7- Canned Cycles - Grooving

You are required to produce a component which fulfils the following machining specification:

- Billet size Ø25mm x 40mm stickout from chuck.

- Billet material - Plastic.

- Face to length of 40mm with one cut.

- Outside diameter machined to 18mm in cuts not exceeding 2mm on diameter.

- Machine a groove of depth 6mm x 12mm long starting 8mm from the free end.

Complete the following tasks:

1. Complete an operation sheet for the above machining operations.

2. Using a programming sheet write a program for the component.

3. Give your program a test run and edit it if necessary.

Tutorial 8 - Canned Cycles - External LH Thread.
Table of metric and imperial threads ▥▶

Note: A threading tool is required for this tutorial.

Examine the operations sheet and drawing for TUT 8.

The component is the same as that for TUT 7 except that a screw thread has been added and the component has to be parted off.

The program for TUT 7 can be edited and extended to include the screw thread by using the G76 canned cycle.

The work datum X0 Y0 is the same as that for TUT 7. The start point for G76 will be X22 Z1.5, i.e. 2mm greater in diameter and 1.5 clear of the work in the Z axis.

Tool used: Threading tool

Start Point=X22 Z-11.6

The G76 cycle is defined on two lines.

Line 1
P = Six digit address. Digits 1 & 2 = Number of passes made at route without cutting. Digits 3 & 4 = Run out at end of thread (not supported by Boxford machine so 00 Must be used). Digits 5 & 6 = angle of the tool edge (60 for standard tooling) = 020060

Q = The smallest cut allowed (in microns) = 50

R = Finishing allowance at the root of the thread = 500

Line 2
X (U) = The core diameter of the thread = 18.770

Z (W) = The target point for the thread = -12.000

F = Thread Pitch = 0.750

P = The depth of the thread (in microns) = 615

Q = Depth of cut for the first pass. The remaining passes are automatically controlled by the system to achieve constant material removal.

R = Required for programming taper threads. Represents the difference in radius at the beginning and end of the taper thread (for parallel threads enter 0 or omit value) = omit

Note: After the first movement in the X axis there is a pause whilst the computer looks for the spindle marker point.

The program for TUT 8 is shown on the programming sheet. This can be used to extend TUT 7 at the appropriate block no.

Extend TUT 7, test it and examine its operation.

OPERATIONS SHEET - TURNING

DRAWING NUMBER								
BILLET	MATERIAL	PLASTIC				TYPE NO.	TITLE	
	25 O/DIA.		0 I/DIA.	40		STICKOUT	TUT 8	

OPERATION NUMBER	OPERATION DESCRIPTION	SPINDLE SPEED	TOOL FEED	TOOL NO.
	continued from TUT 7			
10	CHANGE TOOL			3
11	START POSITION	400 rpm	RAPID	
12	CUT Ø20 x 0.75mm PITCH THREAD USING G76 CYCLE			
13	TOOL CHANGE POSITION		RAPID	
14	CHANGE TOOL			2
15	PART OFF TO LENGTH 32mm	200	0.05	
16	TOOL PARK POSITION		RAPID	
17	END PROGRAM			

Drg No.		PROGRAMMING SHEET							MATERIAL: PLASTIC					
		TITLE: TUT8			WRITTEN BY			SHEET No. 1 OF 1						
Description		Prep Code	Misc Code	Axis Coordinates			Feed Rate	Spindle Speed						To No
	N	G	M	X	Z	I K	F	S	U	W	P	Q	R	T
Metric Units	10	21												
LH Turning Tool	20													0
Park Position	30	00		30	10									
Move to Start	40		04	26	0			200						
Face Off	50	01		-1			0.05							
Rapid Clear	60	00			1									
Move to Start	70			27	0.5									
Axial Cut 1	80	90					0.05		-3.667	-35.5				
Axial Cut 2	90								-5.333					
Axial Cut 3	100								-7					

Operation														
Chamfer Start	110	00		18	0									
Cut Chamfer	120	01		20	-1			0.05						
Park Position	130	00	05	30	10									
Parting Tool	140													0:
Groove Start	150	00	04	22	-11.6				200					
Grooving Cycle 1	160	75											0.5	
Grooving Cycle 2	170	75		15				0.05		-10	2000	1500		
Park Position	180	00	05	30	10									
Threading Tool	190													0:
Thread Start - CSS off	200	97	04	22	1.5				400					
Threading Cycle 1	210	76									020060	50	0.025	
Threading Cycle 2	220	76		18.77	-12			0.75			615	200		
Park Position	230	00	05	30	10									
Parting Tool	240													0:
Spindle and CSS On	250	96	04						200					
Parting Position	260			22	-33.6									
Part Off	270	01		-1				0.04						
Rapid Clear	280	00		25										
Park Position	290		05	30	10									
End Program	300		30											

Exercise 8 - Canned Cycles - External RH Thread.

This exercise is a repeat of Tutorial 6 except that the 20mm diameter has a right hand thread.

Note that the Z value in the G76 cycle for a right hand thread is positive. A new start position for the thread will have to be calculated.

1. Complete an operations sheet, based on TUT 7 for the shape and TUT 8 suitably modified for the thread.

2. Using a programming sheet write a program for the component.

3. Give your program a test run and edit it if necessary.