Trade of Metal Fabrication

Module 4:	Structural Steel Fabrication
Unit 11:	Introduction to CNC
	Phase 2

Table of Contents

List of Figures

List of Tables

Document Release History

Date	Version	Comments
29/01/06	First draft	
13/12/13	SOLAS transfer	

Module 4 – Structural Steel Fabrication

Unit 11 – Introduction to CNC

Duration – 4 Hours

Learning Outcome:

By the end of this unit each apprentice will be able to:

- List the technological developments which have led to the development of modern CNC machines
- List the constructional details which distinguish a CNC machine tool from a conventional machine tool
- State the advantages of CNC machines compared with conventional machines

Key Learning Points:

Rk	- Requirement for complex components microprocessors -computer technology. - Differences between NC and CNC machines. - Machine structure and frame location of servo motors, stepping motors, machine guarding arrangements.
Rk	- Repeatability. - Set-up of times, flexibility in changes of component design. Reduction of operation error.
P	- Communication, information gathering, adaptability, quality awareness.

Training Resources:

- Demonstration machine or visit to workplace to view CNC m/c in operation
- Drawings and illustrations of CNC machines
- Sample components or drawings of complex parts produced on CNC machines
- Manufacturers specifications for various machines
- Course notes/handouts/videos

Key Learning Points Code:

M = Maths D= Drawing RK = Related Knowledge Sc = Science

P = Personal Skills Sk = Skill H = Hazards

CNC Machine Tools

NC and CNC Explained

Numerical Control?

Numerical Control (NC) is the technique of giving instructions to a machine in the form of a code which consists of numbers, letters of the alphabet, punctuation marks and certain other symbols. The machine responds to this coded information in a precise and ordered manner to carry out various machining functions.

Instructions are supplied to the machine as a series of blocks of information. A block of information is a group of commands sufficient to enable the machine to carry out one individual machining operation e.g. move cutter from position 1 to position 2 at a specified feed rate.

Each block is given a sequence number for identification. The blocks are then executed in strict numerical order. An example of a block is as follows:

NI G00 X10 x Y20

The meaning of this information is as follows: NI is block number 1; G00 directs the machine or cutter to move at the rapid traverse rate. X10 and Y20 are the coordinate values of the target points of the movement.

When the instructions are organised in a logical manner they direct the machine tool to carry out a specific task - usually the complete machining of a workpiece or "part". It is thus termed a part program.

Computer Numerical Control?

Computer Numerical Control (CNC) is based on the concepts of NC but utilises a dedicated computer within the machine control unit to store the program. CNC is largely the result of technological progress in microelectronics (the miniaturisation of electronic components and circuitry), rather than any radical departure in the concept of NC.

CNC control units, like the computers on which they are based, operate according to a stored program held in computer memory. This means that part programs are now able to become totally resident within the memory of the control unit, prior to their execution. No longer do the machines have to operate on the "read-block/execute block" principle as in an NC machine. CNC machines have now completely superseded the older NC machines from which they are derived.

Constructional Features of CNC Machines

A conventional machine tool has an intelligent source for error compensation (the operator). During machining a skilled operator can vary the cutting conditions to compensate for deflection, vibration, etc. to generate the desired shape, size and finish.

The CNC machine can only compensate for an error that is detected and communicated to the control. Deflection, vibration etc. cannot, as yet, be easily monitored. For this reason NC machines are made stronger and stiffer to perform to a more accurate standard than their conventional counterparts.

The capacity for varying the conditions while machining is therefore limited. As far as possible the conditions have to be established as the program is produced.

In addition to this, CNC machines are spending more time per shift cutting than conventional machines did in the past. This higher percentage of cutting time results in faster wear rates on the slides and transmission systems.

Conventional machine tools are also designed with the view to having the skilled operator standing directly in front controlling the machine. This is no longer required for CNC since the machine is operating under program control.

Optimum cutting speeds and feeds, continuous path machining, rapid slide movement to bring the tool close to the work and then sudden stopping, all subject the machine to forces which are not encountered on conventional machines.

Machine Structure

Since rigidity plays a major part in the accuracy of a machine tool, modern CNC machines tend to have over-proportionated slide ways, guides and spindles. Thicker cast sections than conventional machines are also used. The use of symmetrical castings assists in reduction of thermal stresses within the machine. These structural design features are employed to cope with the torsional forces and heavy duty cutting imposed on these machines.

Slideways

Conventional machine tool slideways operating under conditions of sliding friction do not exhibit a constant coefficient of friction. Friction coefficient is highest at low velocity. This condition gives rise to 'stick-slip' which produces jerky slide action when movement takes place at low velocities. Numerical control also requires a rapid response of the slides to command signals from the machine control.

To eliminate these problems rolling friction can be used instead of sliding friction. Figure 1 shows a number of ways of achieving this.

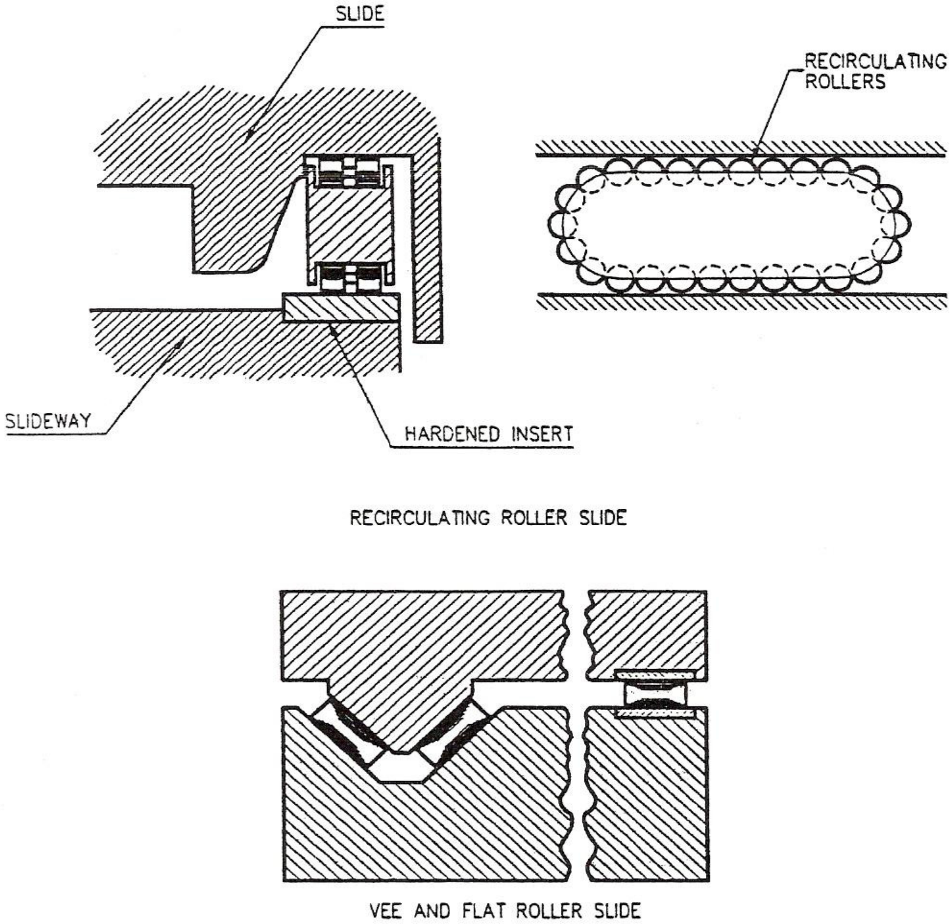


Figure 1 - Anti-Friction Slideways

Each of these methods considerably reduces frictional resistance giving rapid slide response and eliminates 'stick-slip'. This type of slideway is particularly suitable for machines which are not required to machine while the slide is moving e.g. drilling etc. When the slide reaches the point where drilling takes place it is clamped in position. The main disadvantage of this design is the load bearing and clamping capabilities are not as good as a conventional slideway. Machines which are required to machine while the slideway is moving such as milling often utilise a hydrostatic system. See Figure 2 where fluid is pumped in between the slide surfaces at high pressure up to 30,000 Kn/m². Frictional wear and stick-slip are entirely eliminated. An alternative design is to coat the sides with Polytetrafluoroethylene (P.T.F.E.). This material has a very low coefficient of friction.

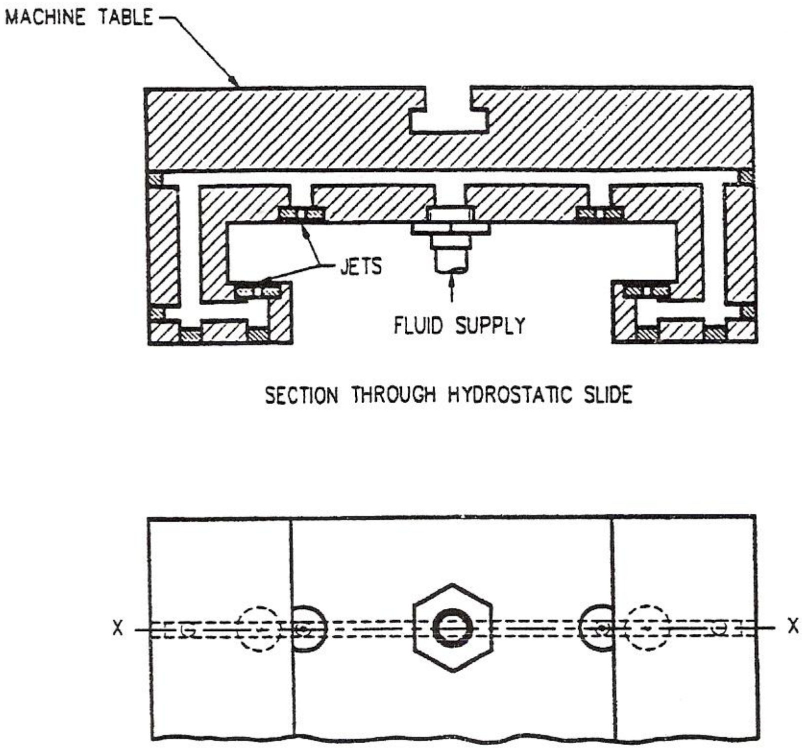


Figure 2 - Hydrostatic Slideway

Leadscrews

The lead screws used on conventional machine tools are usually of the Acme thread form. These threads are very inefficient because of the high frictional resistance between the flanks of the screw and the nut. They also exhibit high backlash, because there must be clearance between the flanks of the nut and the screw. This would not be acceptable on CNC machines.

The alternative most commonly used is the re-circulating ball leadscrew. This type of leadscrew replaces sliding friction with rolling friction. Both the lead screw and nut have a precision ground form into which an endless stream of re-circulating balls which completely fill the track is inserted. This type of leadscrew has an efficiency of up to 90%.

The advantages of recirculating ball screws over Acme screws are:

(1) longer life

(2) less frictional resistance

(3) lower torque required

(4) more precise positioning of slides because backlash is almost completely eliminated

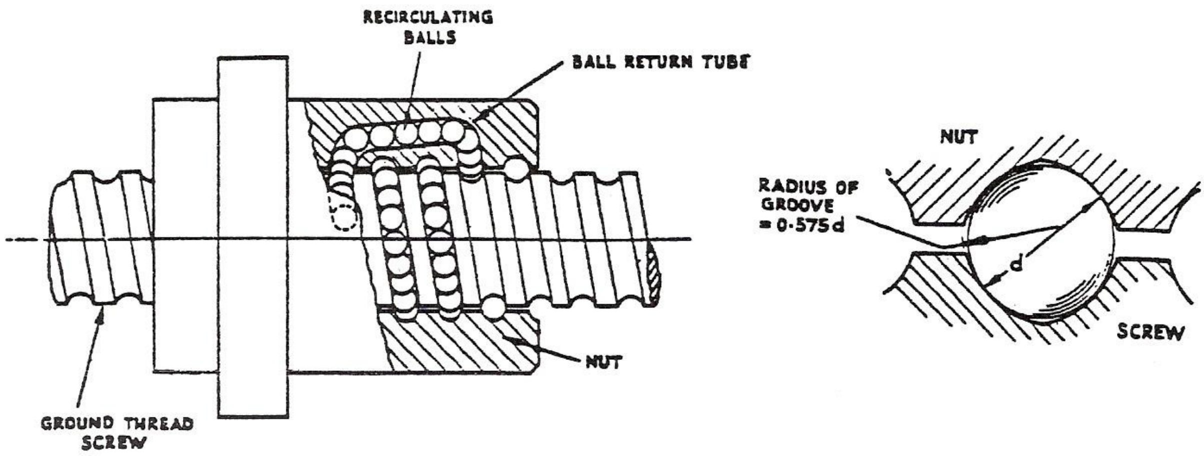


Figure 3 - Re-circulating Ball Leadscrew

Machine Guarding and Swarf Control

Many CNC machines particularly turning centres are almost totally enclosed by an envelope of elaborate guarding. This is considered necessary for the following reasons:

- large amounts of swarf and cutting fluid are present in the cutting zone.
- the high spindle speeds and feed rates create high forces which could cause serious injury in the event of a collision or tool breakage.
- these guards are usually interlocked with the control system through limit switches i.e. the spindle will only operate when the guards are fully closed; see Figure 4.

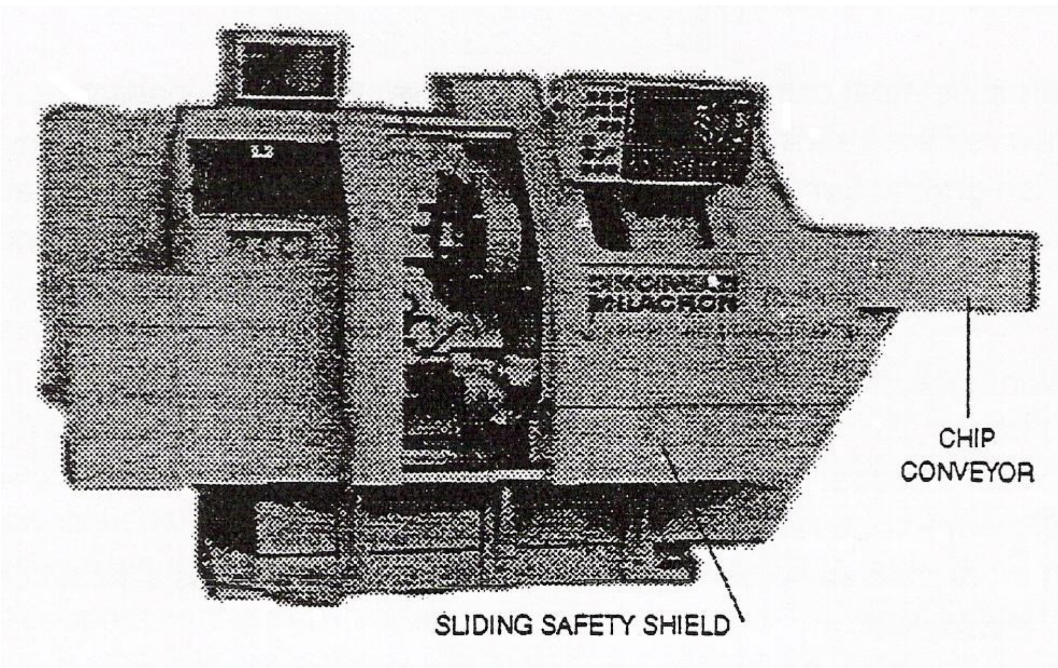


Figure 4 - Machine Guarding

Swarf Control

In conventional machinery swarf removal is given little consideration, it is usually removed by hand by the machinist. With the high rates of metal removal, heat build up and the trend towards unmanned operation the problem of swarf removal requires consideration for CNC machining.

Many manufacturers build in swarf removal equipment as part of the machine such as rotary screw or linear conveyors (see Figure 5) B.C. However, to a large extent, the basic design of the machine can assist greatly in swarf control e.g. slant-bed lathes (Figure 5a) allow the swarf to fall away from the cutting zone naturally. Swarf removal by gravity alone is not always sufficient in itself. This can be assisted by multiple coolant jets around the cutting zone which keep the tool free from accumulated swarf. At the end of a machine cycle and before a new component is loaded the area can often be cleaned of swarf by compressed air jets.

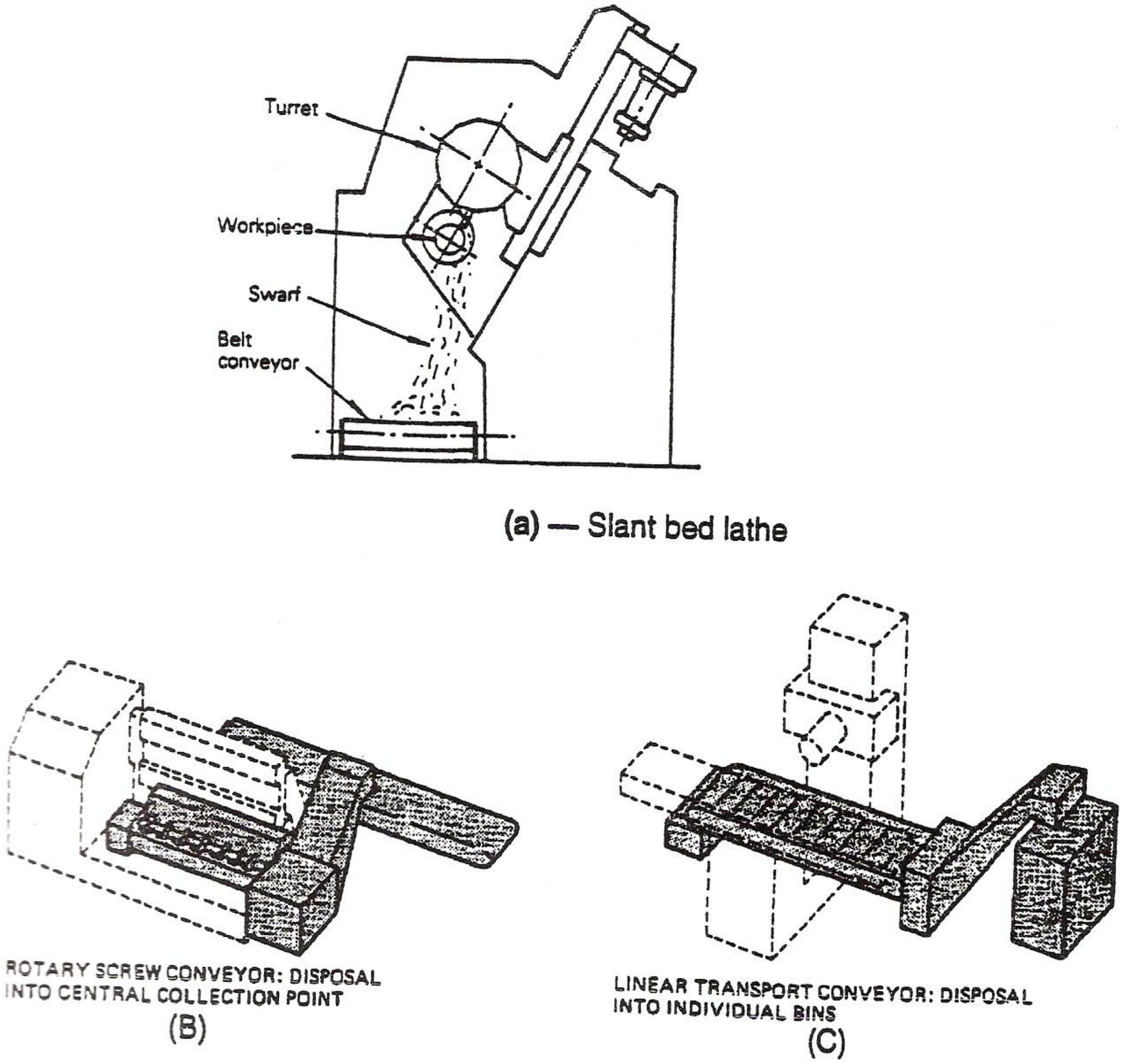


Figure 5 - Swarf Removal Conveyors

Machine Axes

The primary axes of a machine are designated as X, Y, Z and can have positive or negative values.

The Z-axis is always parallel to the main spindle of the machine (see Figure 6). It does not matter whether the spindle carries the workpiece or the tooling, therefore the Z-axis can be either vertical or horizontal. On milling, boring and drilling machines the spindle is the tool rotating means while on cylindrical grinders and lathes the spindle is the work rotating means. Positive Z-movement always increases the distance between the work and the tool.

The X-axis of motion is horizontal and parallel to the work-holding surface. If Z is horizontal, positive X is to the right looking from the spindle towards the workpiece. If Z is vertical, when looking from the spindle towards its supporting column, positive X is to the right.

The Y-axis of motion is perpendicular to both the X and Z axes. Positive Y is in the direction which would make a right-handed set of coordinates.

CNC lathes only have two major axes X and Z. There is no Y-axis.

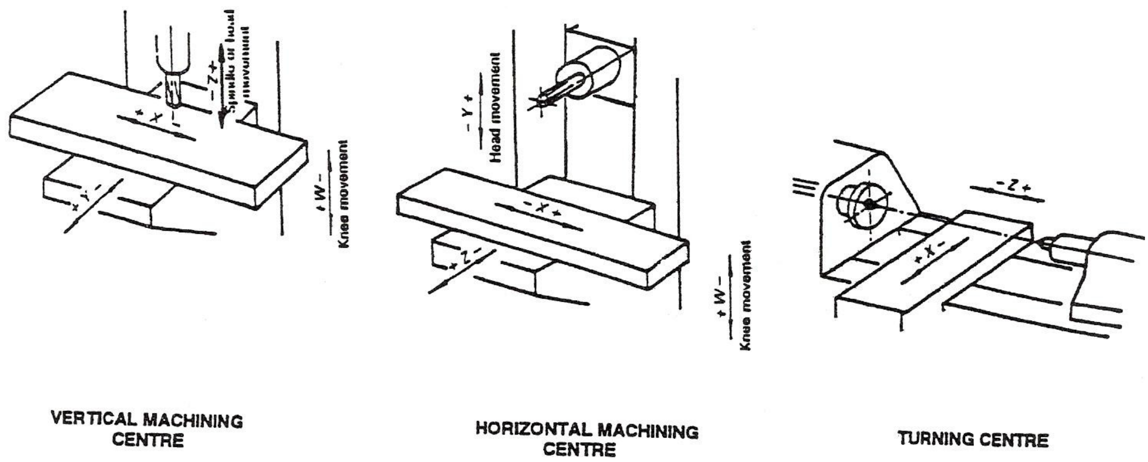


Figure 6 - X, Y, Z Axes

Some CNC machines have additional axes running in parallel with the main axes.

Where there is more than one moving element in the same axis one is called the primary axis and is designated as X Y or Z. Secondary movements in the same axis are designed by U V W in upper case letters.

Right-Hand Rule (Vertical Milling Machine)

The directions of the machine axes are easy to remember by the right-hand rule:

Face the machine and hold your right hand as shown below, with the middle finger in the direction of the tool axis (Z): the thumb will point in the direction of the X axis and the forefinger in the direction of the Y axis.

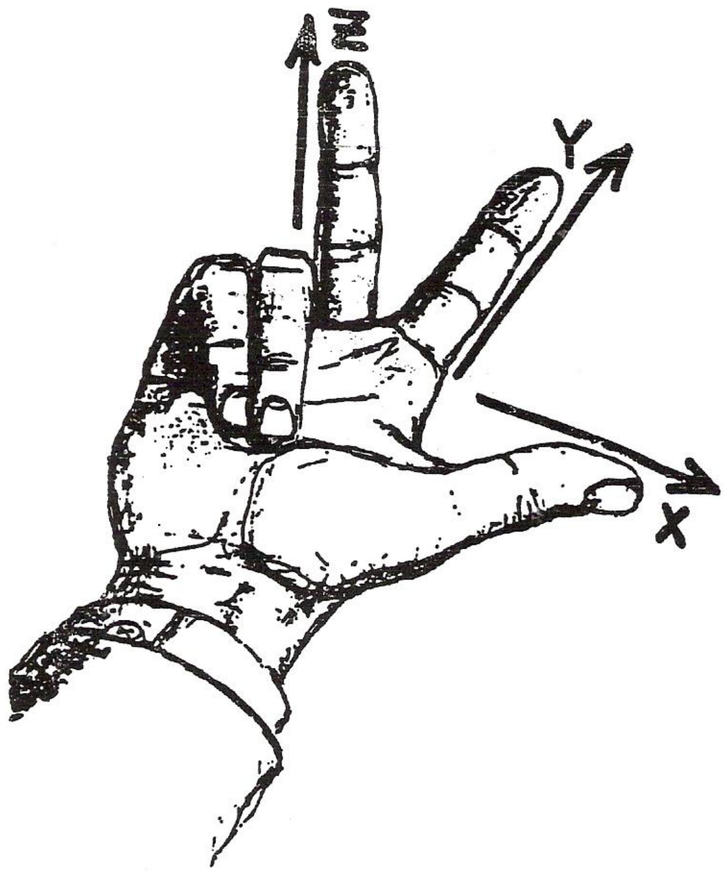


Figure 7 - Right-Hand Rule

Control of Slide Movement on CNC Machines

The control of slide position and velocity of movement can be either accomplished by open loop (non-servo) or closed loop (servo) means. In an open loop system the position and velocity of the slides are not measured.

A block diagram of an open loop control is shown in Figure 8.

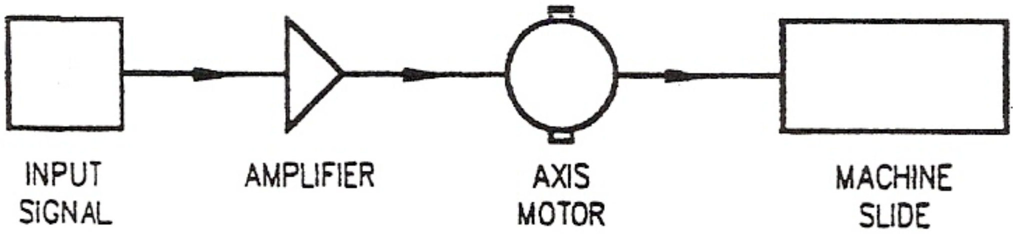


Figure 8 - Open Loop Control

Open loop systems have no means of comparing the final slide position with the position in which it was commanded to go. A special type of electric motor known as a stepper motor is usually used in open loop systems to provide accurate positioning and velocity control of slide movement.

The principle of the stepper motor is that, upon receipt of a digital signal (a pulse); the spindle will rotate through a specified angle (the step). The step size is determined by the design of the motor but will typically be between 1.8 and 7.5 degrees. Thus, if two digital pulses are applied, then the spindle rotor will rotate by 2 steps, or by between 3.6 and 15 degrees depending on the motor design. Thus, by counting (electronically) the number of pulses sent to the stepper motor, and by knowing the lead of the axis leadscrew, the distance traversed can be accurately predicted. There is no need for positional feedback.

Velocity of the axis movement is determined by how quickly the pulses are sent to the stepper motor (the pulse frequency). If the pulses are sent very rapidly, then the feed-rate will be high; if the pulses are sent very slowly then the feed-rate will be low. The speed at which the pulses are transmitted can be accurately governed by the CNC control system. Therefore there is no need for velocity feedback.

In spite of these advantages of no feedback being necessary there are some serious disadvantages in using stepper motors on CNC machine tools, i.e.:

- If the machine axis is stalled through overload the pulses will continue to count and a loss of position will occur
- The maximum power output from stepper motors is relatively low
- Because pulse rate (frequency) is limited the maximum axis feed rate is also restricted therefore the rapid traverse speeds achievable are low

Because of these limitations the stepper motor is only found on small low powered machines and on retro-fitted machines. Retro-fitted machines are conventional machine tools which have been fitted with CNC controllers.

Servo control requires both positional and velocity feedback, i.e. the actual velocity and position of the slideway must be compared to the commanded values. A servo system always forms a closed loop (see Figure 9). The devices which are used to provide the feedback in the closed loop are called transducers. A transducer is a device which converts one form of energy into another form, e.g. mechanical displacement into an electrical signal.

Two types of transducers are used on CNC machines:

- Velocity transducer: The most common velocity transducer is the tacho-generator. This is a device similar to a bicycle dynamo. It provides a voltage proportional to its speed. Tacho-generators usually come as an integral part of a servo motor. The voltage is used to provide feedback from motor speed and hence slide velocity.

- Position transducers: Slideway position is measured by transducers which can be either linear or rotary. Linear transducers operate by measuring slide movement. Rotary transducers operate indirectly by measuring leadscrew rotation or motor shaft rotation and relating this to linear slide movement.

The types of motor used for electrical servo systems can be either AC or DC servo motors. Hydraulic servo systems are also used on some CNC machines.

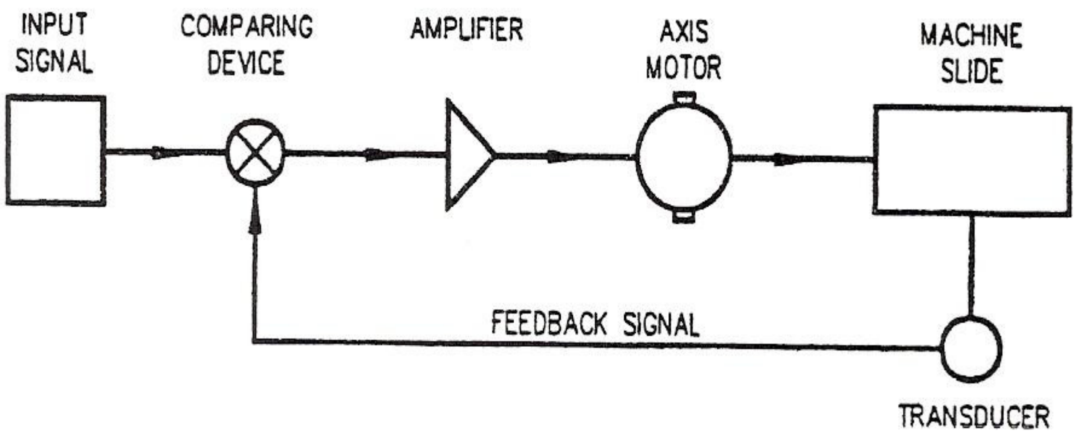


Figure 9 - Block Diagram Closed Loop Control System

Advantages of CNC Machines

Despite the extra costs involved CNC machines have many advantages. These can be summarised as follows:

- **Reduced lead time**
  Lead time is the time between the receipt of a design specification and the time in which manufacture is ready to commence. In many cases standard tooling is all that is required. The need for special jigs and fixtures is almost completely eliminated. Simple work clamping arrangements are usually sufficient.

- **Elimination of operator errors**
  Responsibility is transferred from the operator to the part program. Provided the program is correct, this is always proved in advance of production and the machine is set up properly no errors will occur in the work. Operator fatigue, boredom or inattention will not affect the quality or the duration of machining.

- **Flexibility in changes of component design**
  The modification of a component design can be very easily accommodated by altering the part program. There may be no changes required to jigs and fixtures which would typically be the case for conventional machining.

- **Reduced scrap and inspection**
  Extreme accuracy and repeatability of components produced are features of CNC machining. It is usually only necessary to inspect the first component. Once the program has been proved the repetitive accuracy of the machine maintains a consistent product.
  The cost associated with inspection, re-work and scrap are almost eliminated.

- **Complex one-offs and small batch quantities**
  CNC finds its greatest advantages in small batch quantities and complex one-offs. Machining a small batch quantity is found to be economical with CNC machines because of the fast change-over times. A part program can be prepared away from the machine while the machine is still in production. Storage of programs is not a problem and change over from one batch to another is achieved by loading the correct part program. For complex shaped one-off components and proto-types the CNC machine finds many applications because of the difficulty often encountered in producing these shapes on conventional machines.

- **Lower Labour Cost**
  CNC machines do not necessarily operate at faster cutting speeds than conventional machines, but more time is normally spent in actually cutting metal because the set up times are less. It is also possible for one operator to attend to more than one CNC machine at a time.

- **Accurate Costing and Scheduling**
  The time taken in machining is predictable and consistent which results in greater accuracy in estimating and costing.

Part Programming

Part Programming

Cartesian Coordinates

The main difference between CNC and conventional is that CNC drawings are usually dimensioned in Cartesian coordinates because this makes programming easier. The Cartesian Rectangular Coordinate System forms the basis of NC measurement.

Using rectangular coordinates, any specific point in physical space can be described in mathematical terms along two axes (X and Y). With this system, the location of any point on a flat surface or plane can be defined mathematically with reference to two lines (axes) in the same plane and perpendicular to each other.

This coordinate system can be extended to permit defining the location of any point in three-dimensional space relative to three mutually perpendicular planes. The point would be defined by three imaginary perpendicular planes with each axis being the line of intersection of two planes.

The intersection point of two axes marks the origin point. The origin is surrounded by four quadrants. Each quadrant consists of portions of the number lines forming each axis. The location of a coordinate point is specified in relation to the point of origin where the axes intersect. A two-dimensional application of this system of coordinates is illustrated in Figure 10.

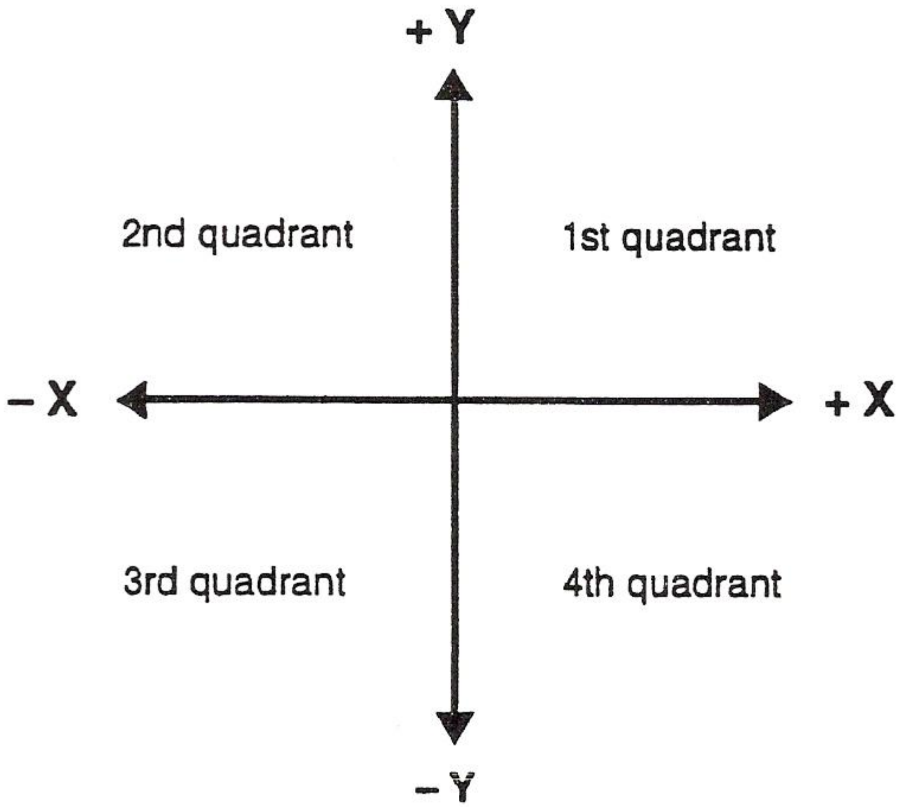


Figure 10 - Two-Dimensional Rectangular Coordinates

X, Y & Z Axes

NC milling is carried out within two planes defined by three-axes (X, Y and Z). The X and Y axes consist of individual number lines that are perpendicular to each other and lay within the same plane. The Z axis is created by extending planes into space above, below and perpendicular to the X and Y axes (see Figure 11).

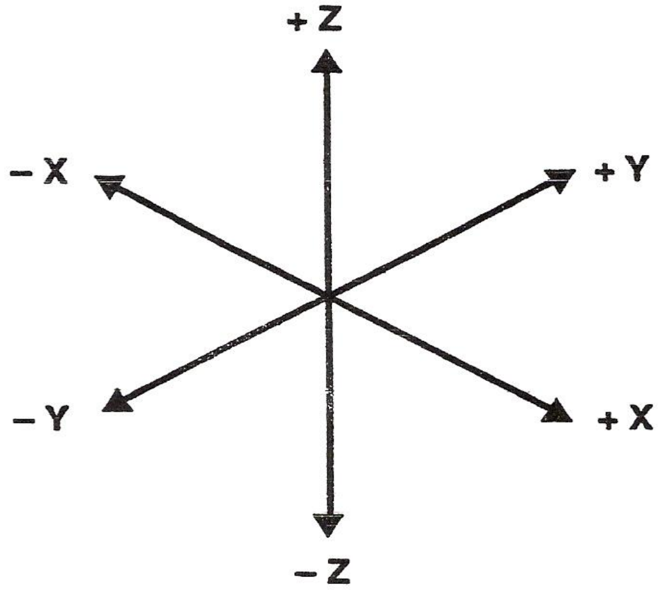


Figure 11 - X, Y and Z Axes

In the Cartesian or right angle coordinate system all points can be described as either incremental or absolute. In absolute measurement each point is always taken from the same zero. This prevents build up of tolerances between each dimension.

In incremental dimensioning also known as chain dimensioning each point is taken from the preceding position. Figure 12 shows the same component dimensioned in absolute and in incremental.

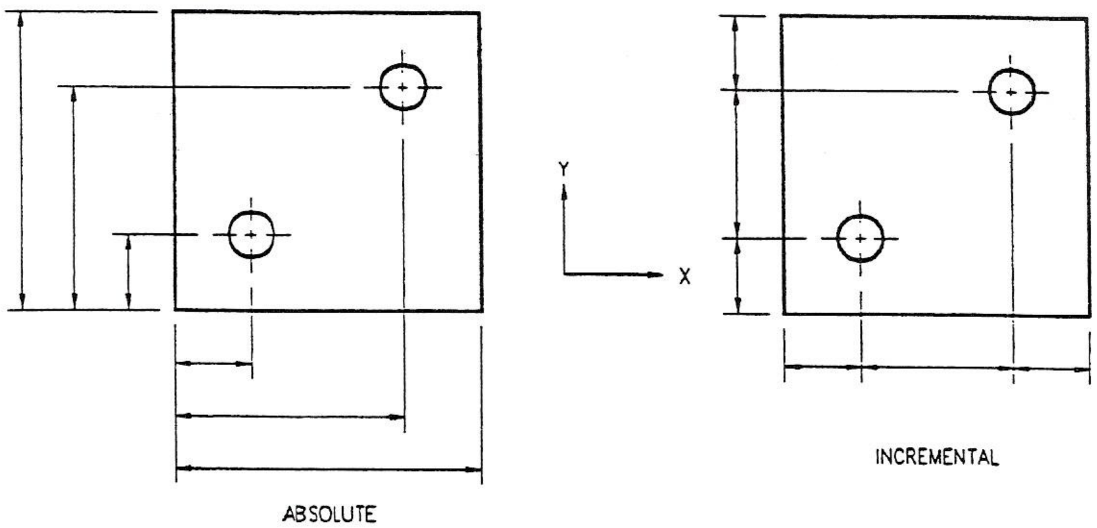


Figure 12 - Absolute and Incremental Dimensioning

Incremental dimensioning is often discouraged because of the possibility of build up in tolerances between individual features. However on CNC machines this problem is reduced because of the inherent accuracy and repeatability of the machines.

In some cases incremental dimensioning is advantageous such as where a pattern of holes is repeated at different locations on a workpiece. Most machines can be programmed either in incremental or absolute mode or switched from one mode to another during programming.

In absolute programming all axis movements are specified in relation to a fixed datum point.

In incremental programming each axis movement is measured from the last position. Each point visited when machining serves as the datum point for the next point or positional move.

Polar Coordinates

This is where features on a drawing are described by a length and an angle measured from a specific point. Figure 13 shows an example of Polar coordinate dimensioning.

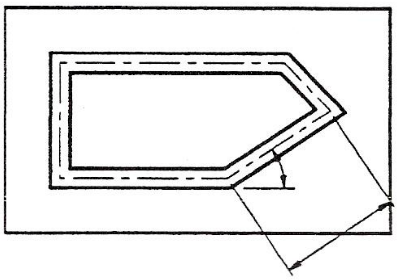


Figure 13 - Dimensioning in Polar Coordinates

Many of the more modem machines allow the use of polar coordinates while programming. This can save a considerable amount of time in programming e.g. if a number of holes around a Pitch Circle Diameter (PCD) are to be drilled - see Figure 14.

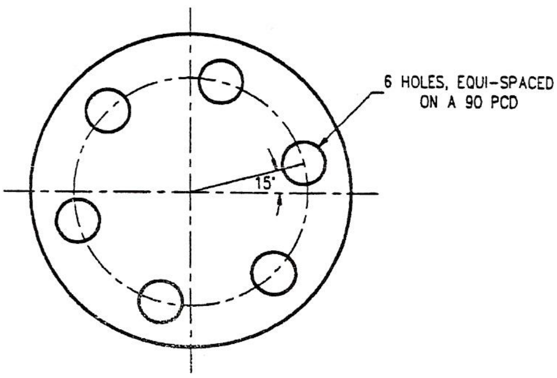


Figure 14 - Pitch Circle Diameter

If you were using Cartesian coordinates for this program you would have to work out the coordinates for each of the hole centres using sine and cosine.

Selection of Zero Point

In order to define certain points on a workpiece in this manner, you will first have to decide where to put the coordinate system on the workpiece, especially where to place the zero point.

For milling, the zero point (or reference point) of all dimensions may be placed anywhere on the workpiece. Let's call this zero point 'workpiece zero' (WZ) or workpiece datum.

To save trouble of unnecessary calculations it is advisable to place workpiece zero at that point on a part drawing on which most of the dimensions are based. See examples Figure 15 and Figure 16.

A workpiece datum maybe defined as a point, line or surface from which dimensions are referenced. It mayor may not be within the workpiece area.

The symbol used to denote workpiece zero is:

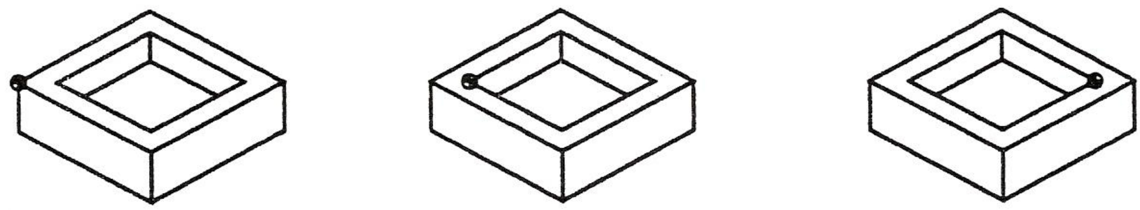


Figure 15 - Example of Zero Point Options on Workpiece

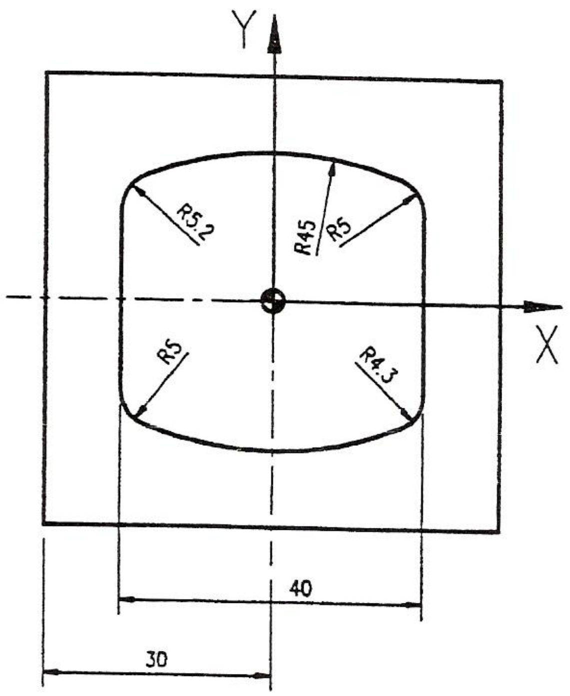


Figure 16 - Examples of Component with Zero Point in Workpiece Centre

For turning operations the X axis zero is normally positioned on the centreline of the spindle axis. The Z axis zero can be positioned on the front face of the workpiece or on the face of the workpiece or on the face of the chuck. See Figure 17.

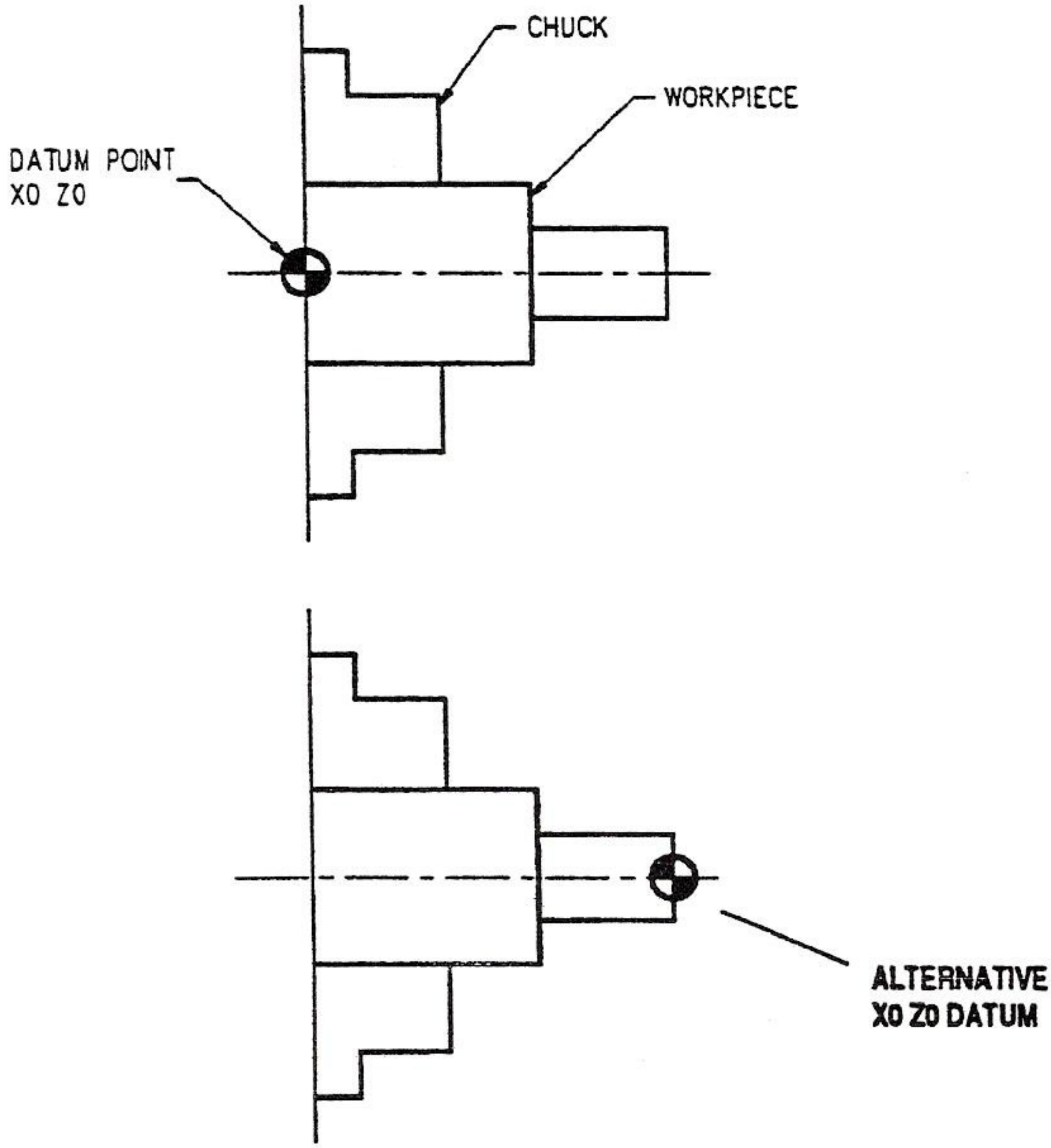


Figure 17 - X0 Z0 Positions for Turning Operations

Quadrantal use for Milling

In programming it is easier to imagine the tool is moving around the workpiece rather than the slideways moving. Figure 18 below shows how the + and - signs change for X and Y as you move around the four quadrants. X values to the right of the datum are positive (+) and those to the left are negative (-). Y values above the datum are positive (+) and those below are negative (-). Z values above the datum are positive and when the tool moves below the datum it is negative.

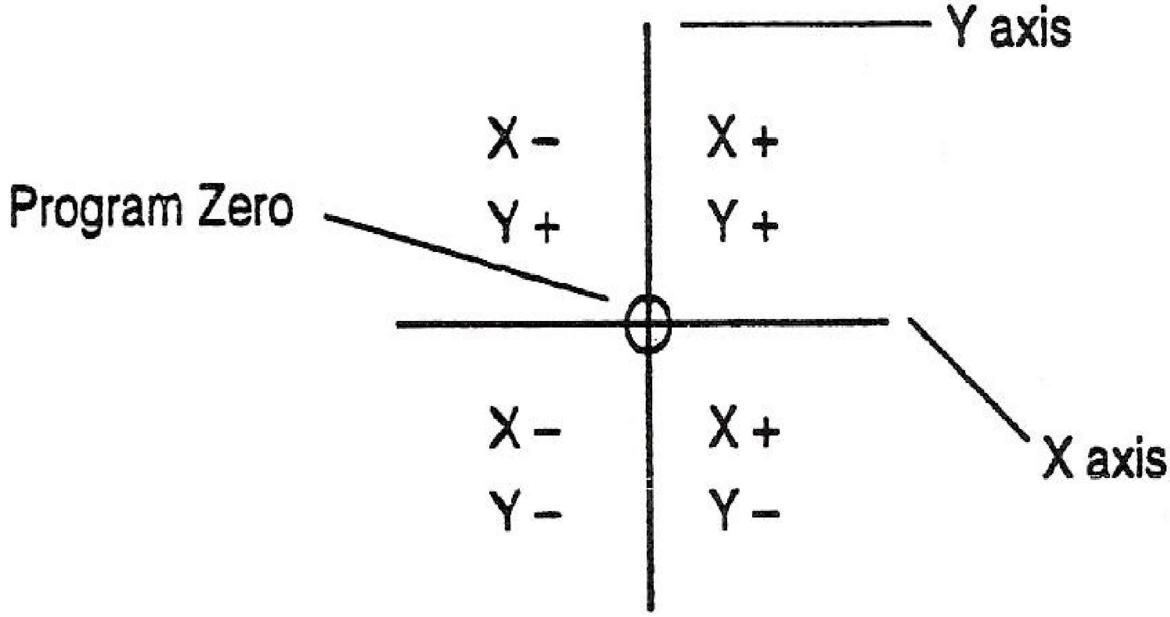


Figure 18 - Absolute Signs for Four Quadrants

In incremental programming X + tells the tool to move to the right from where it is now and X - to the left. Y + tells the tool to move up from its present position and Y - tells it to move down.

Z + tells the tool to move away from the workpiece and Z - tells the tool to move towards on into the workpiece.

Quadrantal use for Turning

The quadrant in use during programming depends on whether the machine has a front or rear mounted tool post and whether the program zero is on the chuck face or the workpiece front face.

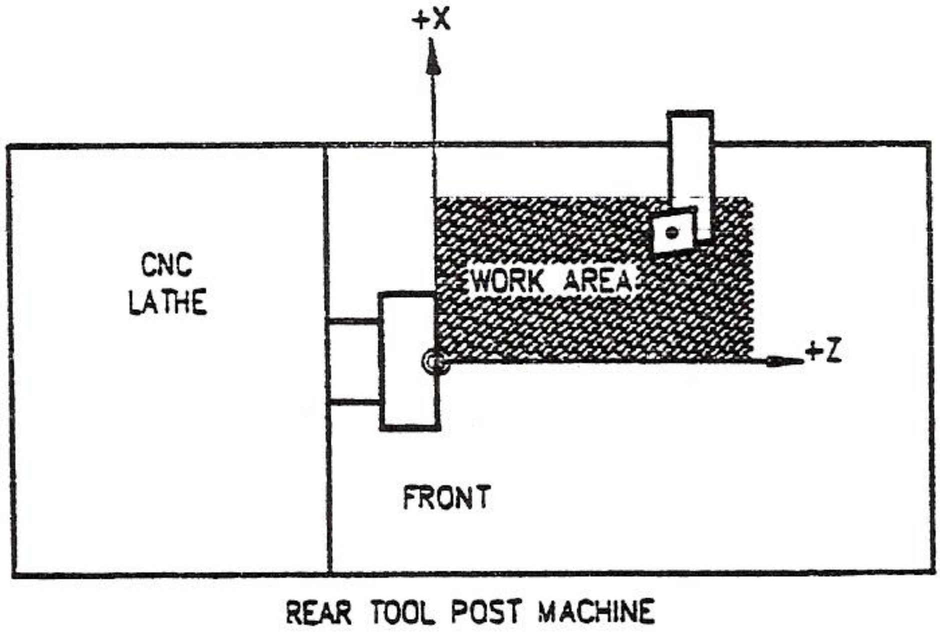


Figure 19 - Rear Tools Post Lathe

Absolute dimensions: If X0 is on the chuck centreline and Z0 is on the chuck face then:
Z movements towards the chuck are negative.

- Z movements away from the chuck are positive;
- X movements on the other side away from the operator are positive;
- X movements towards the operator are negative.

Incremental X movements towards the centreline are negative. Incremental X movements away from the centreline are positive.

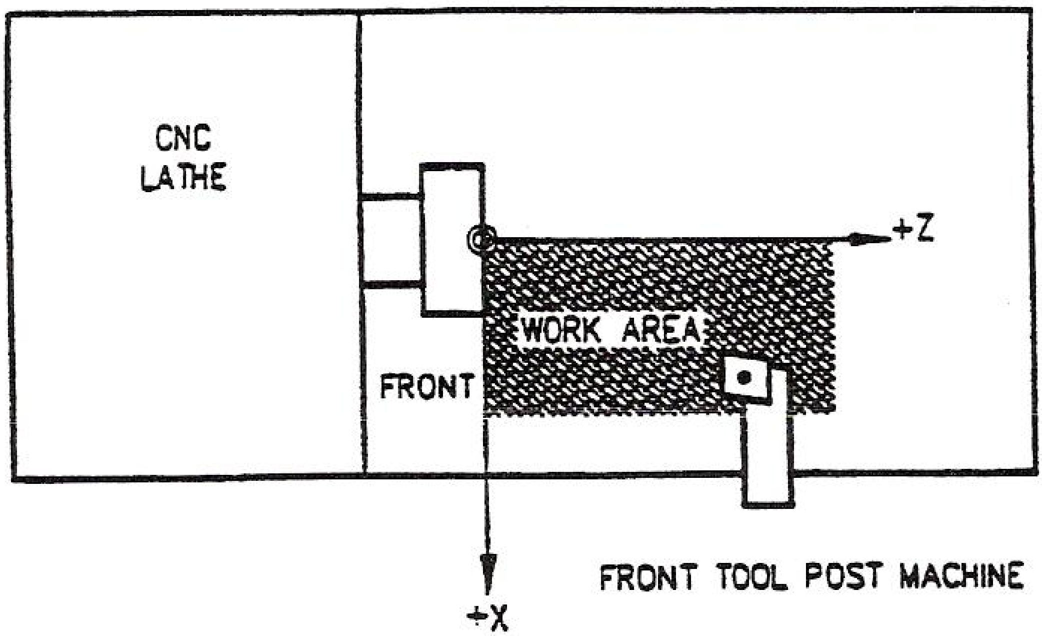


Figure 20 - Front Tool Post Lathe

Front mounted tool post lathe ref. Figure 20.

Absolute dimensions:

- Z movements towards the chuck are negative;
- Z movements away from the chuck are positive;
- X movements towards the operator on the side of the centreline nearest the operator are positive;
- X movements away from the operator are negative.

Incremental X movements towards the centreline are negative. Incremental X movements away from the centreline are positive.

Definition of Points on a Surface

The same identification letters (X, Y, Z) can be used for defining any point on a workpiece by placing the three axes on the part drawing as shown below:

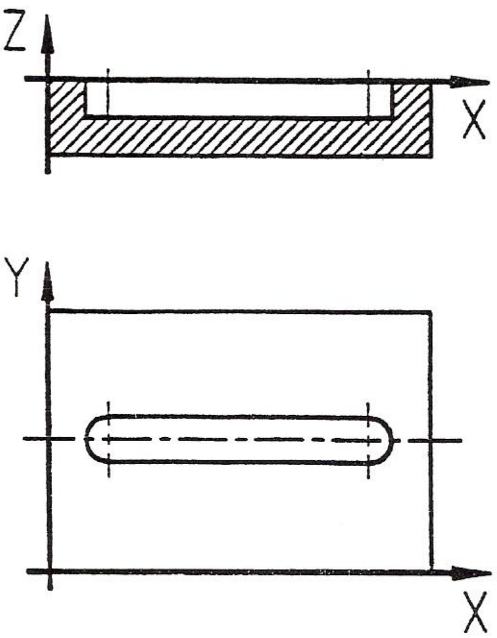


Figure 21 - Definition of Points on a Surface 1

All points on the workpiece surface are defined by X and Y values, while Z values denote the depth of tool infeed.

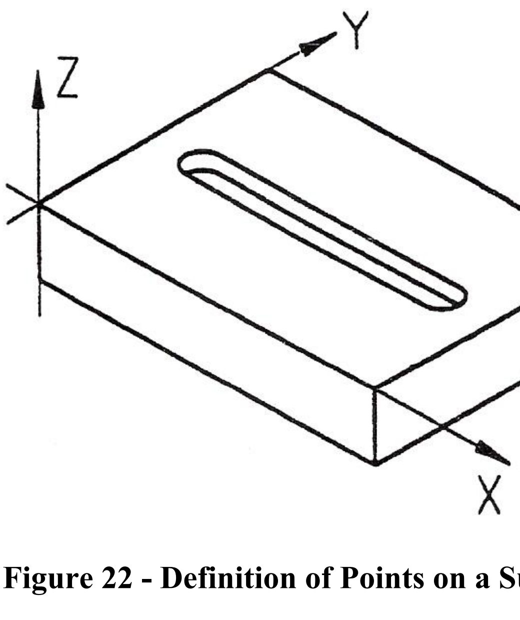


Figure 22 - Definition of Points on a Surface 2

Let's start by defining points on the workpiece surface and forget about the cutting depth (infeed) for the moment.

What we need is a two-dimensional coordinate system constituted by the X axis and the Y axis.

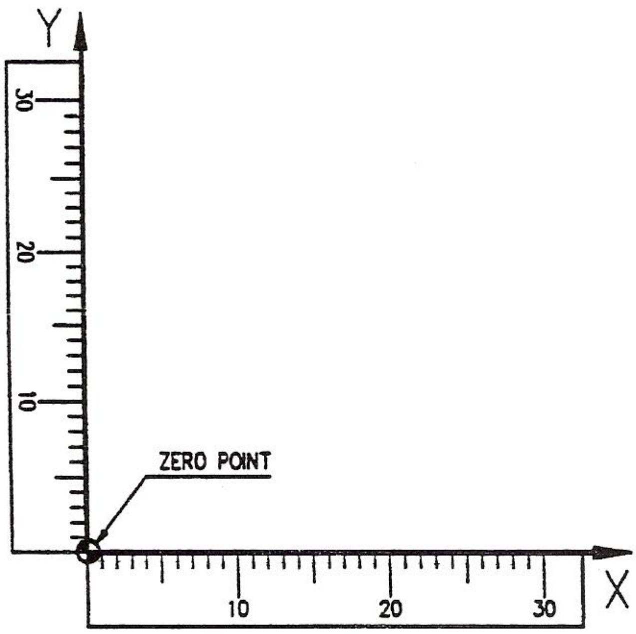


Figure 23 - Definition of Points on a Surface 3

As already stated the point of intersection of these two axes is called zero. The arrows indicate the direction of positive motions (X+ and Y+).

If a graduated scale is placed along each axis, we can define any point on a surface by means of its X and Y values.

All numerical values in the directions of the arrows carry a positive sign; all values in the opposite directions a negative sign.

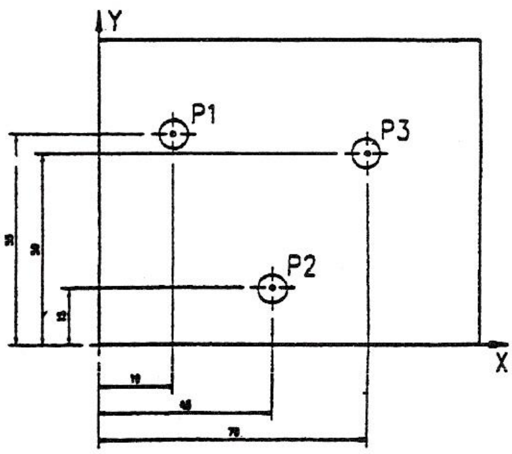
Example:

The points have the following coordinates:

P1: X19 Y55

P2: X45 Y15

P3: X70 Y50



Documentation

Program Sheets

Part programming is normally carried out by using pre-printed part program sheets. These sheets provide a neat and orderly way of setting out the program. They also provide permanent documentation of the job to be machined. An example of a part program sheet is shown in Figure 24. The sheets are laid out in such a way that the information can be written in by hand and then the sheet can be placed in a teletype and the program can be copy typed onto the second line and a punched tape is produced at the same time. See Figure 24.

Coordinate Sheets

Coordinate sheets are used as a supplement to the program sheet. All relevant coordinates are laid out on this sheet. A separate coordinate point will be specified for each point where the cutter needs to change direction. The sheet is useful as an aid in program proving and for locating and editing errors when they occur.

Operation Sheet

This sheet is intended as an aid to the operator. It itemises each operation in sequence and identifies the tools required for each operation and the tool settings.

PART PROGRAMMING SHEET
COMPONENT:
CO-ORD DRAWING:
M/C:
DATE:

DATUM INFORMATION

POSN.	N	X	Y	Z	I	J	K	G	M	T	F	S	OPERATION

SPECIAL INSTRUCTIONS

PREPARED	SHEET OF	DRG. NO.:	ISSUE:

Figure 24 - Programming Sheet

Part-Program Format

The machine control unit (MCU) controls the machine in response to coded commands which make up the part program. These commands are identified by a capital letter which is referred to as an address. The commands also contain numbers which follow the letters. The combination of the letter address and the numerical information is known as a word. Each line of a program is called a block. A block may contain a number of words.

The format in which the words are arranged within a block is known as the part program format. The order in which the words appear in a block may be fixed or variable.

Fixed Sequential Format

The instructions in a block are always given in the same sequence. All instructions must be given in each block including instructions that do not change from block to block e.g.

	G	X	Y	Z	F (Feed Rate)	S (Spindle Speed) RPM	T (Tool No.)
N0001	01	+20	+30	+100	200	1000	01
N0002	01	+20	+50	+100	200	1000	01

Table 1 - Fixed Sequential Format

Tab Sequential Format

The instructions in a block are always given in the same sequence as in the fixed sequential format. However if instructions remain unchanged in succeeding blocks the instructions need not be repeated but the tab character must be punched to ensure that the same number of tab characters appear in each block e.g.

	G	X	Y	Z	F (Feed Rate)	S (Spindle Speed) RPM	T (Tool No.)
N0001	01	20	30	100	200	1000	01
N0002	00		50				

Table 2 - Tab Sequential Format

Word Address

Each word is preceded by its letter address. This system enables instructions which remain unchanged from the preceding block to be omitted from successive blocks. This format is adopted by most CNC machine control units e.g.

N0001	G01	X+20	Y+20	Z+100	F1000	S1000
N0002		X+50				
N0003			Y+50			
N0004		X–10				
N0005			Y+90			

Table 3 - Word Address

The address format refers to the form in which the words must take for example:

N4: 4 digits after N

G2: preparatory function 'G' followed by 2 digits

X43: X-dimension followed by 4 digits before the decimal point and three digits following the decimal point.

There is usually a maximum of 7 digits that can be written for X, Y and Z axes. However not every CNC control will allow a decimal point to be used and there are two other methods of representing dimensions as follows.

Leading Zero Suppression

This is the removal or suppression of any zeros on the left of the number when written out in the seven digit format, e.g. the dimension 130.75 mm written in leading zero suppression format (7 digit entry).

7 digit entry 4 before, 3 after = 0130 . 750
decimal point position of imaginary decimal point

with Leading zero suppression = 130750

:. For a machine control of this type an X dimension of 130.75 would be written as

X130750

Trailing Zero Suppression

With this system any zeros to the right of the decimal point are removed or suppressed. So an X dimension of 130.75 would become X013075 for a control which accepts trailing zero suppression.

The majority of modem CNC machines allow the use of decimal point programming i.e. you just write the dimension up to a maximum of 7 digits.

Part Program Code Listings

- N: Block sequence number address. Blocks are often inserted in steps of 5 to allow for blocks to be inserted if accidentally omitted.
- X, Y, Z: These addresses signify the X, V, Z axes movements.
- I, J, K: These addresses are used for programming circular moves
- T: Tool numbers address. The programmer must assign a number e.g. T01 to each tool used in the program.
- S: Spindle speed address. The letter is followed by numbers which indicate the spindle speed.
- F: Feed rate address - followed by numbers as in spindle speed.

M: Miscellaneous Functions

These are machine management functions such as starting and stopping. The M letter is followed by 2 digits. The common standardised M functions are as follows:

- M00: program stop
- M02: end of program
- M30: end of program with rewind to program start
- M08: flood coolant on
- M09: coolant off
- M06: tool change
- M03: spindle on clockwise
- M04: spindle on anti-clockwise
- M05: spindle off
- M07: mist coolant on

G: Preparatory Function Addresses

G functions are preparatory functions used to change the mode of movement of the machine, such as rapid slide movement, circular movement, controlled feed rate, absolute or incremental movement etc. The code consists of the letter G followed two digits. Some common standardised G functions are as follows:

- G00: rapid slide movement
- G01: linear interpolation
- G02: circular interpolation clockwise
- G03: circular interpolation anti-clockwise
- G04: dwell
- G40: cancel cutter radius compensation
- G41: cutter compensation left
- G42: cutter compensation right
- G70: inch units
- G71: metric units
- G90: absolute coordinates
- G91: incremental coordinates

% Program start character. Used at the beginning of a program to indicate the start.

Some command functions are modal which means that the command remains in effect until cancelled or superseded by a command of the same type.

Control of Slide Movement

There are two basic types of slide movement which are usually applied on CNC machines as follows:

- point-to-point control
- continuous path

Point-to-Point Control is used where the machine slide is required to reach a particular point in the shortest possible time. NO machining takes place while the slide is moving.

Point-to-Point Control would be suitable for machines which are only used for drilling or boring i.e. no machining is taking place as the slide is moving from the centre line of one hole to the coordinate position of the next. The path the tool or slide takes in getting from one point to the next is unimportant. When the machine is operating in point-to-point mode the G00 preparatory function is operational.

Continuous Path Control can take the form of Linear or Circular Interpolation. Interpolation is the process of joining up programmed points to generate a smooth path.

If the segments joining the points are straight lines the process is called Linear Interpolation. If the segments joining the points are arcs of circles the process is called Circular Interpolation. Any machining operation where the tool is cutting as the slides are moving in a controlled manner requires continuous path control.

Linear Interpolation

Linear interpolation means machining in a straight line. This can be either horizontal, vertical or at an angle in any direction. The G01 code is entered into the control with a feed rate value e.g. N2, G01, X200, Y140, F400.

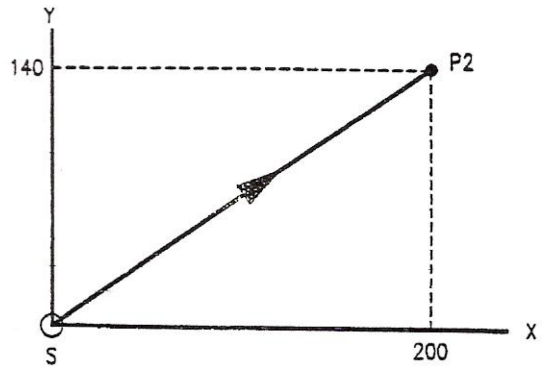


Figure 25 - Linear Interpolation

In block No.2 the tool travels along a straight line from current position (S) to P2 at a feed rate of 400 mm/min. (see Figure 25).

Circular interpolation refers to the programming of circular arcs up to a complete circle. In order to program an arc with input in Cartesian coordinates the data input is guided by the control upon inputting the command G02 or G03. The control will request:

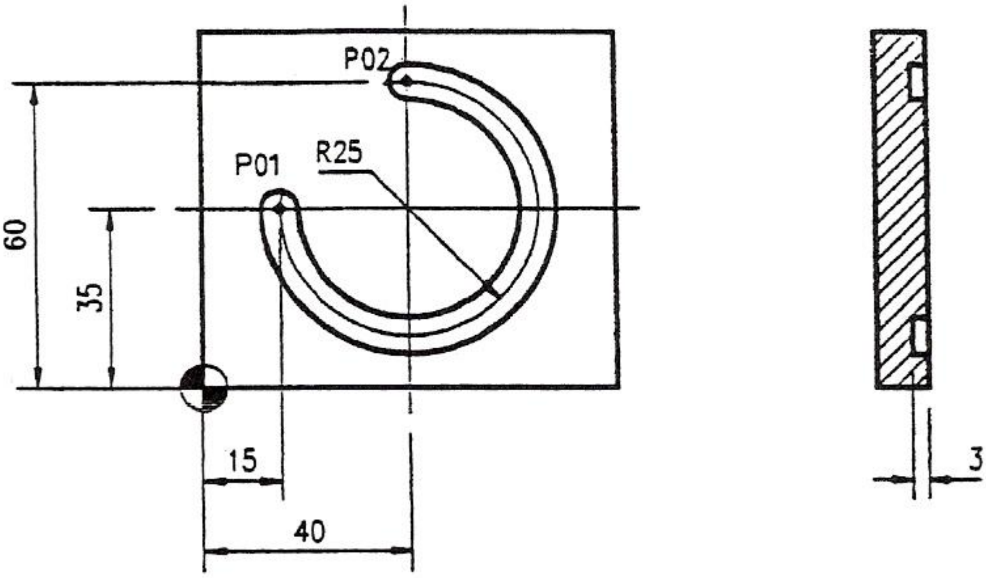
- X: 1st coordinate of target point
- Y: 2nd coordinate of target point
- I: 1st coordinate of circle centre
- J: 2nd coordinate of circle centre

I is used to specify the centre of the arc in the X direction

J is used to specify the centre of the arc in the Y direction

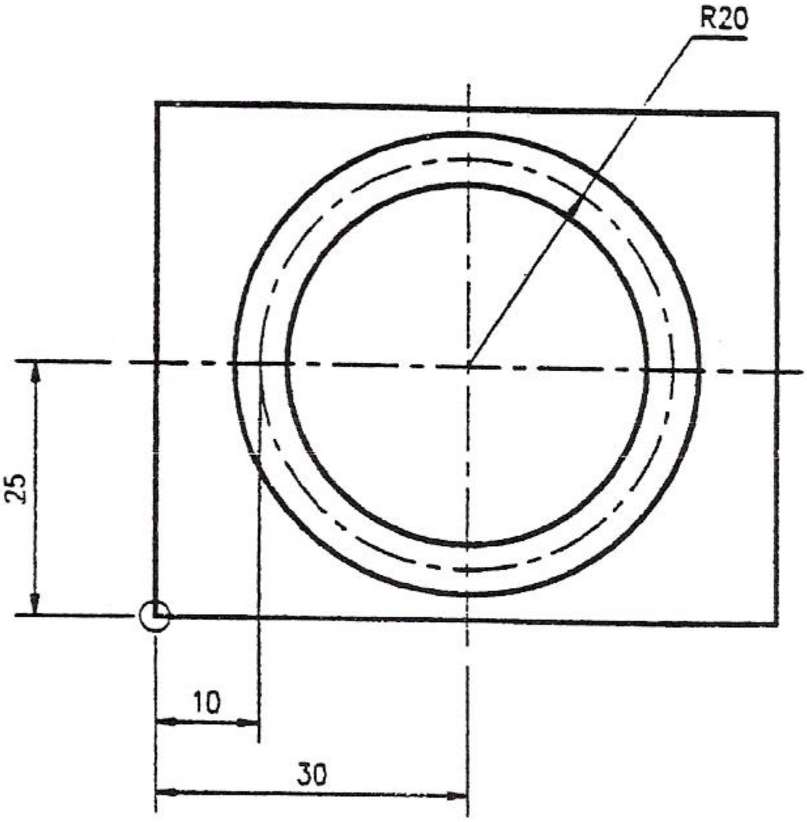
K is used to specify the centre of the arc in the Z direction

Note: Some machine control units require the I, J and K values to be given incrementally.



N	G	X	Y	Z	I	J	S	F	M
N1	G00	X0	Y0	Z + 100			S500	F100	M03
N2	G00	X+15	Y+35						
N3	G00			Z + 3					M07
N4	G01			Z - 3				F50	
N5	G03	X + 40	Y + 60		I + 25	JO		F100	

Figure 26 - Example of G03 Command in use



N	G	X	Y	Z	I	J	S	F	M
N1	G00	X0	Y0	Z + 100			S500	F100	M03
N2	G00	X+10	Y+25						
N3	G00			Z + 3					M07
N4	G01			Z – 5				F50	
N5	G02	X+10	Y+25		I + 20	J0		F100	
N6	G00			Z + 3					
N7	G00			Z + 100					
N8	G00	X0	Y0						

Figure 27 - Example of G02 Command in use

Geometry and Trigonometry

Important Geometrical Theorems

Chords and Circles: If two chords of a circle intersect, either within or without the circle, the product of the two segments or one chord is equal to the product of the two segments of the other chord.

Figure 28 shows at (a) the chords intersecting inside the circle, at (b) the chords intersecting outside the circle, and at (c) the special case of one chord just touching a circle so that the chord is a tangent.

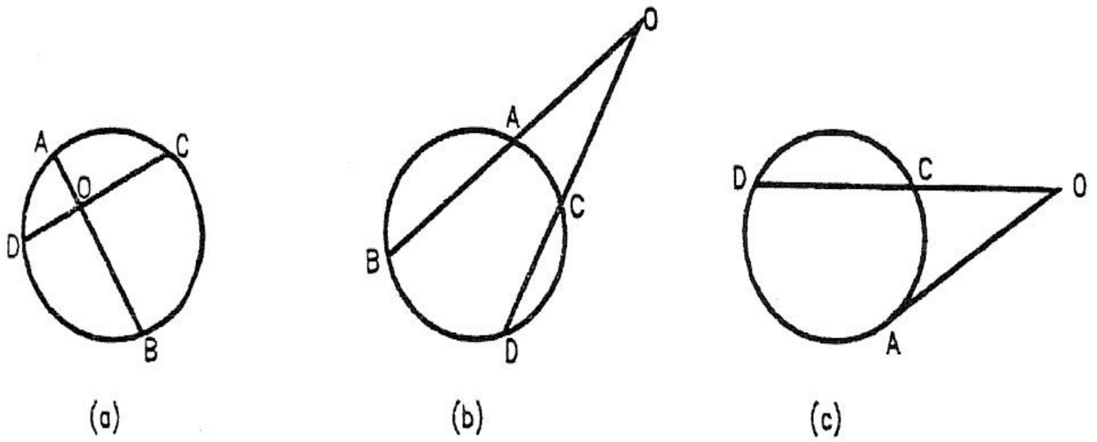


Figure 28 - Important Geometric Theorems

In Figure 28(a) and (b): OA x OB = OC x OD

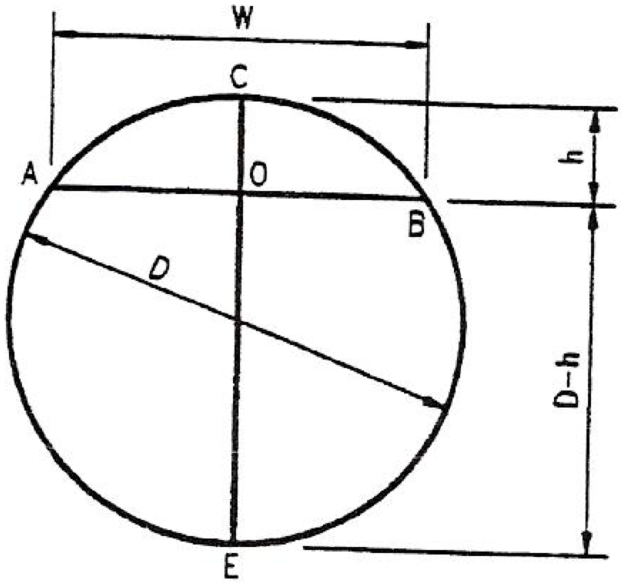
In Figure 28(c): $(OA)^2$ = OC x OD

An important application occurs when one chord is the diameter of the circle, the other chord intersecting the diameter at right angles with the circle, as shown in Figure 26.

The chord AB is bisected and AO = OB.

Let D = diameter of circle,

W = length of chord AB, and h = height of segment.



	AO x OB	= OE x OC
:.	W/2 x W/2	= (D – h) h
and	$W^2/4$	= Dh - h^2
:.	W^2	= 4 (Dh - h^2)
and	W	= 2$\sqrt{}$ (Dh - h^2)

If D is required, using $W^2/4$	= Dh - h^2
Dh	= $W^2/4 + h^2$
D	= $W^2/4h + h$

Intersecting Straight Lines

If two straight lines intersect, vertically opposite angles are equal. In Figure 29(a) angles A are equal, and angles B are equal.

If a straight line intersects two parallel straight lines, corresponding angles on the same side of the line are equal. In Figure 29(b), angles A are equal, and angles B are equal.

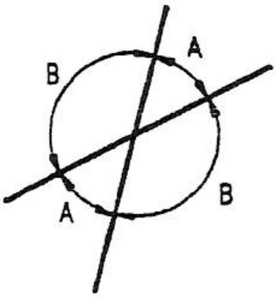


Fig (a)

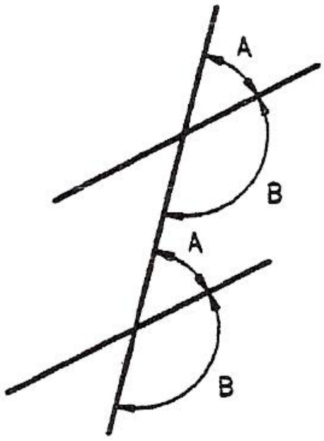


Fig (b)

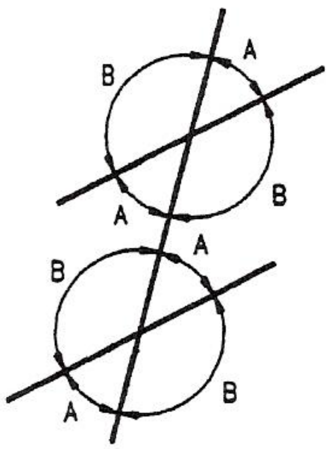


Fig (c)

Figure 29 - Intersecting Straight Lines

If the above two theorems are combined, then in Figure 29(c) all four angles A are equal and all four angles B are equal.

Triangles

A triangle is a plane area bounded by three straight lines. Triangles can be classified into three types. Equilateral triangles have sides of equal length; isosceles triangles have two sides of equal length, whilst scalene triangles have sides of different lengths. A triangle which contains a right-angle is termed a right-angled triangle. Such a triangle would be isosceles if the sides containing the right-angle were of equal length.

Figure 30 shows a triangle ABC, with AC continued to D. and CE drawn parallel to AB. By use of the theorems stated earlier:

<BCE = <ABC

and <ECD = <BAC

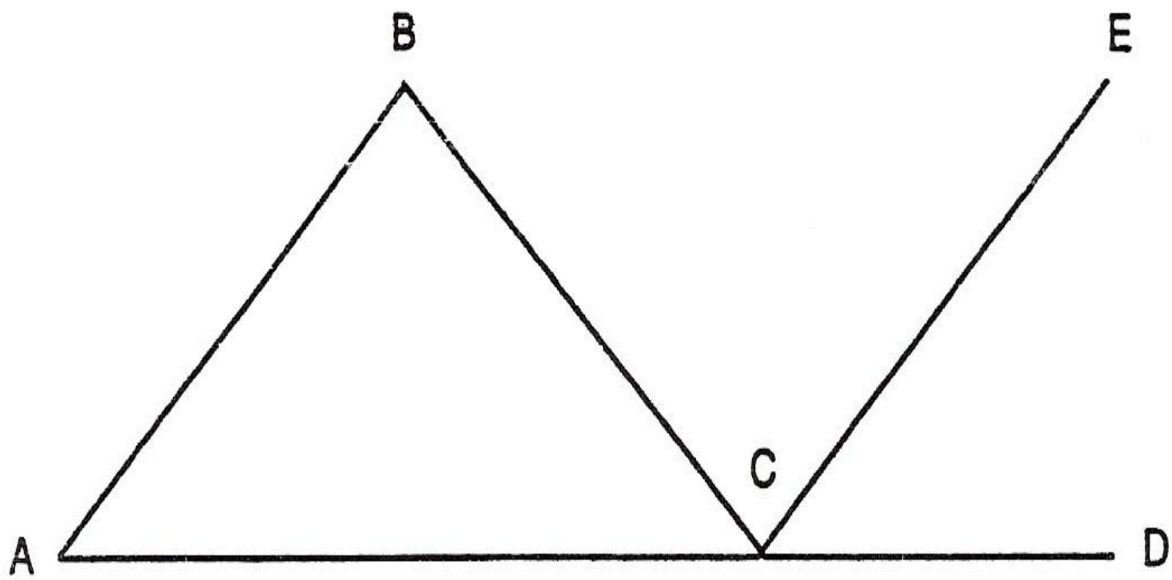


Figure 30 - Triangles

The sum of the three angles in the triangle therefore is the same as

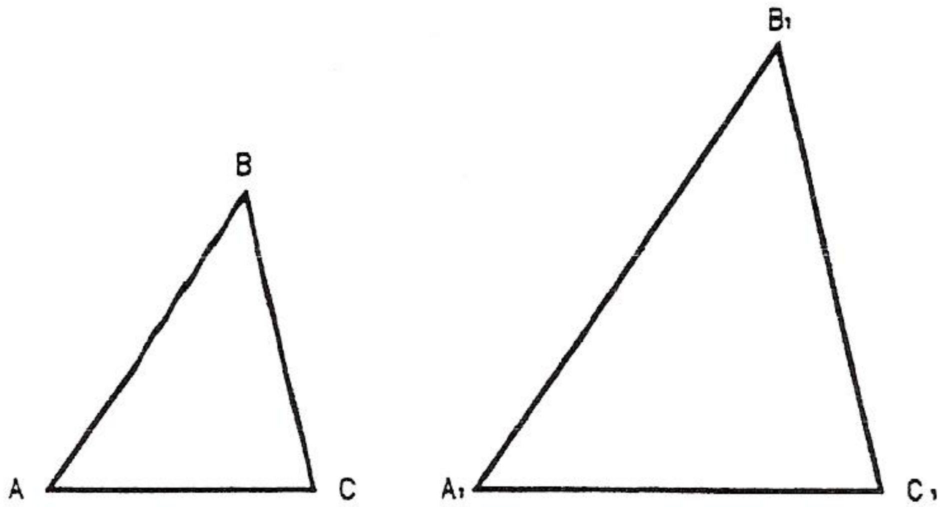
<BCA + <BCE + <ECD = <ACD

ACD is a straight line of 180° (i.e. two right-angles); hence the sum of the three angles in a triangle is 180°.

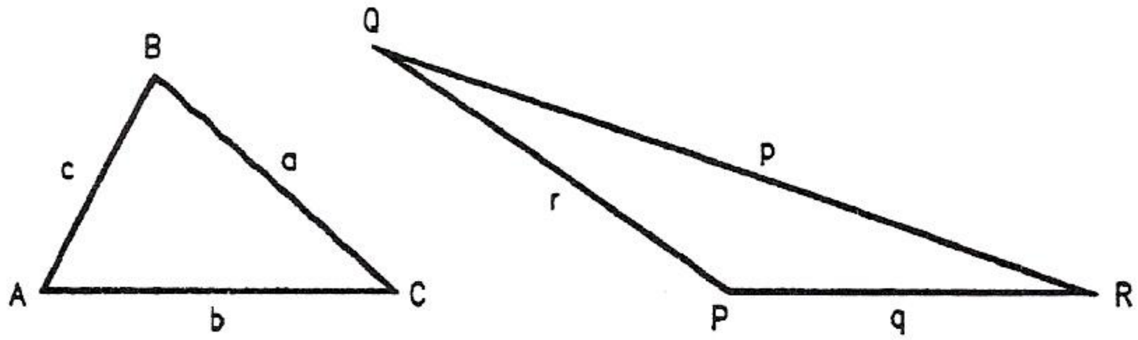
Similar triangles are triangles which have the three angles in one triangle equal to the three angles of the other, whereas congruent triangles are equal in all respects. Consequently, the areas of congruent triangles are equal, but the areas of similar triangles are not. The corresponding sides of similar triangles are proportional to each other.

Two similar triangles are shown below, for which:

$$\frac{AB}{A_1 B_1} = \frac{BC}{B_1 C_1} = \frac{CA}{C_1 A_1}$$



Trigonometry literally means the measurement of triangles. In order that particular angles and/or sides may be indicated precisely, it is conventional practice to letter the angles with capital letters and the sides with small letters. Furthermore, side a is opposite angle A, side y opposite angle Y and so on. This standard convention is illustrated in the figure below.



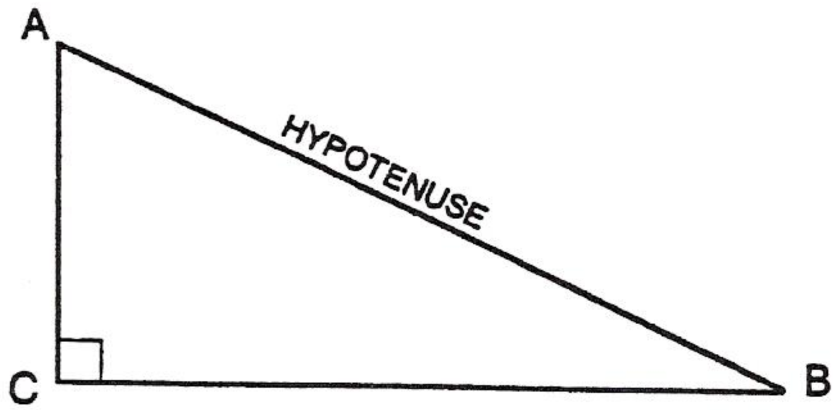
Pythagoras Theorem

This theorem can be used to find the length of one side of a right angled triangle if the lengths of the other two sides are known.

The theorem states that if we take the lengths of the two smaller sides of a right angled triangle, square them and then add them together the result will be equal to the length of the longest side (the hypotenuse) squared.

In the triangle ABC below this can be expressed mathematically as follows:

$$AB^2 = AC^2 + BC^2$$



:. In order to find the length of A B we can say:

$$AB = \sqrt{AC^2 + BC^2}$$

By transposing the formula we can find the length of any side if the lengths of the other two are known.

The Theorem of Pythagoras states that the square on the hypotenuse of a right angle is equal to the sum of the squares on the other two sides.

Example: In the triangle ABC shown, the length AC = 15 and CB = 20. Calculate the length of A B (hypotenuse):

$$AB^2 = AC^2 + CB^2$$

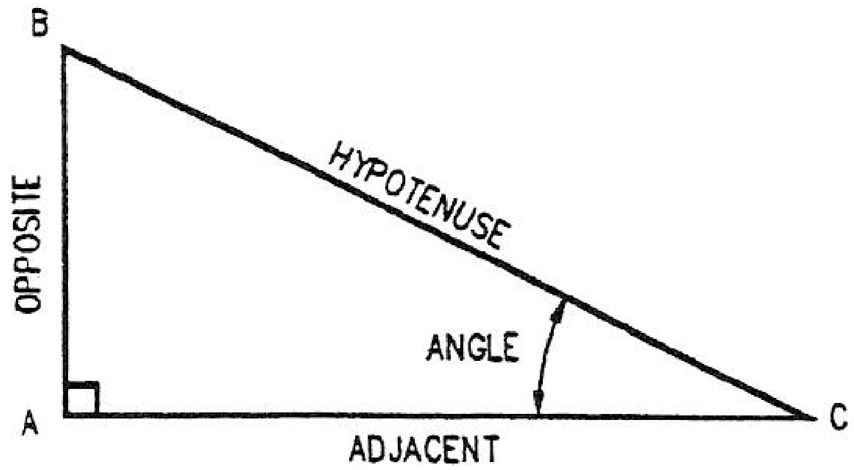
$$AB^2 = 225 + 400$$

$$AB = \sqrt{625}$$

$$AB = 25$$

Trigonometry

The ratios of the lengths of the sides in similar right-angled triangles are always the same, provided the angles in each case are the same. They ate called trigonometrical ratios. In order to be able to differentiate which sides and which angles are being considered a special notation and wording is used as follows:



Trigonometrical Ratios

sine of angle $= \dfrac{\text{opposite}}{\text{hypotenuse}}$

cosine of angle $= \dfrac{\text{adjacent}}{\text{hypotenuse}}$

tangent of angle $= \dfrac{\text{opposite}}{\text{adjacent}}$

The use of trig ratios is limited to the solution of right angled triangles only.

Solution of Right-Angled Triangles

In solving problems involving the use of trigonometry a useful method of approach is to ask two questions:

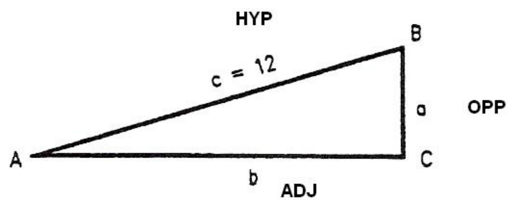
(a) is there a right-angled triangle, if so;

(b) by what ratio must I multiply the known side to obtain the unknown side i.e. what it unknown

$$\text{ratio} = \frac{\text{unknown}}{\text{known side}}$$

Example:

In a right-angled triangle, the hypotenuse is of length 12 mm and one angle is 73°. What is the length of the shortest side?



One angle is 90°, another is 73°, since the angles in a triangle add up to 180°, the other angle is: 180° - (90° + 73°) = 180° - 168° = 17°.

The shortest side will be opposite the smallest angle; it will be side a in the figure above. The required multiplier of the known side is:

$$\text{ratio} = \frac{\text{unknown}}{\text{known side}} = \frac{\text{side opposite A}}{\text{hypotenuse}} = \sin A = \sin 17°$$

The answer is:

known side x sin 17° = 12 sin 17° = 12 x 0.2924 (from Trig. tables)

Answer: = 3.5088

Sine Rule

The sine rule can be used to solve problems of triangles that are not right angled triangles. The sine rule states:

$$\frac{a}{\text{Sin A}} = \frac{b}{\text{Sin B}} = \frac{c}{\text{Sin C}}$$

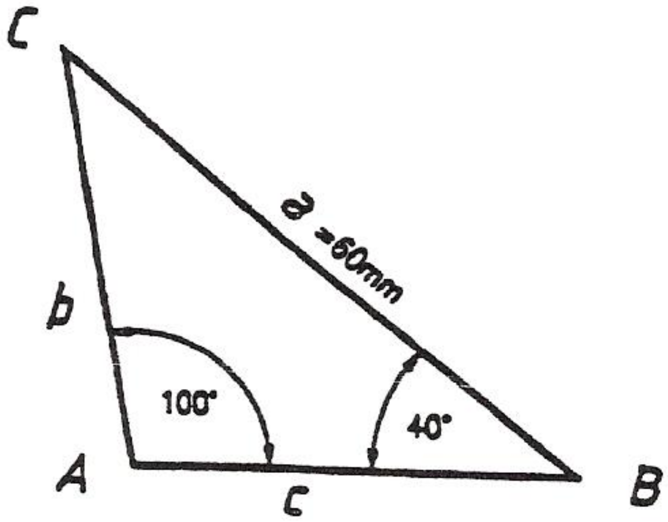
a is the line opposite angle A

b is the line opposite angle B

c is the line opposite angle C

Any two terms can be used together:

i.e. $\frac{a}{\text{Sin A}} = \frac{c}{\text{Sin C}}$ etc.



To find the length AB in the triangle above proceed as follows:

angle C = 180° - (100° + 40°) = 40°

$$\frac{a}{\text{Sin A}} = \frac{c}{\text{Sin C}}$$ from Sine Rule

$$\frac{BC}{\text{Sin A}} = \frac{AB}{\text{Sin C}}$$ sine rule applied

$$AB = \frac{\text{Sin C x BC}}{\text{Sin A}}$$ re-arranged for AB

$$= \frac{40 \text{ x } 60}{\text{Sin } 100}$$ substitute known values

$$= \frac{0.6428 \text{ x } 60}{.9848}$$ from sine tables

$$= 39.163 \text{ mm}$$

The Cosine Rule

The cosine rule states that in any triangle whether it is right-angled or not, the square of a particular side is the sum of the square of the other two sides minus twice the product of the other two sides and the cosine of the angle opposite the particular side. Written as an equation this becomes:

$$a^2 = b^2 + c^2 - 2bc \cos A$$

or

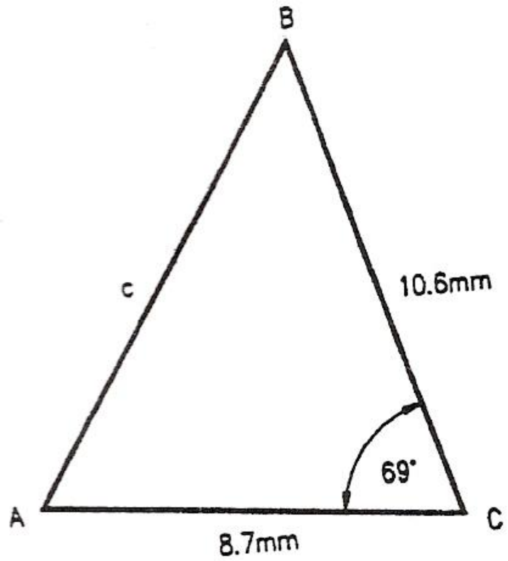
$$b^2 = a^2 + c^2 - 2ac \cos B$$

or

$$c^2 = a^2 + b^2 - 2ab \cos C$$

This form of the equation is used to find the third side of a non right-angled triangle when the other two sides and the included angle is known.

Example:



$$c^2 = a^2 + b^2 - 2bc \cos C$$

$$= 10.6^2 + 8.7^2 - 2(10.6)\,(8.7) \cos 69°$$

$$c = \sqrt{[10.6^2 + 8.7^2 - 2(10.6)\,(8.7) \cos 69°]}$$

$$= 11.0 \text{ mm}$$

Data Input/Storage and Program Proving

Data Input/Storage and Program Proving

The main methods which can be used to input data into a CNC control unit are:

- Punched tape and tape reader
- Magnetic tape
- Magnetic disc
- Host computer
- Manual data input

Punched Tape: This is a low cost method of data input. The tape is available in rolls or can be fan folded.

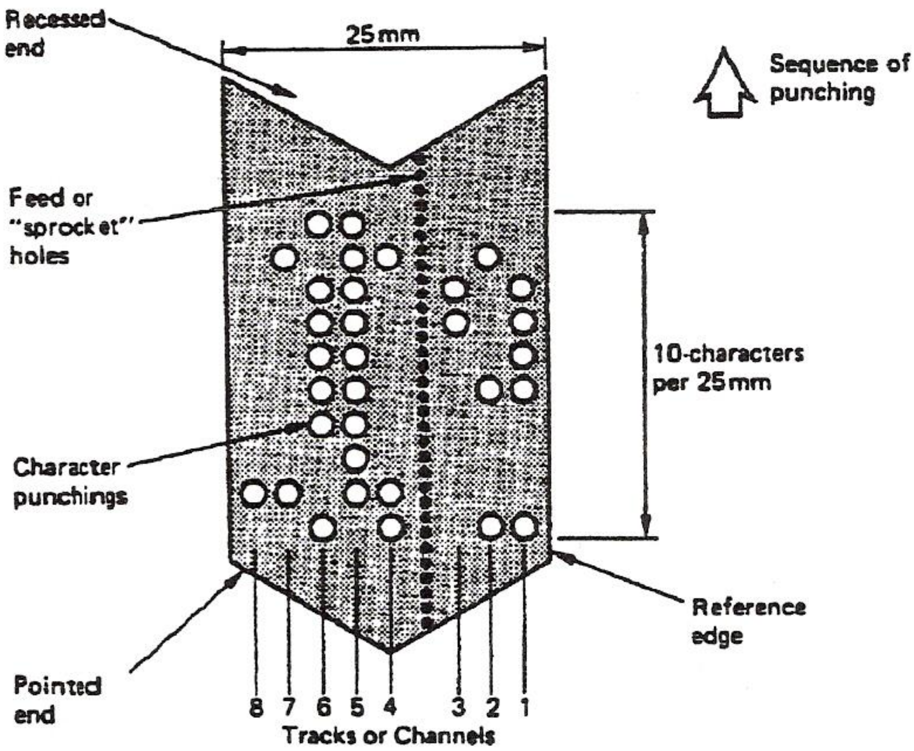


Figure 31 - Section of Punched Tape

Punched tape is available made from paper, polyester, paper/polyester laminates, or polyester/aluminium foil laminates.

Punched tape has certain advantages such as: shop floor suited i.e. insensitive to magnetic fields or oil contamination. It can also be read visually by an experienced person. Tape damage is also immediately noticeable.

The disadvantages of punched tape are the sprocket holes tend to wear or tear with use. The storage density is low and the tape is not erasable and re-usable.

Tape readers: The function of the tape reader is to detect the presence and position of holes in the tape. There are three different types of tape reader:

1. Pneumatic
2. Mechanical
3. Photo-electric

Magnetic Tapes and Discs

Magnetic tape is a cheap and convenient method of storing large volumes of data in a small space. Tape cassettes are easy to handle and store. It can also be erased and re-written as required. The tape recorder fulfils the task of both the tape punch and tape reader since it can record and playback.

However it is impossible for an operator to know if a tape contains any information or not by visual inspection. Magnetic tapes can also be erased accidentally in the presence of a magnetic field.

Magnetic Discs

The data transfer rate is faster for a disc than magnetic tape. The access time, to stored data is faster because the disc is a random access device. That is any single piece of data recorded on the disc can be accessed as easily and as quickly as any other.

Host Computer

The process of transferring part programs from a host computer into the memory of a CNC machine tool is called Direct Numerical Control (DNC). A number of machine tools of different types can be involved.

Manual Data Input (MDI)

This is a term used to describe the method of entering data into the machine control unit using the console keypad. The entering of complete programs other than relatively short ones is not practical as the machine is idle while data is being entered on most machines. The most common use of MDI is for editing programs and for machine setup. This has the advantage that once edited the new program can be saved or re-punched automatically by outputting to the tape punch.

Program Proving

Before a program is used it should be 'proved' to check that the desired operation will take place. The consequences of not proving a part program range from damage to the components and tooling, catastrophic damage to the machine tool or serious injury to the operator or other observers. The following methods can be used to prove a program.

Dry Run

This involves running the program in automatic mode without the component installed in the chuck or on the machine table. The purpose is to verify the tool path.

Plotter

A relatively simple way of checking the programmed component profile is to substitute a pen tip for the cutting tool. For a milling operation a two dimensional trace of the cutter path may be produced on paper by placing a board on top of the machine table.

Single Step or Stepping

This involves the operation "stepping" through the program line by line and actually cutting a component one step at a time. After each step the next movement is carefully checked before execution.

Computer Graphics

The program is fed into a computer using the keyboard, floppy disc or tape. The computer graphics are then used to simulate a test run. The correct sized blank appears on the screen and using animated tool movements it is machined to final shape and size according to the program data.

CNC Setting & Operation

Workpiece and Tool Setting

The machine datum is the point within the machine's range of movement from which the machine makes its programmed dimensional moves. It is an exact point on each axis that the machine can find even after power loss - this is the point the machine slides move to when you reference the machine. It is often called the zero datum or the machine reference point. Three axis machines usually have the Z axis datum position as the spindle fully retracted. When a workpiece is clamped on the m/c table the workpiece datum and the machine datum will not normally coincide. In order to relate the two a floating zero facility is provided. This means that the operator can arbitrarily designate as zero any point on each axes within the range of slide displacement.

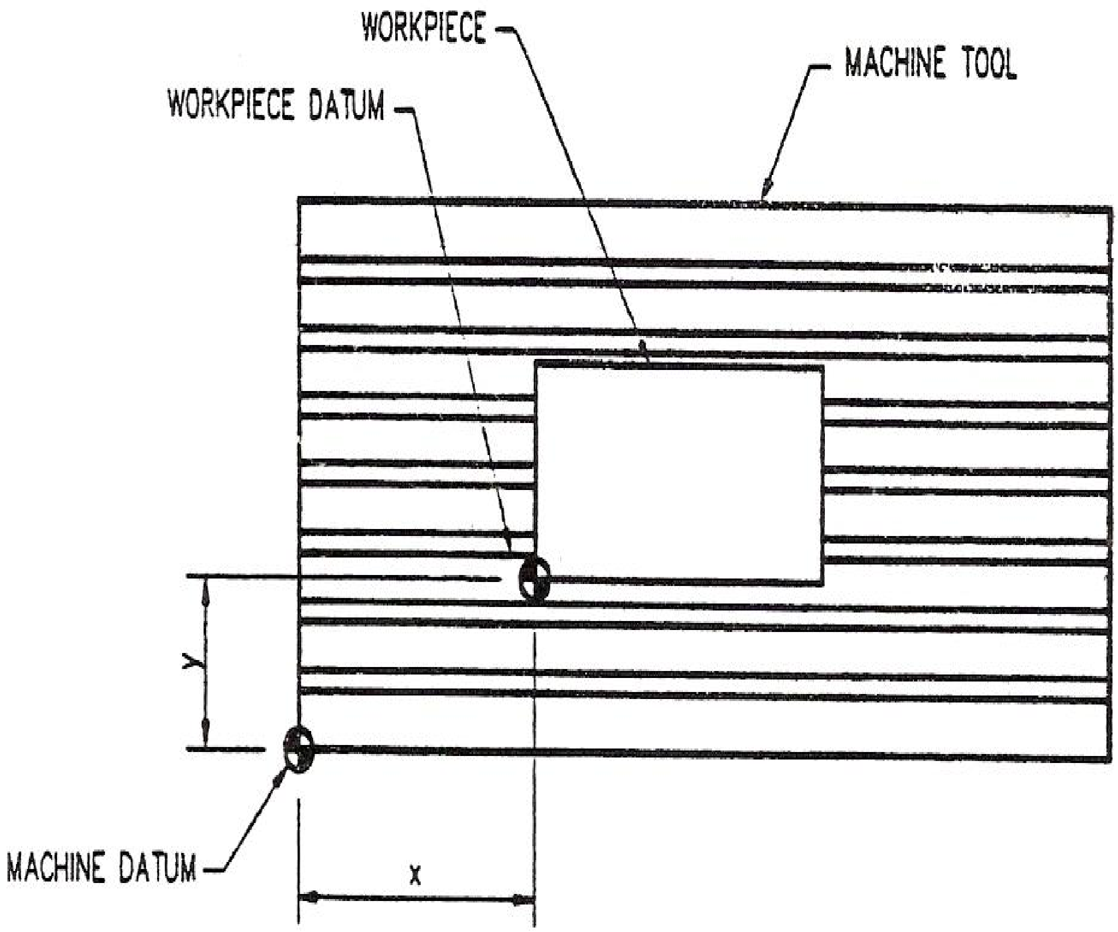


Figure 32 - Machine Datum and Workpiece Datum

Setting Axis Datums

On CNC machines employing fully floating datum facilities it is common to position the workpiece or workholding device on the machine table, for convenience. The tool, or setting probe, is then jogged manually to touch the component in each axis in turn. With the tool or setting probe in the correct setting position (component datum position), one of two actions may be taken. The choice will depend on the machine tool.

The first action involves setting the relevant axis register to zero, by entering at the console. A button marked 'axis zero' may also have to be depressed to confirm the action. Thereafter, the machine zero position is assumed to be that point. All subsequently programmed positional moves will be made with reference to this zero point.

The second action is perhaps more prevalent on CNC turning centres. At start-up, many turning centres send their slides to a known position normally at the extremes of their axis movements. When at this position, indicated by limit switches detecting the limits of travel, the control unit 'knows' that the slides are at known dimensions from the machine datum point. This may be, for example, 800 mm from the back face of the chuck in the Z-axis, and 500 mm from the centre-line of the spindle in the X-axis. The exact dimensions will, of course, vary from machine to machine. The tool can then be jogged to touch on the component as previously described. The dimensions showing on the axis read-outs are then subtracted from the known 'limits of travel' dimensions. The resulting values are then entered into axis offset registers within the CNC control unit. In a roundabout way this creates the datum position for the tool tip. The datum positions are not initialised to X0 and Z0 however; they are computed (within the control unit) from the known 'limits of travel' dimensions, and the values contained in the axis offset registers.

Regardless of the actual procedure, the part programmer programs the part from a datum point relating to the component. The positioning of the component at the machine, and a similar manoeuvre to those described above and then reconciles the position of the component to the datum positions set at the machine tool. Quite obviously, the correct procedure for each individual machine must be followed. To this end, the operating/setting manual for the individual machine must be thoroughly studied.

Workholding Devices

Workholding devices can be broadly divided into two categories:

1. those which hold rotating workpieces and;

2. those which hold fixed workpieces on machines which use rotating cutting tables.

Category 1

Chucks

Chucks may be manually operated or power assisted. It is more common for power-assisted chucks to be used for CNC work where the emphasis is on speed of loading and unloading. Power assistance may be by pneumatic or hydraulic operation, the latter being used for larger applications. Greater gripping power is obtained using hydraulics since the hydraulic fluid is essentially incompressible. Such chucks have a jaw movement of only a few millimetres and so must be initially set for the diameter of workpiece being machined. Automatic chucks are normally operated by a foot pedal but a 'chuck enable' button, on the operating console, has to be depressed before the chuck can be released. This is a safety device to prevent accidental mis-operation of the chuck. Bar feeders may be employed where many identical components are required, and the raw material can be obtained in bar stock form.

When loading components into chucks, it is desirable to locate the component against the back face of the chuck, or a suitably designed spacer. This ensures positive location to resist the applied cutting forces. Where the component is not backed up in this way, the operation is relying on the frictional location provided by the chuck jaws alone.

Collets

Collets or collet chucks are quick-acting fixed-diameter work-holding devices. They are designed for holding close diameter round components. If components are to be machined from lengths of bright bar, the bar stock can be fed through the centre of the collet onto a fixed stop. Collets offer quick, positive and constant re-chucking and afford a wide area of contact for gripping. Because they are of fixed diameter, a set is required to accommodate different-diameter workpieces. Collets may have jaws of different forms to accommodate different sections of components.

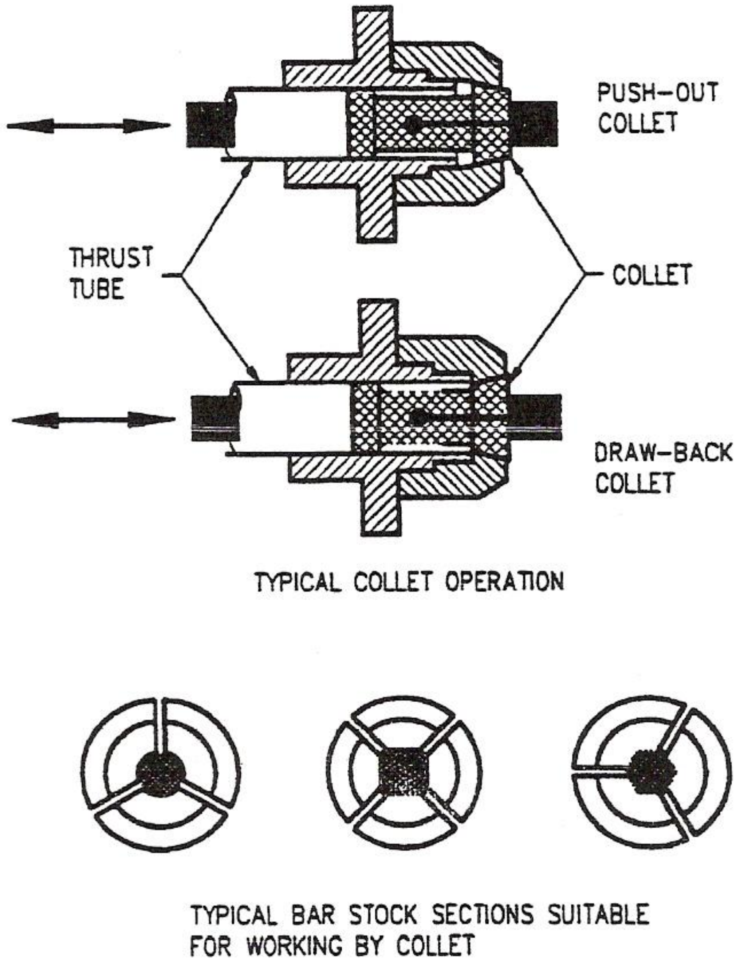


Figure 33 - Collet Details

Collets operate on the principle of moving along a taper. There are different designs in that the taper may be pushed (as in a push-out collet), or pulled (as in a draw-back collet). They may be manually operated or power assisted.

Bar feeders are useful additions to a collet set up for speed of component feeding. The bar stock is fed through the collet automatically for each new component.

Collet details are illustrated in Figure 33.

Turning Fixtures

Where the component is unusually large or irregular (as in the case of castings or forgings), special-purpose turning fixtures may have to be designed. The fixtures will then be mounted directly onto the spindle of the machine itself, or on a faceplate mounted on the spindle. It would be unusual to employ faceplates alone on CNC machines since the setting-up time would be prohibitively long. Robotic devices can easily cope with loading and unloading purpose-designed fixtures employing power operated clamping devices.

Category 2

Workholding Devices on Machines Employing Rotating Cutters

In all cases the workholding device should be positioned at the centre of the worktable in the X-axis on machining centres and milling machines. This ensures the greatest support and minimises any static deflection of the machine table due to the weight of the component and the workholding device. Similarly, positioning the workholding device as near to the column of the machine as practicable minimises any deflection of the machine tool structure due to the effects of overhang. The direction of the cutting forces should always be directed towards a positive, fixed location. Dowels are useful location devices in this respect since they offer positive location in all directions. Finally, if there is flexibility to position the workpiece or the table, other factors that might be considered concern the ease of loading and unloading and the ease of swarf removal from the cutting zone. The component should be positioned such that the cutting action directs the machined swarf away from the operator. Some common workholding methods are discussed below.

Machine Vice: The most versatile workholding device for small prismatic components is the familiar machine vice. This offers simplicity, versatility, rigidity; it can easily be adapted to power-assisted operation and it is readily available in a range of sizes at a reasonable cost. Extra flexibility may be offered by the use of a swivel vice (allowing rotation in the horizontal plane), or a universal vice (allowing rotation in both horizontal and vertical planes). The vice should be clamped to the worktable in such a way that the component is positively located (to resist cutting forces) against the fixed jaw of the vice. Relying on the frictional location of the vice jaws is not recommended. Specially machined replaceable vice jaws can enhance the location and clamping ability of the machine vice, at minimal cost.

Clamping Elements: Clamping elements comprise a range of modular components which can be assembled to form workholding devices. A minimum set would include a range of studs, nuts, washers, clamping strips, packing pieces or stepped blocks, tee-nuts/bolts, etc. They're normally used in conjunction with other standard items of workshop equipment for supporting the work - for example, parallel bars, vee-blocks, angle plates and so on.

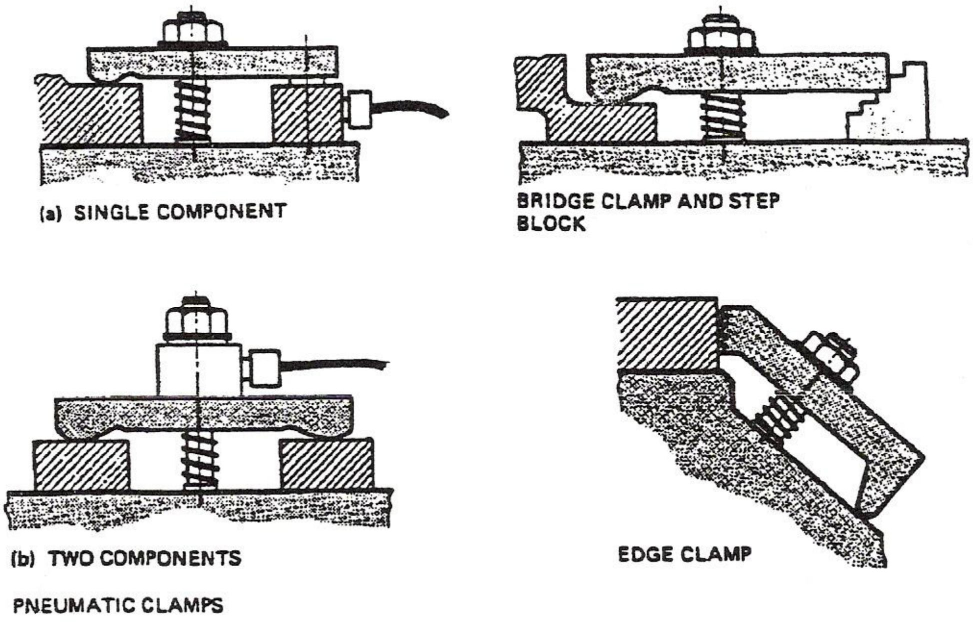


Figure 34 - Clamping Set Operation

The most common set-ups are those of the 'bridge' or 'strap' clamping arrangement or an 'edge' clamping arrangement. The latter is preferred where it is required to machine the whole of the top face of a component without necessitating an intermediate clamp change. These are illustrated in Figure 34. When used in these configurations, certain points should be observed:

- Position studs/bolts as close as possible to the workpiece.
- Pack the rear of the clamp until it is level with, or slightly higher than, the height of the workpiece – never lower!
- Position clamps so that the stud is closer to the workpiece than it is to the packing block.
- Select studs that are as short as possible but long enough for the nut to be fully engaged on the thread.
- Always use spherical clamping washers underneath the clamping nuts.
- Ensure that all clamps and packing are clear of, and do not impede, the intended cutter path.
- Always clamp on a solid part of the workpiece; use supporting devices where necessary.
- Use more than one clamp.
- Before moving clamps, after partial machining, ensure that one or more other clamps are still holding the workpiece in position.
- Springs should be inserted between the machine table and the strap clamp to support the clamp during loading and unloading.

The selection and usage of clamping sets is largely a manual operation. It is possible to automate the operation by using small pneumatic cylinders instead of clamping nuts. They find greatest application in CNC work where components can be loaded away from the machine on sub-tables or pallets. Ready-loaded pallets can then be exchanged rapidly at the machine tool.

Fixtures

Special-purpose fixtures are often employed. The decision to design and manufacture such a fixture will depend on such factors as:

(a) The size, shape and form of the component or raw material.
(b) Suitability, or otherwise, for efficient workholding (and setting) by other means.
(c) The number of components required and the likelihood of repeat orders.
(d) Anticipated increase in productivity.
(e) Projected cost of producing the fixture.
(f) The need to coordinate workholding with automated loading and unloading of components.
(g) The possibility of machining a number of components at the same set-up.
(h) The need to provide extra degrees of movement not provided by the machine tool itself. For example, rotary indexing of components.

If a fixture is to be specially designed, it is a good idea to incorporate some means for establishing the X0, Y0, Z0 datum position for setting the cutting tool. This could take the form of a simple hardened and ground setting block which forms part of the fixture but does not interfere with the machining operations. The operator would then 'touch' on to the setting block and set the appropriate axis registers to zero.

Other Workholding Devices

Many advanced CNC machining centres are equipped with the capability to perform simultaneous machining operations in more than three axes. Extra axes of motion may include rotary motions about the primary linear axes X, Y and Z. Where such capability exists, extreme flexibility for the production of complex components is provided. Where it does not exist, it may be necessary to fall back on traditional workholding devices to provide the extra axes of movement. In this category are the dividing head, rotary table, adjustable angle plate, and so on.

In general, such devices are not fully compatible with the concept of CNC machining techniques. However the specific application will determine their use.

Machine Operating Considerations

Swarf removal from the cutting area may have to be carried out during machining. The program cycle may have to be stopped at pre-determined intervals using program codes to enable the operator to clean away swarf.

Emergency Shutdowns

Before using the machine the operator must be aware of the location of the emergency stop button or buttons. Emergency stops invariably require the program and cutting tool to be re-set at the program start position before machining can re-start.

Feed Override Control

This control usually provides for a feed rate override from 0 - 150%. This allows the operator to manually correct programmed feed rates within these limits. Feed rate override can also be used to regulate jog feed when setting up tools.

Self Assessment

Questions on Background Notes – Module 4.Unit 11

1. Briefly explain Numerical Control in relation to a CNC Machine.

2. List two of the Constructional Features of a CNC Machine.

3. Name some of the advantages of a CNC Machine.

4. In Part Programming what does selection of Zero Point mean?

5. What does the Setting of Axis Datum mean?

Answers to Questions 1-5. Module 4. Unit 11

1.

Numerical Control:

Numerical Control (NC) is the technique of giving instructions to a machine in the form of a code which consists of numbers, letters of the alphabet, punctuation marks and certain other symbols. The machine responds to this coded information in a precise and ordered manner to carry out various machining functions.

Instructions are supplied to the machine as a series of blocks of information. A block of information is a group of commands sufficient to enable the machine to carry out one individual machining operation e.g. move cutter from position 1 to position 2 at a specified feed rate.

Each block is given a sequence number for identification. The blocks are then executed in strict numerical order. An example of a block is as follows:

NI G00 X10 x Y20

The meaning of this information is as follows:

NI is block number 1; G00 directs the machine or cutter to move at the rapid traverse rate. X10 and Y20 are the coordinate values of the target points of the movement.

When the instructions are organised in a logical manner they direct the machine tool to carry out a specific task - usually the complete machining of a workpiece or ‘part’. It is thus termed a part program.

2.

Constructional Features of CNC Machines:

A conventional machine tool has an intelligent source for error compensation (the operator). During machining a skilled operator can vary the cutting conditions to compensate for deflection, vibration etc. to generate the desired shape, size and finish.

The CNC machine can only compensate for an error that is detected and communicated to the control. Deflection, vibration etc. cannot as yet, be easily monitored. For this reason NC machines are made stronger and stiffer to perform to a more accurate standard than their conventional counterparts.

The capacity for varying the conditions while machining is therefore limited. As far as possible the conditions have to be established as the program is produced.

In addition to this, CNC machines are spending more time per shift cutting than conventional machines did in the past. This higher percentage of cutting time results in faster wear rates on the slides and transmission systems.

Conventional machine tools are also designed with the view to having the skilled operator standing directly in front controlling the machine. This is no longer required for CNC since the machine is operating under program control.

Optimum cutting speeds and feeds, continuous path machining, rapid slide movement to bring the tool close to the work and then sudden stopping, all subject the machine to forces which are not encountered on conventional machines.

3.

Advantages of CNC Machines:

- Reduced Lead Time.
- Elimination of Operator Errors.
- Flexibility in changes of Component Design.
- Reduced Scrap and Inspection.
- Complex One-Offs and Small Batch Quantities.
- Lower Labour Cost.
- Accurate Costing and Scheduling.

4.

Selection of Zero Point:

In order to define certain points on a workpiece in this manner, you will first have to decide where to put the coordinate system on the workpiece, especially where to place the zero point. For milling the zero point (or reference point) of all dimensions may be placed anywhere on the workpiece. Let's call this zero point 'workpiece zero' (WZ) or workpiece datum. To save trouble of unnecessary calculations it is advisable to place workpiece zero at that point on a part drawing on which most of the dimensions are based.

Cont.

4. Continued.

A workpiece datum maybe defined as a point, line or surface from which dimensions are referenced. It may or may not be within the workpiece area.

The Symbol used to denote Workpiece Zero is:

Figure 15: Example of Zero Point Options on Workpiece.

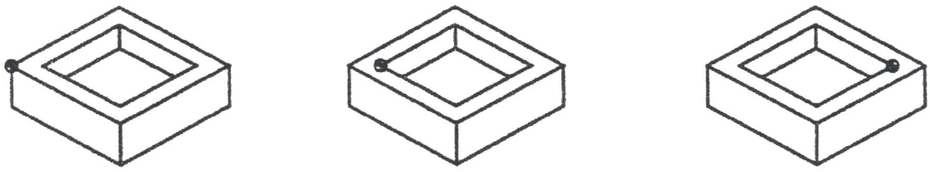
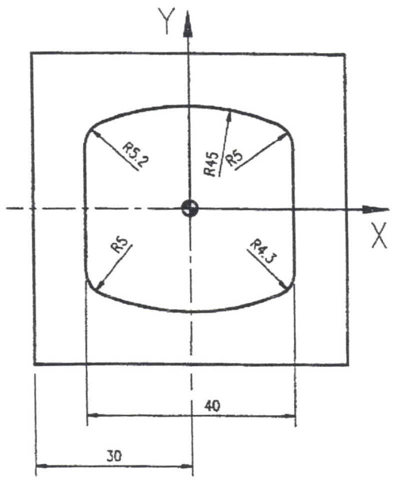


Figure 16: Example of Component with Zero Point in Workpiece Centre.



5.

Setting Axis Datums:

On CNC machines employing fully floating datum facilities it is common to position the workpiece or workholding device on the machine table, for convenience. The tool, or setting probe, is then jogged manually to touch the component in each axis in turn. With the tool or setting probe in the correct setting position (component datum position), one of two actions may be taken. The choice will depend on the machine tool.

The first action involves setting the relevant axis register to zero by entering at the console. A button marked 'axis zero' may also have to be depressed to confirm the action. Thereafter, the machine zero position is assumed to be that point. All subsequently programmed positional moves will be made with reference to this zero point.

The second action is perhaps more prevalent on CNC turning centres. At start-up, many turning centres send their slides to a known position normally at the extremes of their axis movements. When at this position indicated by limit switches detecting the limits of travel, the control unit 'knows' that the slides are at known dimensions from the machine datum point. This may be, for example, 800 mm from the back face of the chuck in the Z-axis, and 500 mm from the centre-line of the spindle in the X-axis. The exact dimensions will, of course, vary from machine to machine. The tool can then be jogged to touch on the component as previously described. The dimensions showing on the axis read-outs are then subtracted from the known 'limits of travel' dimensions. The resulting values are then entered into axis offset registers within the CNC control unit. In a roundabout way this creates the datum position for the tool tip. The datum positions are not initialised to X0 and Z0 however; they are computed (within the control unit) from the known 'limits of travel' dimensions, and the values contained in the axis offset registers.

5. Continued.

Regardless of the actual procedure, the part programmer programs the part from a datum point relating to the component. The positioning of the component at the machine, and a similar manoeuvre to those described above and then reconciles the position of the component to the datum positions set at the machine tool. Quite obviously, the correct procedure for each individual machine must be followed. To this end the operating /setting manual for the individual machine must be thoroughly studied.

Index