ORDNANCE FACTORIES INSTITUTE OF LEARNING

MEDAK, YEDDUMAILARAM – 520 205, A.P.

COMPUTER NUMERICAL CONTROL TECHNOLOGY

PART PROGRAMMING

CNC

COMPUTER NUMERICALCONTROL
TECHNOLOGY

PART PROGRAMMING

CNC PART PROGRAMMING

INDEX

SECTION 1

NC/CNC MACHINES, CONTROL SYSTEM FEATURES

NC/CNC MACHINES, CONTROL SYSTEM FEATURES MAJOR PARTS OF CNC SYSTEM, ADVANTAGES OF CNC MACHINES

INTRODUCTION :

Development of NC/CNC machines is considered as an outstanding contribution to machine tool engineering. It is definitely a step in automation of the machining processes with a flexibility which makes the technology more versatile and widens the range of application. NC technology merely incorporates the automation of machine tool with the aid of modern electronics.

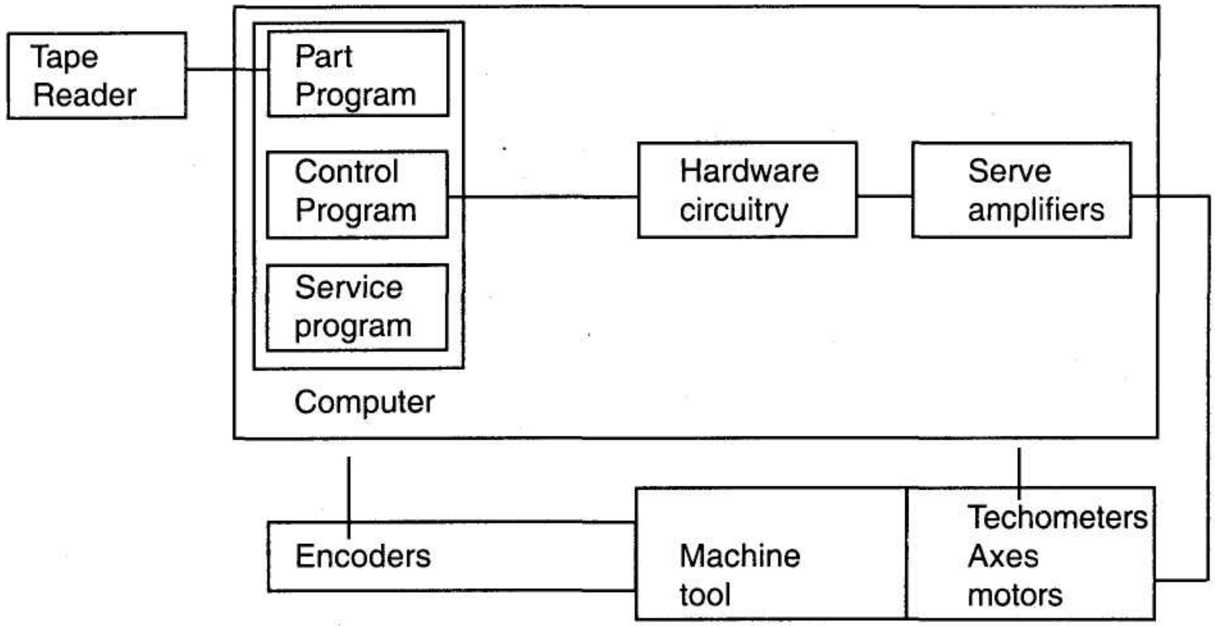
NC MACHINES :

Numerical control is defined as a form of programmable automation in which the process is controlled by numbers, letters and symbols. A machine tool is said to be numerically controlled if it operates in a semiautomatic or automatic cycle as per instructions transmitted to it in a coded form In numerical control, the numbers from a program of instructions designed for a particular work part or job. When the job changes, the program of instructions is changed. This technology has been applied to a wide variety of operations including turning, milling, sheet metal working, welding, inspection etc.,

CNC MACHINES :

In case of computer numerical control machine tools, a dedicated computer is used to perform all basic NC functions. The complete part program to produce a component is input and stored in the computer memory and the information for each operation is fed to the machine tools. The part programs can be stored in the memory of the computer and used in future. CNC machine tools are widely used due to many new control features available on these machines.

Schematic diagram of a CNC system.



PRINCIPAL CHARACTERISTICS OF NC/CNC SYSTEM :

The basic features of NC/CNC system.

1. Machine tool.
2. Control unit
3. Input data
4. Measuring system
5. Accuracy
6. Auxiliary functions

1. Machine tool

The major information is the type of machine (i.e. Vertical milling machine, Horizontal milling machine etc.,) and it must be followed by additional information such as.

Number of machine axes

Maximum allowable traveling dimensions of each axis.

Maximum spindle power

Range of speeds and feeds.

Constant possibilities.

Static Dynamic characteristics.

2. Machine control unit

Basic information includes

Number of motion control channels.

Type of control structure - Analogue or Digital

Type of system - Point to point, Straight line, Continuous path contouring.

Type of interpolation - Linear, Circular, Parabolic or Combination of these.

Maximum feed rate.

3. Input data

Input data includes information about the control medium, information about computer programs should also be given. Knowledge of the following must be provided.

Control medium: perforated tape, magnetic tape, etc.

Capability of manual handling of input data

Type of dimensional programming: Absolute, Incremental or both etc.,

Number of digits in each dimensional word etc.,

Input resolution

Information about programming methods and languages

List of Preparatory (G) & Miscellaneous (M) functions

Tool changing codes

Speed and Feed range codes

Tape reader type - Mechanical or photo electric etc.,

Tape code - ISO, EI A

Recommended order of words in a block & number of digits in each word

Use of algebric signs.

4. Measuring system

Features of the measuring system

Method of coupling the measuring element

Absolute or Incremental measurement

Type of element - Encoder, Resolver, Inductosyn etc.,

5. Accuracy

Positioning accuracy : Difference between required and actual position of machine slide.

Contour accuracy : Gain in a contouring system

Repeatability : Difference between accuracy on repeating the Operatio

6. Auxiliary information

Floating Zero, Zero offsets, Fixed Zero

Backlash take-up circuit.

Compensation capabilities for length and radius of tool

Provision for mirror images, scaling etc.,

NC/CNC System Classification :

a) Based on feed back control

b) Based on control system features.

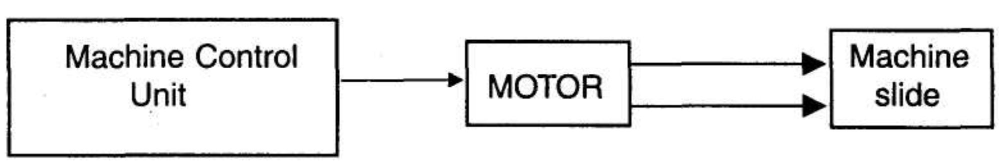
a) Classification based on Feed back control system

Based on feed back control, the NC/CNC systems are classified as Open loop & Close loop control systems.

i) Open loop control system

Machine tool control in which there is no provision to compare the actual position of the cutting tool or work piece with the input command value are called open loop systems. I open loop system the actual displacement of the slide may vary with change in extern; condition and due to wear of the components of the drive mechanism. Open loop systems are less expensive then closed loop systems due to the absence of monitoring devices and the maintenance is not complicated.

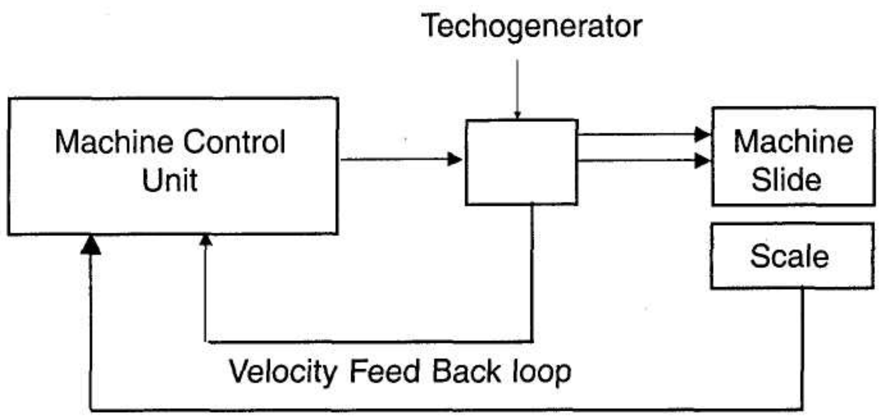
Block diagram of an open loop system:



ii) Closed loop control system

In a closed loop control system the actual output from the system i.e. actual displacement of the machine slide is compared with the input signal. The closed loop systems are characterized by the presence of feed back devices in the system. In the closed loop control system the displacement can be achieved to a very high degree of accuracy because a measuring or monitoring device is used to determine the displacement of the slide.

Block diagram of a closed loop system.



Displacement Feed Back Loop

b) CLASSIFICATION BASED ON CONTROL SYSTEM FEATURE

Based on control system feature, the NC/CNC control systems are classified as .

1) Point to point control system
2) Straight line control system
3) Continuous path / contouring control system

1) Point to point control system

In point to point control system, control requires to position the machine tool slides to the pre determined coordinate point. The tool moves to the predetermined position in the shortest possible time. This control system is suitable for the drilling, boring, tapping, punching and jig boring machines.

2) Straight line control system

In straight line control system, in addition to point to point control, control to machine along a straight line at controlled feed rate is provided. This is suitable for straight line milling and turning operations.

3) Continuous path / contouring control system

In contouring control, several axes can be simultaneously controlled. This enable machining of various contours / profiles.

MAIN ELEMENTS OF CNC MACHINES

To enable electronic automation with high rate of metal removal at optimum cutting conditions, maintaining high repetitive accuracies with at most safety to the operator and the machine, CNC machines are specially designed.

The main elements of CNC machines are :

i) Machine structure
ii) Guide ways
iii) Spindle bearings & mounting
iv) Drive units
v) Mech. Power transmission
vi) Position feed back elements / systems
vii) Additional accessories / equipment
viii) Control software
ix) Chip removal system
x) Safety features

i) Machine structure

Structures are designed to withstand static, dynamic & thermal loads providing **high** stiffness, rigidity & damping properties. The material used is generally mechanite cast iron special casting with nickel & copper elements. Welded structures also in wide usage.

ii) Guide ways

Guide ways are designed to reduce/ eliminate friction, providing high, precision. This is achieved through aerostatics / hydrostatic guide ways, tycoway bearing. LM guide ways and the surfaces of counter guides coated with PTFE (Poly Tetra Ethylene) etc.

iii) Spindle bearings & mounting

Designed for high accuracies, stiffness, stability and to minimise torsional strain providing high rpm range.

iv) Drive units

AC/ DC servo motors and drive systems with infinitely variable speed and high response are used.

v) Mech. Power transmission

Specially designed with minimum gear transmission and isolated to reduce thermal effects, etc. Sliding friction is converted to rolling friction by re-circulating ball screws with nuts arrangement etc. providing precision movement eliminating backlash, stick-slip etc.

vi) Position feedback elements / systems

Linear / rotary transducers, tachogenerators etc., are provided for precise control of the movements of the machine slides etc.,

vii) Additional accessories / equipment

Level of automation depends on the accessories/equipment and further enhance the optimum utilization of the CNC machine. The equipment's such as Automatic tool changer, Automatic attachment changers, Work changers, Electronic probes, Tool monitoring system etc.

viii) Control software

Automation level & optimum utilization of the CNC machine depends on features provided in the control system. Such as Simultaneous control of no. of axes. Compensation functions, Mirror image, Scaling etc.

ix) Chip removal system

Efficient chip removal system eliminates thermal effects & thus improves the quality of cutting and the job being machined.

x) Safety

Suitable covers for guide-ways etc. and electronic interlocks for the safety of the operating personnel and machine are provided.

ADVANTAGES OF CNC MACHINE

Flexibility
Small batch size
Reduced work-in-process inventory
Reduced tooling
Reduced lead time
Reliable operation
Repetitive quality
Reduced scrap rate
Optimum machine utilization
Increased operational safety
Reduction in manufacturing costs
Short response time to implement design changes.

SECTION 2

CO-ORDINATE SYSTEMS

COORDINATE SYSTEM

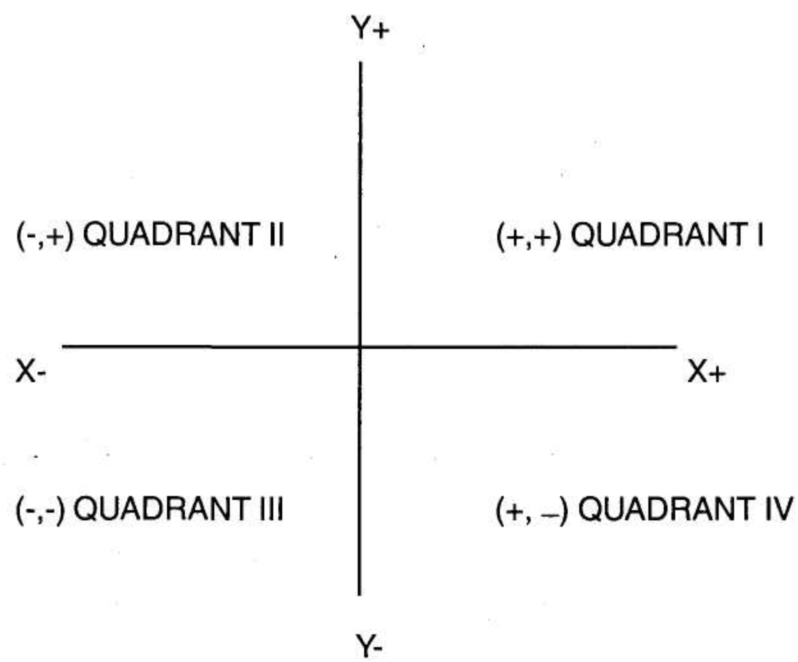
Coordinate is the relative position of a point with reference to the datum point generally denoted by zero point and there are mainly two types of coordinate systems that may be employed by a control system to position the tool or cutter in relation to the work piece.

1. Cartesian Coordinate system
2. Polar Coordinate system

Each have their application and may be used independently or mixed according to the features present with in the component.

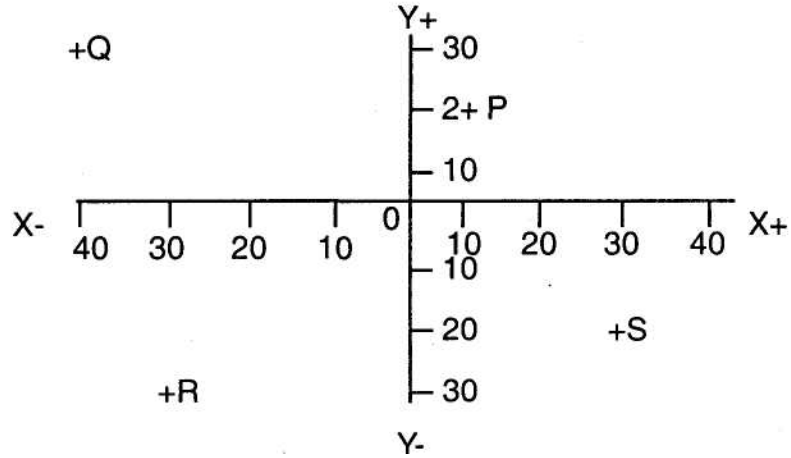
1. CARTESIAN CO-ORDINATE SYSTEM

In Cartesian coordinate system the axial lines are drawn at right angles to each other with respective to a datum then it sets off four areas called quadrants. The horizontal line is called X-axis and vertical line is called Y-axis. It is represented as given below.



1. If the point lies in quadrant-I both X and Y coordinate are positive sign.
2. If the point lies in quadrant-II, X is negative sign and Y is positive sign. •
3. If the point lies in quadrant III, both X and Y coordinates are negative sign.
4. If the point lies in quadrant IV X is positive and Y is negative sign.

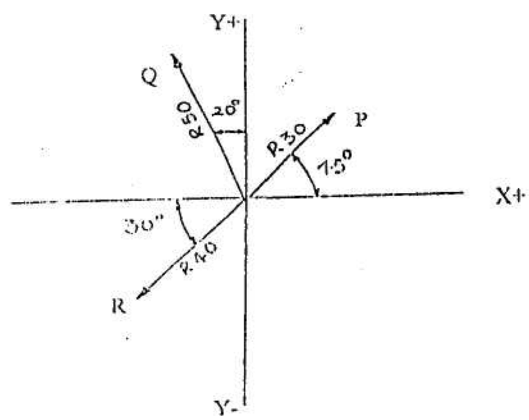
In Cartesian coordinate system point is defined by its distance from its perpendicular axis and sign.



POINT	X CO-ORDINATE	Y CO-ORDINATE
P	+10	+20
Q	-40	+30
R	-30	-30
S	+30	-20

POLAR COORDINATE SYSTEM

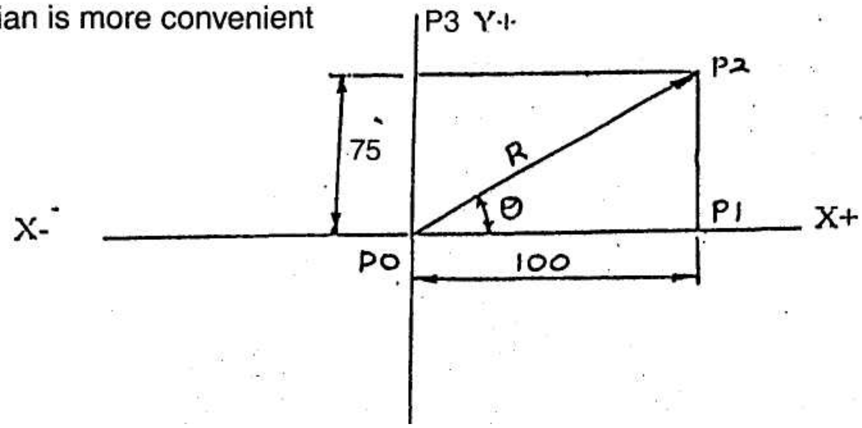
In Polar coordinate system the point is represented by a radius*(distance from zero point) and angle (Angle from horizontal axis).



POINT	RADIUS	ANGLE
P	30	45deg.
Q	50	110 deg
R	40	210 deg.

In CNC Programming, control systems will accept the both coordinate systems but depends on the component features some times Cartesian system is more convenient and in some cases polar is more convenient.

In this Cartesian is more convenient



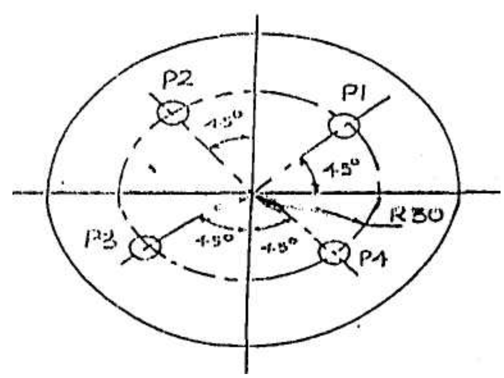
CARTESIAN SYSTEM

POINT	X CO-ORDINATE	Y CO-ORDINATE
PO	**0**	**0**
P1	**100**	**0**
P2	**100**	**75**
P3	**0**	**75**

POLAR SYSTEM

POINT	RADIUS	ANGLE
PO	0	0 deg.
P1	100	0 deg.
P2	R	0 deg.
P3	75	90 deg.

POINT	RADIAUS	ANGLE, deg.
P1	30	45
P2	30	135
P3	30	225
P4	30	315

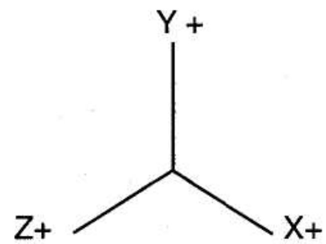


In this polar is more convenient

AXIS IDENTIFICATION

Controlled axis on CNC machine tools are identified according to established standard: BS:3635 part 1-1972 illustrates the axis classification of twenty five CNC machine.

The basis of axis classification is the 3 Dimentional cartesian coordinate system . The is the system employed for graphical plotting in mathematics. In machine tool terms the ax correspond to longitudinal, transverse and vertical planes of movement. The three dimensions of movement are identified by the upper case letters X,Y and Z. It is also necessary to able I identify the direction of movement along each of the controlled axis. Direction of movement is specified by either"+ or -" from an established machine datum according to established standards.



Z-AXIS

The Z-axis of motion is always parallel to the main spindle of the machine. It does n< matter whether the spindle carries a rotating tool or a rotating work piece. On vertical machinir centres and vertical laths the Z-Axis will be vertical. On horizontal machining centres an CNC turning centers the Z-Axis will be horizontal.

Positive Z movement (+Z) is in the direction that increases the distance between the work piece and the tool. On vertical machining centres +Z movement is always away from *or* machine work table. On horizontal machining centers and turning centres the direction 4 motion is always away from the spindle.

X-AXIS

The X-Axis is always horizontal and is parallel to the work holding surface. If the Z-ax is also horizontal as in horizontal boring machines, the positive X-axis is to the right when looking from the spindle towards the work piece. When the Z-axis is vertical i.e., as in a vertical jig boring machine, the positive X-axis is to the right on single column machines when looking from the spindle towards its supporting column.

Y-AXIS

The Y-axis is perpendicular to both X and Z axes in order to determine the direction 1 positive Y axis consider the +X axis rotated into Y-position. This position will advance a rig hand screw in +Z direction. This destination will be made clear by reference to the right-hand rule.

ADDITIONAL AXIS MOVEMENT

It is common for CNC machines and turning centres to have additional linear axis of movement often in parallel with the three primary axis. For example a vertical milling machine may have saddle/knee movement and spindle quill movement both operating in the Z-axis. Obviously the control system must be able to distinguish from the other in order to command appropriate movement of the correct element.

In general where there is more than one moving element in the same axis, one is identified as being the primary movement and is allocated the primary axis designation X,Y or Z. Secondary movement in the same axis, are then designated by the upper case letters, U,V,W corresponding to motion in the X, Y, and Z axis respectively.

It is also possible for rotary movements to be provided as part of the original machine in the form of built-in rotary tables. These rotary axis movements are identified by the upper case letters A, B and C which correspond to rotary movements about the X, Y and Z axis respectively. Clock wise rotation is designated as positive movement and counter clockwise rotation as negative movement, positive (clock wise) rotation identified by looking in the +X,+ Y and +Z direction respectively.

METHOD OF LISTING THE COORDINATES OF POINTS IN NC/CNC SYSTEM

Two types of coordinate systems are used to define and control the position of the tool in relation to the workpiece. Each system has its own application and the two coordinate systems may be used independent or may be mixed within a CNC part program according to the machining requirements of the component.

The coordinate data input systems used are

1) Absolute coordinate data input system

2) Incremental coordinate data input system

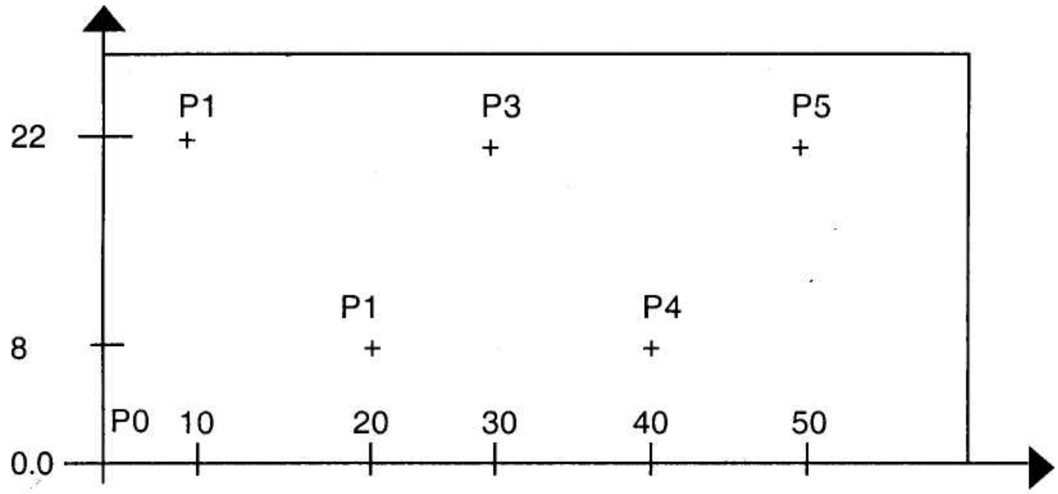
ABSOLUTE CO-ORDINATE DATA INPUT SYSTEM

In the absolute system the coordinate points are always referred with reference to the same datum.

The datum positions in the X-axis, Y-axis and Z-axis are defined by the user/programmer before starting the operation on the machine.

INCREMENTAL COORDINATE DATA INPUT SYSTEM

In the incremental system the coordinate of axis point are calculated with reference to previous point i.e., the point at which the cutting tool is positioned is taken as datum point for calculating the coordinate of the next point to which movement is to be made.



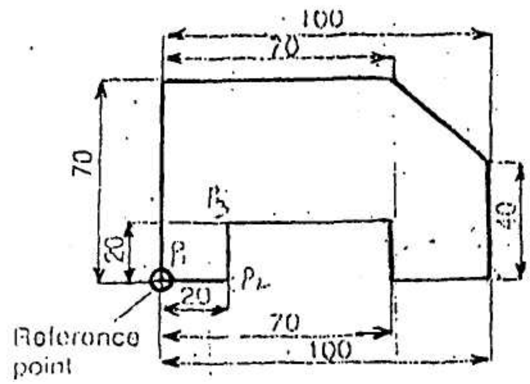
ABSOLUTE CO-ORDINATE SYSTEM

POINT	X CO-ORDINATE	Y CO-ORDINATE
PO	0	0
P1	10	22
P2	20	8
P3	30	22
P4	40	8
P5	50	22

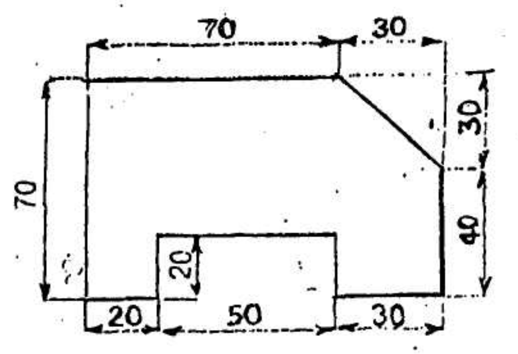
INCREMENTAL COORDINATE SYSTEM

POINT	X CO-ORDINATE	YCO-ORDINATE
P0	0	0
P1	10	22
P2	10	-14
P3	10	14
P4	10	-14
P5	10	14

Absolute Dimensions

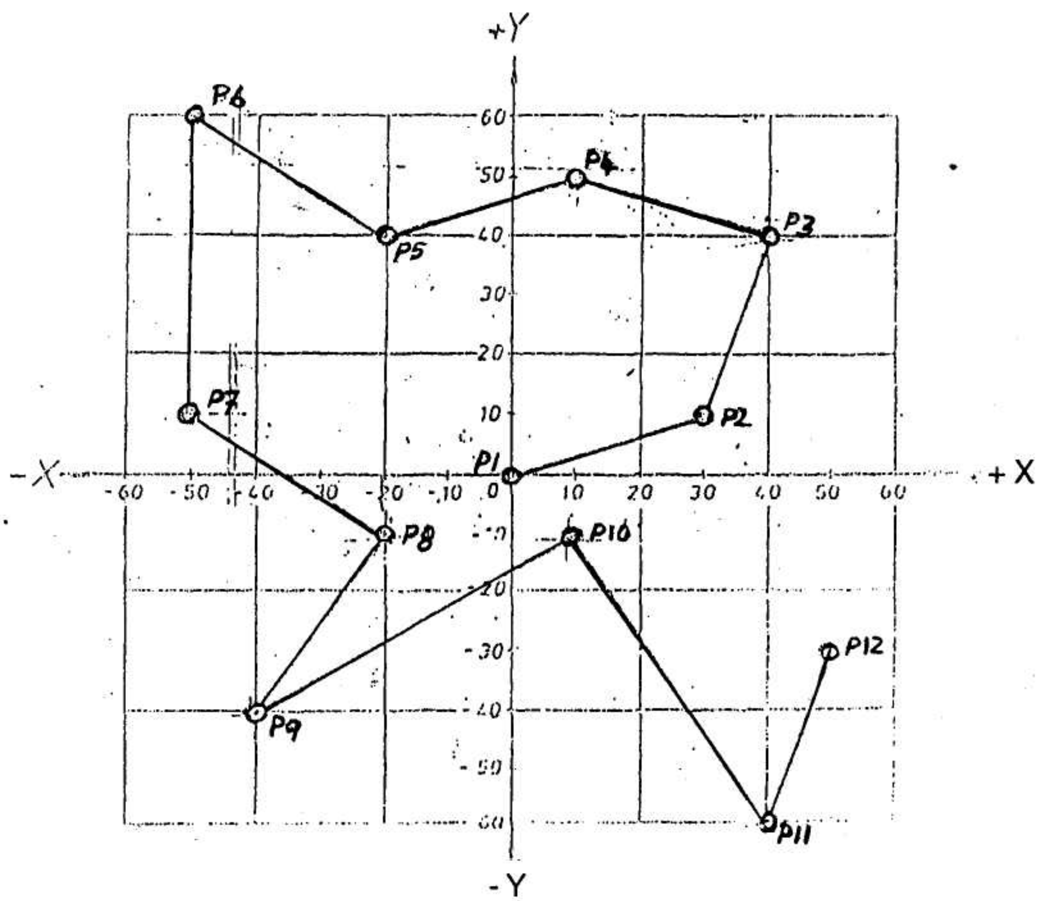


Incremental Dimenstions

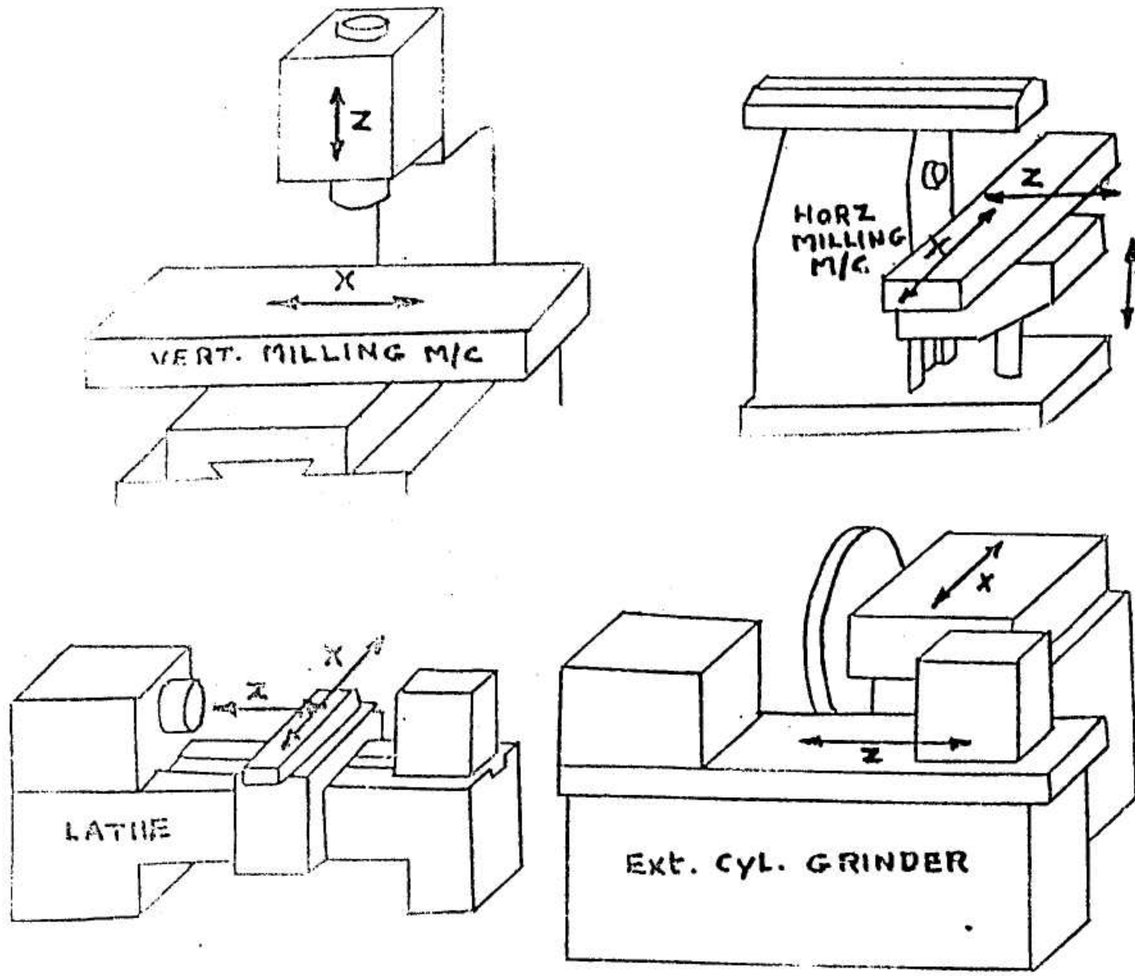


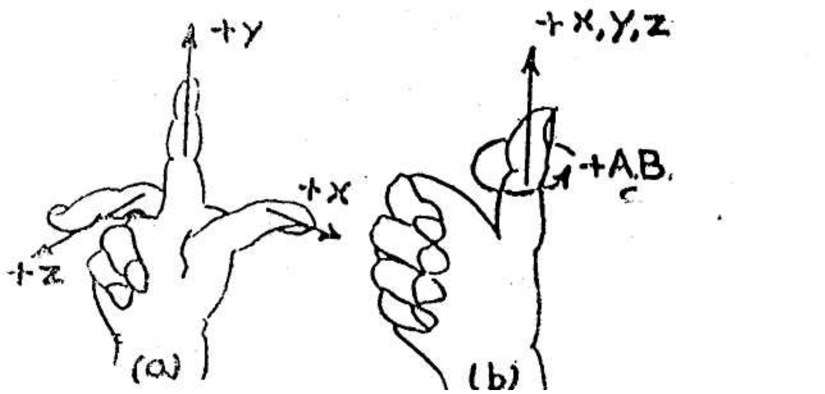
This point has the function of a coordinate zero point.

CARTESIAN COORDINATE SYSTEM



AXES





THE RIGHT HAND RULE

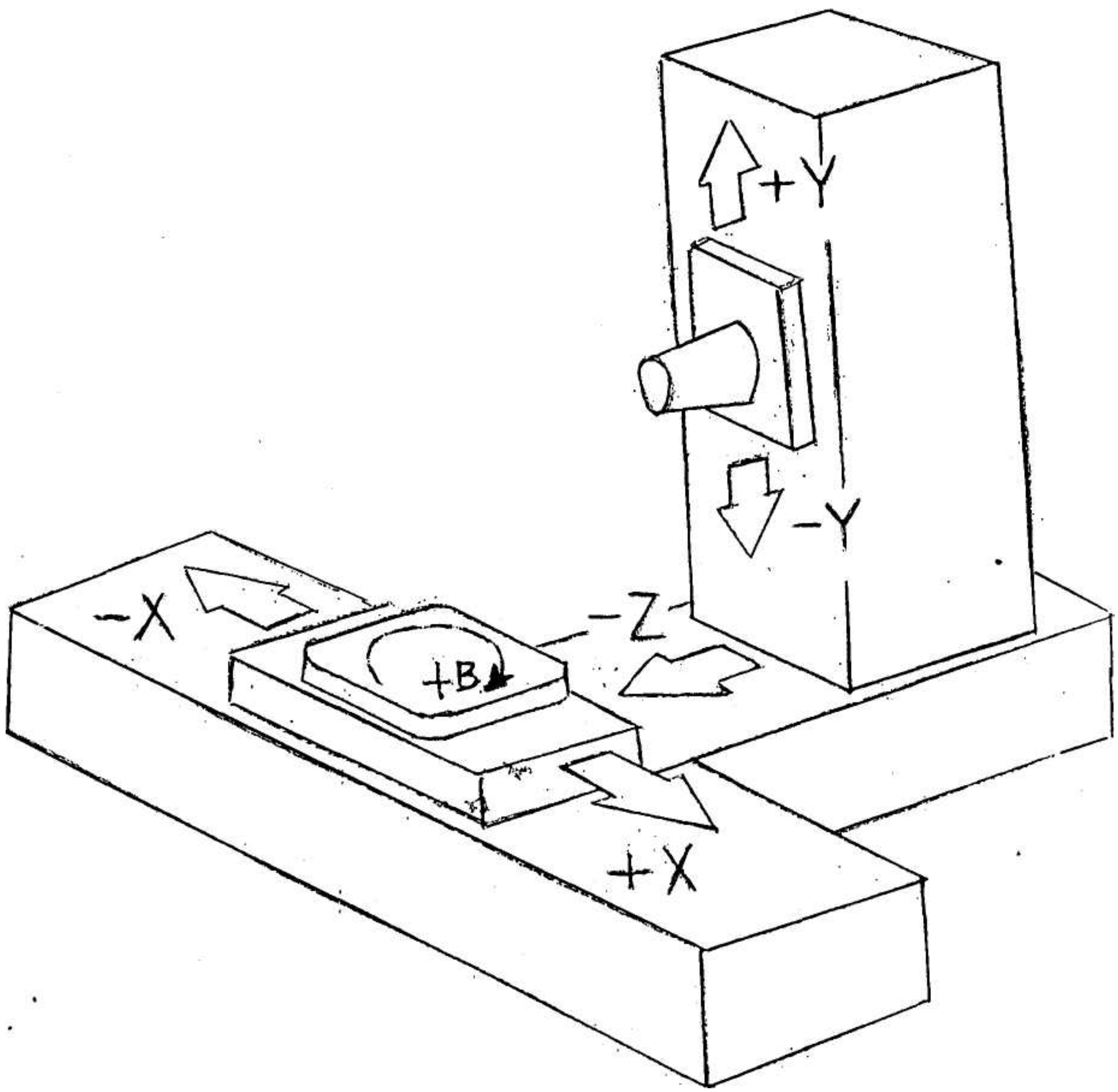
a) Relationship between Axes

b) Relationship between positive linear and rotary axes.

BASIC CNC COURSE

AXES

MACHINING CENTRES



SECTION 3

CNC MACHINE PROGRAMMING

CNC MACHINE PROGRAMMING

Part programs for CNC machines are prepared in any of the two ways.

a) Manually, coded in the individual Machine's control language.

b) Computer Aided, coded in the generic language and then post-processed to a Machine's specific language.

1.1 PART PROGRAM

The part programming is the process of converting information from drawing specifications of a component to a coded paper tape that will produce the desired component on a CNC machine.

The part program is a series of coded information that direct the operations of the CNC machine. The instructions contain all the machine and the control functions necessary to make the machine perform a specific task. Determining which functions are required, coding them, and arranging them in the proper sequence is called manual programming.

1.2 CONCEPT OF CNC MACHINING

CNC machine needs adoption of production drawing to the coordinate concept of machine. For the effective optimized operation of an CNC machine, the process of making the part program encompasses complete study of production drawing and its adoption to CNC machine concept, sequencing of operations with generation of specific tooling requirement etc... This needs an systematic approach or a concept.

'Selection of Workpiece.

With the machining center, remarkable, reduction in the machining time is realised compared with the conventional machine when.

a) The workpiece is of complex shape.

b) The machining process is long.

c) Multiple plane machining is required.

'Preparation of operation layout

On selecting the component for loading, on to the CNC Machine, operation layout is prepared.

While preparing the operation layout the following points, are studied in detail, a)

Loading or resting face.

b) Locating/butting surfaces

c) Clamping provision

d) Operations possible on the CNC Machine.

e) Other machines required for completing the part.

1.2.1 LOADING ON RESTING FACE.

Loading or resting surface is selected such that maximum operations can be completed in one setup, otherwise two or more setups depending on the complexity of the component can be planned. Loading face should provide rigid setup to the component while machining, as well as it should not generate distortion due to tightening of clamps which can lead to inaccuracy of machined product. If required the resting face can bi pre machined before loading on to the CNC machine.

1.2.2. LOCATING/BUTTING SURFACE

Locating holes on the mounting face can be used for locating the component on to the fixture. With reference to these locating holes, coordinates are calculated. Locating hole not only positions but also prevents the component from getting disturbed from the fixture while machining.

If there is no provision to provide locating holes, on mounting face, two mutually perpendicular faces can be machined for locating purpose. In this case locating surface; are to be pre machined before loading on to CNC machine.

1.2.3.PROVISION OF CLAMPING

As for as possible clamping are to be provided with in the component. This prevents collision of tools with clamps. If this is not possible, clamps can be developed from outside and care must be taken during programming tool paths to avoid collision. Hole: or cut-outs if required for clamping have to be provided before loading on to the CNC machine.

1.2.4OPERATIONS POSSIBLE ON THE CNC MACHINE

All the operations possible on the CNC machine are to be listed under CNC machine as for as possible try to complete the job on the CNC machine itself.

1.2.5 OPERATIONS NOT POSSIBLE ON CNC MACHINE

Because of any reasons beyond the control, such as want of special tools, lack o approach can be planned on other conventional machine. Now the operation layout provides the complete process for machining the component.

1.3. PREPARATION OF SEQUENCE OF OPERATIONS & TOOLING SHEET

This provides tool description, length, diameter of tool and tool number, operation description, rotary table position, spindle speed and feed.

1.3.1 OPERATION DESCRIPTION

Operations described in the order of machining, i.e. first rough milling, rough boring, semi finish milling, semi finish boring, spot drilling, drilling & tapping finish milling, and finish boring. This order is maintained to provide time for cooling during finishing operations to get good geometrical accuracies.

As for as possible, if one tool is taken and with that all possible operations are completed, such as using rough milling cutter all sides are rough milled & using spot drill all holes on all sides are spot drilled etc. If that is not possible a tool can be called second time for various reasons.

to give sufficient time for cooling the tool, if the tool is subjected to maximum cutting.

Some operations are inter dependent on other operations in such case it may be necessary to call the tool for 2nd time.

The process sheet should be made as precise as possible. Since it helps quick changes in case of drawing amendments and also for, tape/program correction. Entering each coordinate system directly on drawing helps working out coordinates.

1.3.2TOOLING SHEET

Tool description, length, diameter and corresponding description of operation, to be done entered in a sequential form. This data helps the programmer to calculate tool path.

a) Tool length & diameter:

Automatic tool changer has the limitation on tool length & diameter and weight.

Tool lengths with respect to interference and also machine's machining range design.

1.3.3TOOL NUMBER

In order to call a tool and get the tool into the spindle, each tool is to be identified by certain number to be called in program. Care must be taken not to duplicate tool numbers.

Tool numbers entered in main program under letter T. Example T1010. T25221 etc., number of digits that follows vary according to the control system.

1.3.4 ROTARY TABLE POSITION

Rotary table will have mainly four positions 0, 90, 180, 270 degrees and in between positions vary according to machine design such, as from , 1 to 360 degrees are decimals of that. Table, position is entered under letter B in programme for example B0, B55, B90, B 270 etc.

1.3.5SPINDLE SPEED & FEED

Spindle speed in RPM has to be mentioned against each operation. Spindle speed entered under letter S.

Feed is given in millimeters/min. in case of CNC milling machines and mm/rev in ca; of CNC Turning machines. Feed is entered under letter F.

From Cutting speed the spindle speed in RPM, can be calculated by using the formula

V= *n* DS /1000 Where S = spindle speed in RPM

V = Cutting speed in meters/min.

D = Diameter of cutter

S = 1000V / *n* D (diameter to be turned in case of CNC Turning M/C)

7i = 3.14

Tool length, diameter, table position are to be reviewed after fixture design and clamping plans.

1.4 FIXTURE DESIGN

On preparation of sequence of operations & tooling sheet, fixture design activity carried out. While designing the fixture the following points are taken into account.

a) Proper resting, location & clamping

b) Compactness

c) Chip disposal

d) Easy loading & unloaing of component.

1.4.1 CLMPING AND POSITION PLAN

Clamping & Position plan provides the details of fixture, clamps and their relative positions and also fixture location on machine table. This facilitates quick location and clamping.

From the clamping & position plan position layout is prepared. It is a schematic diagram indicating X,Y,Z coordinates from origin. This facilitates quick calculations of coordinates.

1.5 coordinate sheet:

With the help of position layout and component drawing, the coordinates of all positions basing on operation sequence-are calculated. This includes X,Y,Z coordinates to position the tool at desired tool clearance/approach position before the start of the cutting the material and retrieval of tool to safe zone before positioning for the next cut. Work zero points list basing on which coordinates are calculated are also to be listed out.

1.6 PREPARATION OF PROGRAMME

With the help of the calculated coordinates one can start writing program, after familiarizing with machine program language and the format of it.

Programming language consists, of basically preparatory functions, miscellaneous functions, speed, feed, tool and tool offsets and so on.

1.7 PREPARATION OF TAPE

The program prepared is entered into a flexowriter which types out the program on a program sheet and at the same time punches out the data on a paper tape. Alternatively the program can be entered into a PC and this program can be copies on to a floppy disk.

This program is fed into the memory of CNC system of the machine and checked for possible bugs. After the program prove out and acceptance of component produced, the proved program is punched out on a paper tape through tape reader and punching unit of the machine or can be copied on, to a floppy and stored as a master copy for future use.

#NOTE

For the effective adoption of the concept of CNC machining, we need to create various data banks.

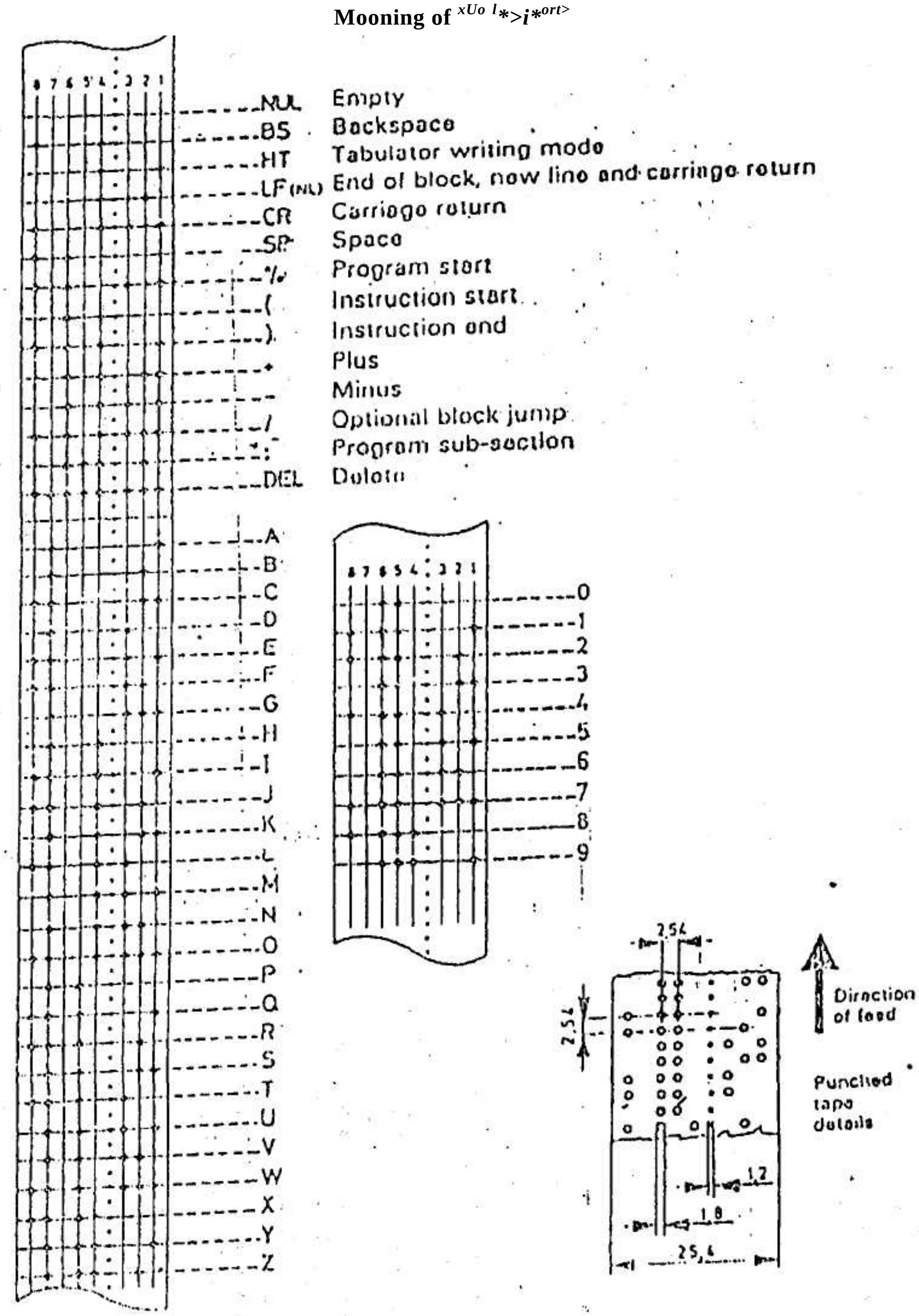
a) CNC machine capacity specifications and its control system's capabilities.

b) Cutting tool details

.d) Technology data such as

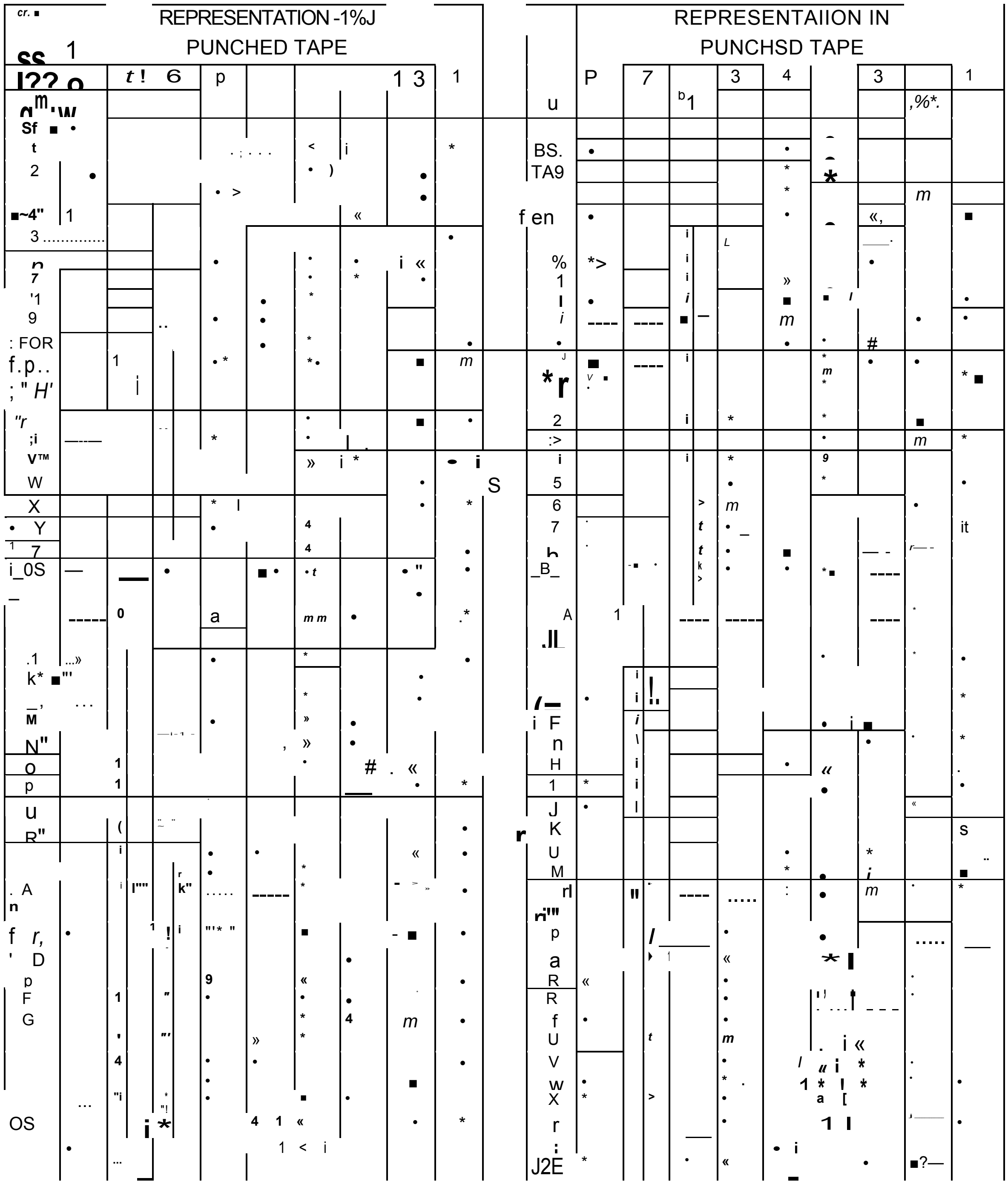
Cutting speeds, feeds as applicable to cutting tool material with respect to component's material.

ISO Standards for punched tape



The punched tape has 8 signal tracks, plus a drive perforation. The characters and coded. Tracks 1 -7 are information tracks, the 8th track is used for parity checking. The total number of holes in the ISO code is always an even number. The parity check ca therefore immediately recognises a fault in the tape punch (missing hole in the tape).

The character * program start* must precede the first program block.



Representation of the EIA numerical control code

Representation of the ISO numerical control code.

1.8 PROGRAMMING LANGUAGE

From the preceding paragraphs we have, understood that for the operation of the CNC machine, a part program has to be fed, to the control. In other words the control to be told how the tool has to make a movement, in a straight line or a circle, this and other information need for the control, to perform the operation, must be programmed in a language under stood by the control system of that machine.

The programming language, as any language, consists of words. Each word is a command, composed of a code letter, called address and a string of numerals.

Example: S 1000 S is the address for spindle speed
S 1000 means spindle speed 1000 rpm

Several words are combined together, is called a block. This describes a complete working step or one sequence of operation.

Example: S1000 M03 LF

Rotate spindle at 1000 rpm in clockwise direction.
LF is the character to indicate "End of block".

For programming any of the two codes can be chosen, ISO or EIA. The explanations herein follows ISO Code (DIN 66025) basing on Sinumerik control system.

NOTE: Program should follow only one code throughout, mixing of both the codes in one program is not allowed.

1.9 TAPE CODE

One inch, 8 channel opaque paper tape is used for data transfer. Each line combination of holes defines a unique character. The character in each code are defined to have even or odd parity

ISO Even number of holes

EIA Odd number of holes

The control automatically recognizes the correct tape format on reading the first character % (ISO) or EOB (EIA), which represents program start.

1.9.1 TAPE FORMAT

- Fixed format
- Tab sequential format
- * Word address format

FIXED FORMAT

This was used for point to point positioning systems, where the length of information block is relatively small. The words are presented in a specific order and all digits must be entered.

EXAMPLE : N G X Y S M EOB

13001+6400+000025003 EOB

13101+0101+025025003 EOB

Comparing with other formats, in the fixed format more characters are generally required for equivalent block and the length of the block can not be varied.

TAB SEQUENTIAL FORMAT

It was used in point to point as well as contouring systems. In this case, the tab character is used in between to separate the words. The tab is omitted from the end of the last word in the block, instead, the EOB code is written indicating end of block. In this format words, must be written in specific order. When a word is not used, in a certain block, it may be omitted, but tab character must be written.

Example:	N	G	X	Y	S	M	EOB
	130	01	+6400		250	03	EOB
	131		+0100	+0250			EOB

WORD ADDRESS FORMAT

This is the format used in today's control systems. Number of characters in a word is variable. Similarly number of words and their sequence in a block is also variable, hence called as variable block, Word & Address format.

In word address format each word consist of a alphabetic character called address followed by a string of numerals.

This format is precisely defined for each control system.

Example:

i) Sinumerik 3T version 4

%04, N04G02 XL+053 ZL+053 AL035 ID053 KD053 F05 L5 S04T04R2 RL+053 BD03 M02 H04.

ii) Sinumerik 3M version 4

%04 N04 G02 D03 XL+053 YL+053 ZL+053 ID053 JDO53 KD053 F05
S04 T04 H04 R2 RL+053 L5 PD033 M02 AL035

iii) Sinumerik 850M

%04 N04 G02/G03 D03, XL+053 YL+053, ZL+053 QL+053 AL035 ID053 JD05I
KD053 F05 L03 S05 T08 R03 RL+053 BD033 M4 H08 P04 UD053.

EXPLANATION :

First Character	Address	
Second character	L	Absolute/ Incremental dimension
Second character	D	Incremental
Symbol	+	Absolute dimension with +ve or -ve sign
First digit	0	Leading zero's not required
Second digit	decades	Position in digit string
Second & third digit	decades	Digit string position before and after decimal point
Character	LF	End of block

% Program start. The numerals followed indicates the program number.

N Address of block number. Digits that follows indicates the block number.

G Address G followed by the number indicates the G code.

X I Axis address, address with digits indicates
Y I the positioning data.
Z I

D Address D followed by number indicates the tool offset number.

A Address A, numerals indicate angle

I I Address of interpolation parameter
J I for X,Y,Z axis
K I

F Address for Feed expressed in mm/min or mm/rev.

S Address for Spindle speed in revolutions and in CSS mode, meters/min.

T Address for tool number

R Address for variables - R parameters

L Address L followed by numerals indicates the Subroutine number

P Number of passes of subroutine

U | Address for radius with circular interpolation
P I
B I

H Auxiliary functions

M M Codes- miscellaneous functions

1.10 CONFIGURATION OF PROGRAM

A part program comprises of

* The character for beginning of program
* A number of blocks
* The character for end of program

The character for beginning of the program precedes the first block in the part program. The character for end of program is contained in the last block of the program.

Subroutines and cycles can be components of the program.

SKIPPABLE BLOCKS :

The blocks which need not be executed during every program run, can be by entering the / character at the beginning of the block. This is activated via machine control panel or via the interface controller.

REMARKS / COMMENTS :

Remarks or instructions to operator can also be included in the program. The text is enclosed between the start of remark character"(" and the end of remark character")". The remark should not contain %, LF, (,) characters. The length of remarks is limited and differs from control to control. It is advisable to write the remarks at the end of the block or in a separate line.

1.11 MISCELLANEOUS FUNCTIONS

These are also called as" M Codes", activates switching on & off of motors like spindle motors and coolant motors etc.

Some miscellaneous functions are defined in ISO/DIN 66025, others are defined by the machine tool manufacturer, in accordance to the PLC program developed for a specific machine

1.11.1 M CODES: PROGRAM CONTROLLING

MOO Program stop (Unconditional)
Interrupts the program and machine stops.
Machining can be resumed by pressing the cycle start key.

M01 Program stop (Conditional)
This is same as MOO but activated only if optional stop switch is set to on.
With optional stop switch is set to off, M01 code is ignored.

M02 End of subprogram
This indicates the end of program and the control is reset. This code is commanded in the last block of the program.

M30 End of program
This is same as M 02 function. Control resets the program to the beginning.

M17 End of subprogram (Sinumerik control)

M99 End of subprogram (Fanuc control)
This is written in the last block of the subprogram indicating end of subprogram. Main program takes over control from here on.

M98 Call subprogram (For Fanuc control only)

1.11.2M CODES : SPINDLE CONTROLLING

M03 Spindle on clock-wise rotation

M04 Spindle on counter clock-wise rotation

M05 Spindle rotation stop 1.11.3M

1.11.3CODES : COOLANT CONTROLLING

M07 1) Coolant through spindle on

2) Mist coolant on

M08 External coolant on

M09 Coolant off

1.11.4M CODES : TOOL CHANGE

M06 Tool change

NOTE:

For the M codes defined by the Machine tool manufacturer it is necessary to look in to Working instructions for operation & programming provided by the machine tool builder for that specific machine.

1.12 AUXILLIARY FUNCTIONS:

These are also called as "H Codes" and are similar to M Codes for switching on/off and movements, which are not directly controlled by CNC. H Codes also used for tool length offsets.

1.13 PREPARATORY FUNCTIONS:

These are also called "G codes". These codes define.

- type of interpolation
- type of dimensioning
- activates operating conditions with in the control

G codes are divided into several groups. The program block will contain only one function from each group.

Some G codes are model (i.e. stored). Some are effective for a single block.

The machine tools are set to a reset condition by the machine tool manufacturer. That is when the machine is switched on, set G codes and functions will be active.

1.13.1 DIMENTIONING

Movement of a tool with in the machine coordinate system can be commanded by either absolute or incremental data input codes.

G 90 Absolute position data input

All dimensional inputs refers to a fixed datum, generally the work zero.

G 91 Incremental position data input

All dimensional inputs refers to the distance to be traversed and the sign, indicates the direction of movement.

Units of dimensions, data input in inches and data input in millimeters is controlled by G70 and G71 respectively in Sinumerik control, in Fanuc control by G20 and G21 respectively.

1.13.2 MACHINE ZERO

NC Machines are set to a coordinate system. The origin of this machine coordinate system is called the machine zero. When an NC machine is switched on, referencing is carried out by traversing each axis to reference point set by the machine tool builder. This operation sets/synchronizes the control system to the machine coordinate system

1.13.3 ZERO OFFSETS AND WORK COORDINATE SYSTEM

This is the coordinate distance of each axis by which the Work piece datum / work zero is offset from machine zero.
These offsets are entered into any of the zero offsets provided in the control system from MDI/CRT panel and commanded in the part program. Activation of this command sets commanded work coordinate system.

Fanuc control provides six work coordinate systems that is 6 sets of zero offsets, G54 G55, G56, G57 , G58 & G59. In addition by command G52, a local coordinate system can be set up with in the selected work coordinate system. Further by G92 command work coordinate system within the machine coordinate system can be created with the reference to the current tool position.

Sinumerik system provides four work coordinate systems, G54, G55, G56 & G5' In addition, by the additive zero off set commands G58 & G59, a coordinate system ca be setup with in the selected work coordinate system.

Additive zero offsets are programmable.

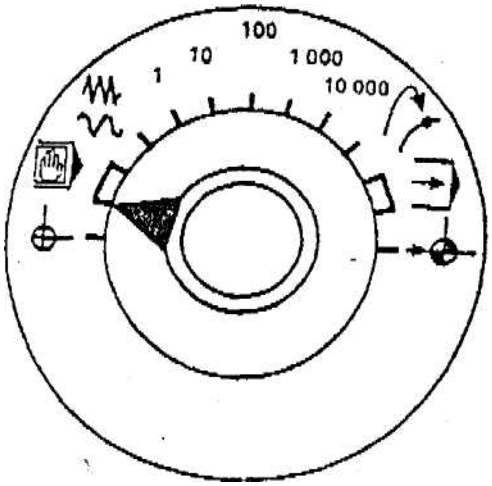
Axis values can be command in machine coordinate system by suppressing current work coordinate system with G53. G 53 is not a model function.

1.14 TOOL OFFSETS

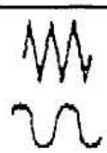
Usually several tools are used for machining one work piece. These tools will have different tool lengths. It is very trouble-some to alter the program considering the length of each tool used. Therefore the standard tool is selected and the difference between the position of nose of standard tool and the position of nose of other tools are measure! These measured values are entered in tool offset data page. Alternatively tool length can be measured using tool presetters. These offsets are controlled by "T" number (T0101 etc) in lathes and by "D" numbers (D1, D2 etc) some times by "H" numbers also (H1, H2 etc) in milling machines / machining centers. These tool offsets can be called in part program and machining can be carried out without altering the program.

1.14.1 Zero offsets and tool offsets are measured and entered in CNC control by operating personnel before starting the machining of workpiece. Accuracy of job depends on correctness of zero offsets and tool offsets.

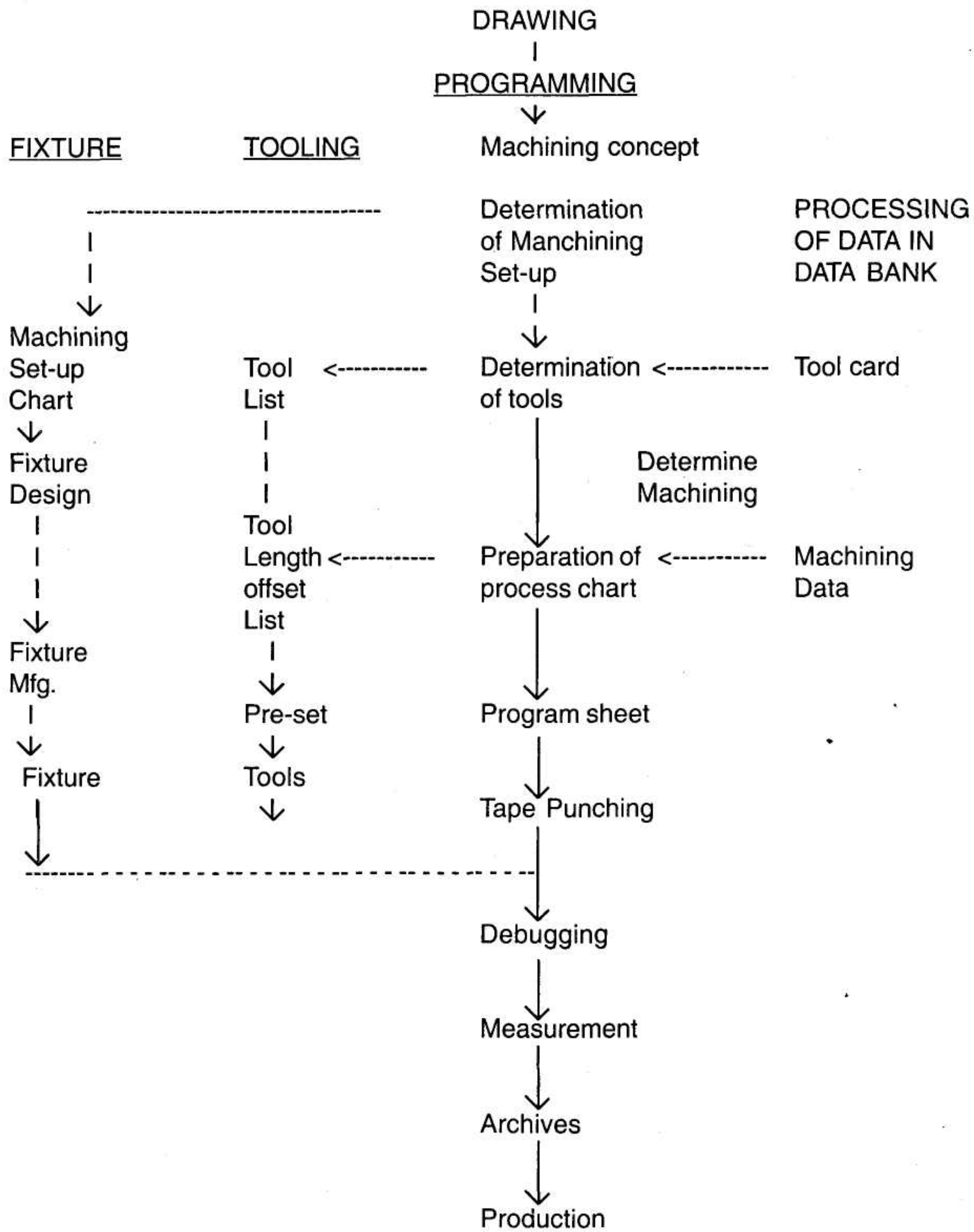
OPERATING MODE SELECTOR SWITCH



This rotary switch with 13 latched positions enables you to select the following operating modes:

Symbol on the selector switch	Function	Designation of the operating mode
	Acutal value setting	Preset Preset setpoint (1st position)
	Manual data Input/automatic	MDI AUTOMATIC Manual Data input Automatic (2nd and 3rd position)
	FEED/JOG	JOG Jogging (4th position)
1 10,100, 1000,10 000	Incremental Traverse	Inc. FEED.... Incremental Feed (5th position)
	Repositioning Reapproach contour	REPOS Reposition (10th position)
	Automatic operation Processing of stored programs	AUTOMATIC (11th and 12th position)
	Traverse to Reference point	REFPOINT Reference Point (13th positon)

NC MACHINING SYSTEM - A CONCEPT



WHAT IS PART PROGRAMMING?

NC machine receives information through a punched tape or floppy disk.

The tape is prepared in accordance with a program manuscript written for the operation to be carried-out on CNC machine to produce the job.

The program is prepared by listing the coordinate values of the entire tool path as suited to machine, to complete machining.

The coordinate values are prefixed with preparatory functions indicating the type of movement required, point to point, straight or circular from one coordinate to another.

Also the coordinates are suffixed with miscellaneous codes for initiating machine tool functions like start, stop spindle movement, coolant on / off etc.

In addition to these coded functions, spindle speeds, feeds and the required command for the machine to execute and it is called a block of information.

All these elements represent a line of information and form one meaningful command for the machine to execute and it is called a block of information.

A number of blocks written sequentially form a part program. Program for the particular component.

SECTION 4

PREPARATORY FUNCTIONS
&
MISCELLANEOUS FUNCTIONS

EXAMPLES
OF
PREPARATORY FUNCTIONS / G CODES
MISCELLANEOUS FUNCTIONS/M CODES

AS APPLICABLE TO SOME OF THE MACHINES AT OFMK

1. M/C HMT CNC LATHE STC 25 AND SBCNC
   CONTROL SYSTEM SINUMERIK 3T 4/4B

2. M/C KWS CNC LATHE WSU-12
   CONTROL SYSTEM FANUC 11T A

3. M/C COOPER VERTICAL MACHINING CENTER VA50
   CONTROL SYSTEM FANUC 11M A

4. M/C HMT BERARDI HORIZANTAL MACHINING CENTER
   CONTROL SYSTEM SINUMERIK 850 M

MISCELLANEOUS FUNCTIONS M/C : HMT STC 25 & SB CNC LATHE
SYSTEM : SINUMERIK 3T 4/4B

Sl.No.	Group	Code	Format	Value Limitation	Function
1	M1	MOO	02		Program Stop unconditional
2		M02			End. of Program (Alone in block)
3		M17			End. of Subroutine (in last Block)
4		M30			End of program (Same as M02)
5	M2	M03			Spindle start- CW
6		M04			Spindle start - CCW
7		M05			Spindle stop
8		M19		0.5 to 359.5 deg. in steps of 0.5 deg.	Oriented spindle stop and angle with S address
9	M3	M08			Coolant motor on
10		M09			Coolant motor off
11		M20			Chuck high pressure OD Clamp
12		M21			Chuck high pressure ID Clamp
13		M22			Chuck Low pressure OD Clamp
14		M23			Chuck Low pressure ID Clamp
15		M24			Spindle neutral
16		M25			Spindle speed range-1
17		M26			Spindle speed range-2
18		M27			Spindle speed range-3
19		M37			Chip conveyer on
20		M38			Chip conveyer off
21		M40			Tailstock quill forward
22		M41			Tailstock quill reverse
23		M44			Tailstock declamp+plunger lock + quill reverse
24		M45			Tailstock declamp+plunger unlock + quill forward
25		M46			Tailstock inhibit on
26		M47			Tailstock inhibit off
27		M48			Steady rest jaws close
28		M49			Steady rest jaws open

NOTE:
1) M05 Reset state (or after M02/M30, Control on)
2) M19 Block wise all others self retaining.

PREPARATORY FUNCTIONS **M/C : HMT STC 25 & SB CNC LATHE**
SYSTEM : SINUMERIK 3T 4/4B

SI.No.	Group	Code	Format	Value Limitation	Function
1	G1	GOO	02		Rapid traverse
2.G01					Linear interpolation
3		G02		I&K	Circular interpolation - CW
4		G03		o to +9999.999	Circular interpolation - CCW
5		G33		0.00+to 400	Thread cutting
6	G2	G-04		0 to +99999.999	Dwell time under X address (Seconds)
7.	G3	G09			Reduced speed, Exact stop
8.	G5	G40			Cancel tool tip radius compensation
9		G41			Tool tip radius compensation-left
10		G42			Tool tip radius compensation-right
11.	G7	G53			Suppress the zero offsets
12.	G8	G54			Settable zero offset-1
13		G55			Settable zero offset-2
14		G56			Settable zero offset-3
15		G57			Settable zero offset-4
16.	G9	G59			Programmable zero offset (additive)
17	G10	G70			Inch Data input (inch)
18.		G71			Metric data input (mm)
19.	G12	G90	I	X&Z	Absolute data input
20		G91	I	0 to+ 99999.99	Incremental data input
21	G13	G92		1to9999RPIVi	Spindle speed limit under S address
22	G14	G94		1 to 1500 mm/min	Feed per mini
2 3 ,		G95		.001to50mm/min	Feed per rev / constant RPM
24		G96		S in meters/min.	CSS under S address & feed/rev.
25		G97			Suppress CSS (G96)

NOTE :1) G04, G59, G92: No other G code can be written in that block.

2) G01, G40, G54, G90, G95, G71 Reset state (after M02/M30 and control switch on)

3) G04, G09, G53, Block wise, all others self retaining.

MISCELLANEOUS FUNCTIONS **M/C : KWS CNC LATHE**

System: FANUC 11T

Sl.No.	Code	Format	Function
1	MOO	02	Programme stop unconditional
2	M01		Programme Stop conditional
3	M02		End of Programme
4	M03		Spindle on - CCW
5	M04		Spindle on - CW
6	M05		Spindle stop
7	M08		Coolant on
8	M09		Coolant off
9	M10		Collect or chuck close
10	M11		Collect or chuck open
11	M12		Low chucking pressure
12	M13		Regular chucking pressure
13	M20		Quill left
14	M21		Quill right
15	M22		Reverse indexing
16	M23		Forward indexing
17	M24		Steady rest (#1) Open
18	M25		Steady rest (#1) Close
19	M26		Steady rest (#2) Open
20	M27		Steady rest (#2) Close
21	M28		Parts catcher open
22	M29		Parts catcher close
23	M30		Programme end and rewind
24	M37		Ignore up to speed feed hold signal
25	M40		Dry run without spindle running
26	M41		Dry run with spindle running
27	M48		Cancel M49
28	M49		Feed rate and spindle speed override inactive
29	M90		In position chuck on
30	M91		In position chuck off
31	M92		Soft jaws boring
32	M96		Pullout in lead enable
33	M97		Pullout in lead disable
34	M98		Subprogram Call
35	M99		Subprogram end

PREPARATORY FUNCTIONS **M/C : KWS CNC LATHE**

System: FANUC 11T

Sl.No.	**Code**	**Function & Meaning**
1	GOO	Rapid traverse (Positioning)
2	G01	Linear interpolation
3	G02	Circular interpolation - CW
4	G03	Circular interpolation - CCW
5	G04	Dwell
6	G20	Cutting Cycle A
7	G21	Thread cutting cycle
8	G22	Stored stroke limit on
9	G23	Stored stroke limit off
10	G29	Cutting cycle B
11	G33	Constant lead threading
12	G40	Tool nose radius compensation Cancel
13	G41	Tool nose radius compensation -Left
14	G42	Tool nose radius compensation -Right
15	G70	Inch data input (inch)
16	G71	Metric data input (mm)
17	G72	Finishing cycle
18	G73	Stock removal in turning
19	G74	Stock removal in facing
20	G75	Pattern repeating
21	G76	Peck drilling
22	G77	Grooving x axis
23	G78	Thread cutting cycle
24	G90	Absolute data input
25	G91	Incremental data input
26	G92	Position preset
27	G94	Feed in mm per minute/inch per minute
28	G95 .	Feed in mm / inch per revolution
29	G96	Constant surface feed on
30	G97	Constant surface feed off

MISCELLANEOUS FUNCTIONS- M/C : HMT MACHINING CENTER ALT 161
System : SINUMERIK 850M

Sl.No.	Code	Function	A	B	C
1	MOO	Unconditional Programmed stop	X		X
2	M01	Conditional programmed stop	X		X
3	M02	End of program	X		X
4	M03	Spindle rotation - CW		X	
5	M04	Spindle rotation - CCW		X	
6	M05	Spindle rotation stop			X
7	M06	Tool change		X	
8	M07	Coolant on (Through spindle)		X	
9	M08	Coolant on (External)		X	
10	M09	Coolant off		x	
11	M10	Permits that the X,Y,Z axes, at the end of motion, stop and release from the NC.			X
12	M11	Reset of M10 (Forced on CNC starting)		x	
13	M12	Enable work feed with spindle stopped		x	
14	M13	Reset M12 (Forced on CNC startup)		x	
15	M17	End of subroutine			x
16	M19	Spindle stop oriented under S address			x
17	M27	Coolant included during the rapid feed of X-Y-Z-W-B. It is erased by M59 interrupted by stop		x	
18	M29	Exclusion of Z axis retraction (locking X axis to NC). It permits Z axis motion with preferential tool in spindle	x	x	
19	M30	Program end with rewinding	x		x
20	M36	Reset of M37		x	
21	M37	Feed rate down rated by 1:: 100		x	
22	M38	Quill retraction in reduced torque. It operates only with L907 spindle oriented		x	
23	M40	Reset of forced range (M42) Forces an automatic range change		x	
24	M42	Forces the second range		x	

SI.No.	Code	Function	A	B	C
25	M52	Erases spindle speed control reached (Program in constant cutting speed)	x	x	
26	M56	Coolant blown out off the spindle before tool change Carried out during block execution time (block in which it was programmed)	x	x	
27	M59	Erases M27		x	
28	M61	Pallet loading		x	
29	M62	Pallet unloading		x	
30	M69	Pin rise for indexed position of axis B		x	
31	M75	Cooling pump at high press, on		x	
32	M76	Reset of M75		x	
33	M84	Enabling of tool selection after using the emergency push-button panel for tools.	x	x	

NOTE: A -This function operates within the block in which it was programmed. It erases itself at the end of the block.

B- This function is on at the beginning of the block.

C- This function is on at the end of the block.

PREPARATORY FUNCTIONS- M/C : HMT MACHINING CENTER ALT 160
System: SINUMERIK 850M

Sl.No.	Code	Function	Note
1.	GOO	Rapid traverse	Stored
2	G01	Linear interpolation	Stored
3	G02	Circular interpolation -CW (Note-1)	Stored
4	G03	Circular interpolation-CCW (Note-1)	Stored
5	G04	Dwell X or F in sec, S in rev (Note-2)	Self-Erase
6	G09	Exact stop	Self-Erase
7	G10	Polar coordinate Rapid traverse	stored
8	G11	Polar coordinate linear interpolation	Stored
9	G16*	Plane selection with axes choice	Stored
10	G17	Selection of X-Y work plane	Stored
11	G18	Selection of z-x work plane	Stored
12	G19	Selection of y-z work plane	Stored
13	G25	Min. machining area limit (Note-2)	Self-Erase
14	G26	Max. machining area limit (Note-2)	Self-Erase
15	G33	Threading	Stored
16	G34	Threading with lead increasing step	Stored
17	G35	Threading with lead decreasing step	Stored
18	G40*	Deactivation of Cutter radius compensation.	Stored
19	G41	Tool radius offset to left	Stored
20	G42	Tool radius offset to right	Stored
21	G53	Deactivation of zero offsets	Self-Erase
22	G54	Settable zero offset-1 (Note-2)	Stored
23	G55	Settable zero offset-2 (Note-2)	Stored
24	G56	Settable zero offset-3 (Note-2)	Stored
25	G57	Settable zero offset-4 (Note-2)	Stored
26	G58	Programmable zero offset (Note-3)	Stored
27	G59	Programmable zero offset (Note-3)	Stored
28	G60	Exact stop	Stored
29	G62	Continuous working block passage with deceleration	Stored
30	G63	Threading without encoder (Note-4)	Stored
31	G64	Continuous working block passage without deceleration	Stored

SNo	Code	Function	Note
32	G68	Rotary axis module for the shortest stroke (Note-5)	Self-Erase
33	G70	Inch data input	Stored
34	G71	Metric data input	Stored
35	G80*	Reset of canned cycle	Stored
36	G81	Drilling/Spot drilling cycle	Stored
37	G82	Drilling-Countersink cycle	Stored
38	G83	Drilling cycle	Stored
39	G84	Tapping cycle (Note-6)	Stored
40	G85	Reaming cycle	Stored
41	G86	Reaming cycle	Stored
42	G87	Reaming cycle	Stored
43	G88	Reaming cycle	Stored
44	G89	Reaming cycle •	Stored
45	G90	Absolute data programming	Stored
46	G91	Incremental data programming	Stored
47	G92	Limit on Spindle RPM Under R address (Note-3)	Self-erase
48	G94	Feed rate in mm/min	Stored
49	G95	Feed rate in mm/rev	Stored
50	G96	Constant cutting speed	Stored
51	G97	Reset of G96	Stored

NOTE: * Functions forced on NC, on starting or after M30 and reset

G92,G96 & G97 are used only at constant speed, with facing devices.

STORED = function which remains stored until an opposite function is operating.

SELF-ERASE = function which is erased at end of the block in which it is programmed.

NOTE-1: G02+Z(W) Clockwise Helical milling
G03+Z (W) Counter clockwise Helical milling

NOTE-2: In the block which follows such functions always program L999 subroutine in order to empty the intermediate memory Example: - G54 L999

NOTE-3: It is not possible to programme other functions in the blocks where these functions already exist.

NOTE-4: G63 function is not used since controlled tapping with G84 function provided.

NOTE-5: G68 function is used in subroutine of tool magazine motion, L800. G68 is active only if the considered axis has been indicated, in the machine data, as RO TARY axis and as MODULE 0-360 degrees.

NOTE-6: G84 canned tapping cycle has been changed from the original version in order to improve the cycle times and allow tapping to take place without regard to the programmed S values. The M33 function, which is decoded by the PLC, ii programmed in the subroutine-and checks that the spindle has reached the proper number of revolutions before allowing the program to continue.

PRORAMMING NOTE:

Use U address for directly programming circle radius or polar coordinates. I address followed by the proper value is used to program the radius, while > address followed by the proper value is used to program the angle when programming with polar coordinates.

MISCELLANEOUS FUNCTIONS - MACHINE : VERTICAL MACHINING CENTER-VA50
SYSTEM :FANUC11M

Sl. No.	Code	Function	Operation of Machine	Operational Classification
1	MOO	Programme stop	Spindle stop coolant off	A
2	M01	Optional stop		A
3	M02	End of programme	Control unit reset	A
4	M03	Spindle on - CW		B
5	M04	Spindle on - CCW		B
6	M05	Spindle stop		A
7	M06	Tool change	Automatic tool change	B
8	M07	Mist on		B
9	M08	Coolant on		B
10	M09	Mist/coolant off		A
11	M19	Spindle orientation		B
12	M30	End of programme tape rewind	Control until reset tape rewind	A
13	M48	Over ride cancel / release		A
14	M49	Over ride cancel		A
15	M98	Subprogram callout		A
16	M99	End of subprogram		A

NOTE : In operational classification "A", the auxiliary function command gets started after the completion of the movement command. In class 'B' both commands start simultaneously Mark * indicates optional.

PREPARATORY FUNCTIONS MACHINE: VERTICAL MACHINING CENTRE. VA5
SYSTEM: FANUC 11M

Sl.No.	GROUP	CODE	FUNCTION/MEANING
1	01	GOO	Rapid traverse (Positioning)
2		G01	Linear interpolation
3		G02	Circular interpolation - CW
4		G03	Circular interpolation - CCW
5	00	G04	Dwell
6		G09	Exact Stop check
7		G10	Work zero offset value setting
8	02	G17*	X-Y Plane selection
9		G18	Z-X Plane selection
10		G19	Y-Z Plane selection
11	06	G20	Inch data input
12		G21*	Metric data input (mm)
13	04	G22	Stored stroke limit on
14		G23*	Stored stroke limit off
15	00	G27	Reference point return check
16		G28	Return to reference point
17		G29	Return from reference point
18		G30	Return to 2nd, 3rd, 4th reference point
19		G31	Skip function
20	01	G33	Thread cutting
21	07	G40	Cutter radius compensation cancel
22		G41	Cutter radius compensation - Left
23		G42	Cutter radius compensation - Right
24	08	G43	Tool length compensation + direction
25		G44	Tool length compensation - direction
26		G49*	Tool length compensation cancel
27	09	G45	Tool offset expansion
28		G46	Tool offset reduction
29		G47	Tool offset double expansion
30		G48	Tool offset double reduction

SL	GROUP	CODE NO	FUNCTION/MEANING
31	12	G54	Selection of work coordinate system-1
32		G55	Selection of work coordinate system -2
33		G56	Selection of work coordinate system -3
34		G57	Selection of work coordinate system -4
35		G58	Selection of work coordinate system -5
36		G59	Selection of work coordinate system -6
37	00	G60	Unidirectional positioning
38	13	G61	Exact stop check mode
39		G64	Continuous cutting mode
40	00	G65	User Macro simple call out
41	14	G66	User Macro modal call out.
42		G67	User Macro modal call out cancel
43	09	G73	Peck drilling cycle
44		G75	Counter tapping cycle
45		G76	Fine boring cycle
46		G80*	Canned cycle cancel
47		G81	Drilling cycle spot boring
48		G82	Drilling cycle counter boring
49		G83	Peck drilling cycle
50		G84	Tapping cycle
51		G85	Boring cycle
52		G86	Boring cycle
53		G87	Back boring cycle
54		G88	Boring cycle
55		G89	Boring cycle
56	03	G90	Absolute data input
57		G91*	Incremental data input
58	00 .	G92	Programming of absolute zero point
59'	05	G94*	Feed per minute
60		G95	Feed per revolution
61	10	G98*	Return to initial point in canned cycle
62		G99	Return to R point in canned cycle

NOTE:

G codes marked * are set when the power is turned on or after pressing reset button. G00 or G01, G43.G44 or G49: G90 or G91; and G94 or G 95 can be selected with parameter setting. G20 or G21 that was specified before turning the power on is set. G codes of group '00' are non modal and are effective only in block where they are specified.

Cautions concerning G codes :

1) An alarm is displayed if G codes, which are not listed in G code table or G codes for unprovided options are specified.

2) Many G codes of different groups can be specified in the same block.
When two or more G codes of the same group are specified in the same block, the one specified last is effective.

3) When the G code of group 01 is specified, when the canned cycle is executed, the canned cycle is cancelled automatically to the state of G80. However G codes of group 01 are not affected by G codes of canned cycles.

PROGRAMMING PROCEDURE

Step 1 : Select reference point, program zero.

Step 2 : Determine coordinates (Absolute or chain dimensions, zero offset?)

Step 3 : Prepare working plan-determine step by step
- * tool motions
- * feed rates
- * spindle speeds
- * tools used
- * coolant supply
- * etc.

Step 4 : Write program-translate operating steps into programming language.

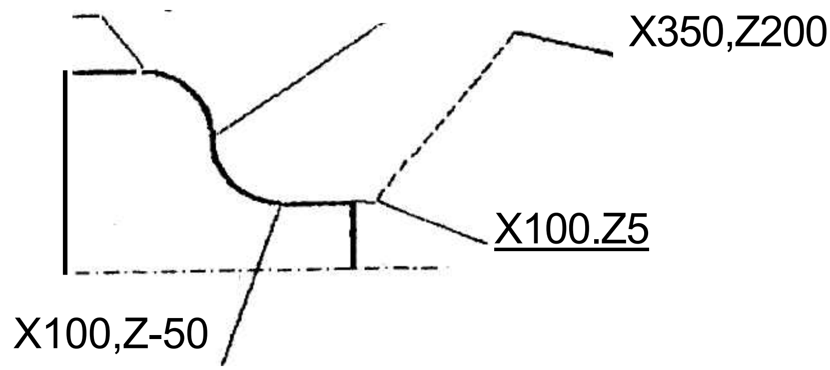
Step 5 : Key in program

Step 6 : test and edit program

Step 7 : start auto cycle

Step 8 : archive proved program

MOVEMENT COMMANDS:



GOO. G01. GQ2. G03& G33

GOO: RAPID POSITIONING

Move tool from X350. Z-200 to X 100, Z5 Since
Tool moves in air, the code used is GOO

N30G00X100Z5LF

G01 : LINEAR TRAVERSE
(Straight line movement at set feed rate)

Next tool is moved to X100.Z-50 .

Since tool cuts the material, code used is G01.

N40G01 X100Z-50 F0.2 LF

GO2: CIRCULAR INTERPOLATION (CLOCK-WISE)

Next tool is moved to X200.Z-100.
Since tool moves along arc in cock-wise direction, code used is G02.

N50 G02 X200 Z-100 150 K0 LF.

I & K ARE CALLED AS INTERPOLATION PARAMETERS. REPRESENTING COORDINATE VALUES OF ARC/CIRCLE CENTER FROM THE START POINT OF THE ARC / CIRCLE, PARALLEL TO X & Z AXIS RESPECTIVLY.

THEIR VALUES ARE ALWAYS GIVEN IN INCRIMENTAL DATA INPUT SYSTEM

ARCS CAN BE PROGRAMED BY IT'S RADIUS ALSO

N50 G02 X200 Z-100 B50 LF

NOTE: If included angle of arc at center is less than OR equal to 180 degrees, radius value will be positive (B50)

If included angle of arc at center is more than 180 degrees, than radius value will be negative (B-50)

G03: CIRCULAR INTERPOLATION
(COUNTER CLOCK-WISE)

Next tool is moved to X300. Z-150.
Since tool moves along ARC in counter clock-wise direction, code used in G03.

N50 G03 X300 Z-150 10 K-50 LF
OR N50 G03 X300
Z-150 B50 LF

G33: THREAD CUTTING

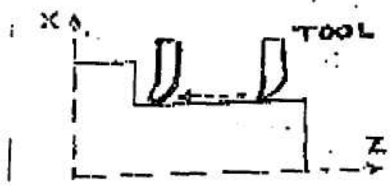
N60 GOO X99 Z5 LF
N61 G33 Z.50 K2 LF
N62G00X102LF
N63 Z5 LF
N64 X99.5 LF
N65 G33 Z-50 K2 LF

Program sequences follows the pattern till last depth of cut.

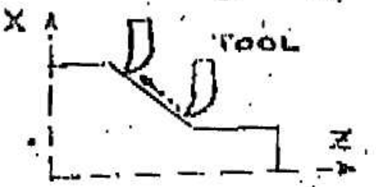
ADDRESS K REPRESENTS THE PITCH OF THE THREAD FOR SINUMERIK- SYSTEM & *F' FOR FANUC SYSTEM Feed programmed will be suppressed during G33 code execution

LATHE

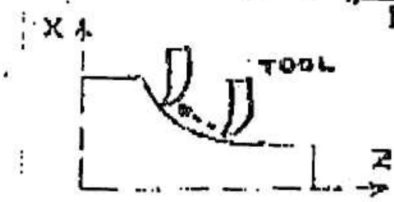
STRAIGHT : G01 Z_{EP} . .



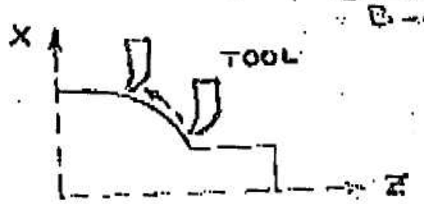
TAPER : G01 X_{EP} . . Z_{EP} . .



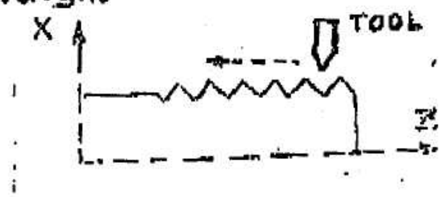
ARC (CW) : G02 X_{EP} . . Z_{EP} . . I_{SP} . . K_{SP} . . / B . .



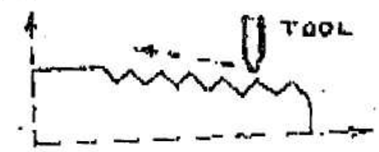
ARC (CCW) : G03 X_{EP} . . Z_{EP} . . I_{SP} . . K_{SP} . . / B . .



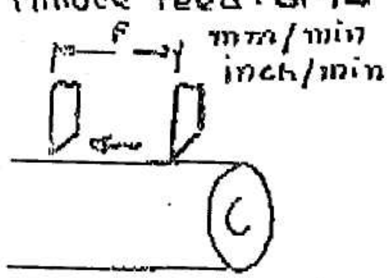
THREAD : G33 Z_{EP} . . . K_{L} . . .
Straight



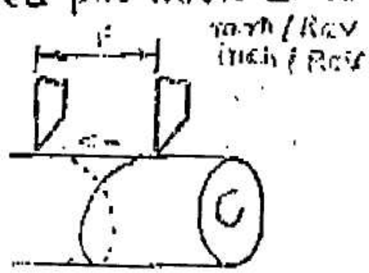
THREAD : G33 X_{EP} . . Z_{EP} . . I . . K_{L} . .
Taper



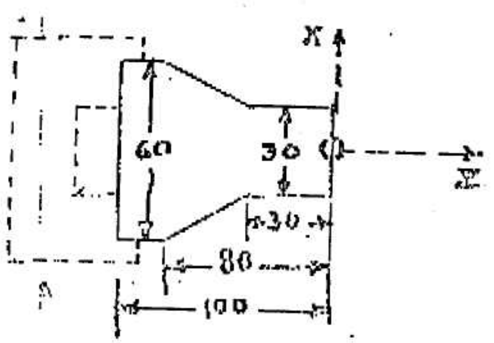
PER Minute Feed : G94

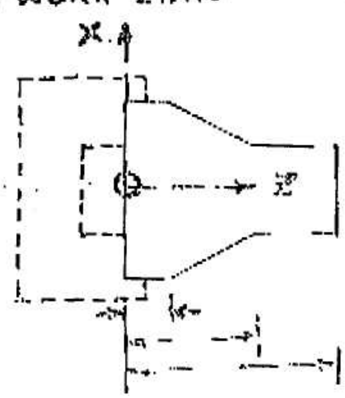


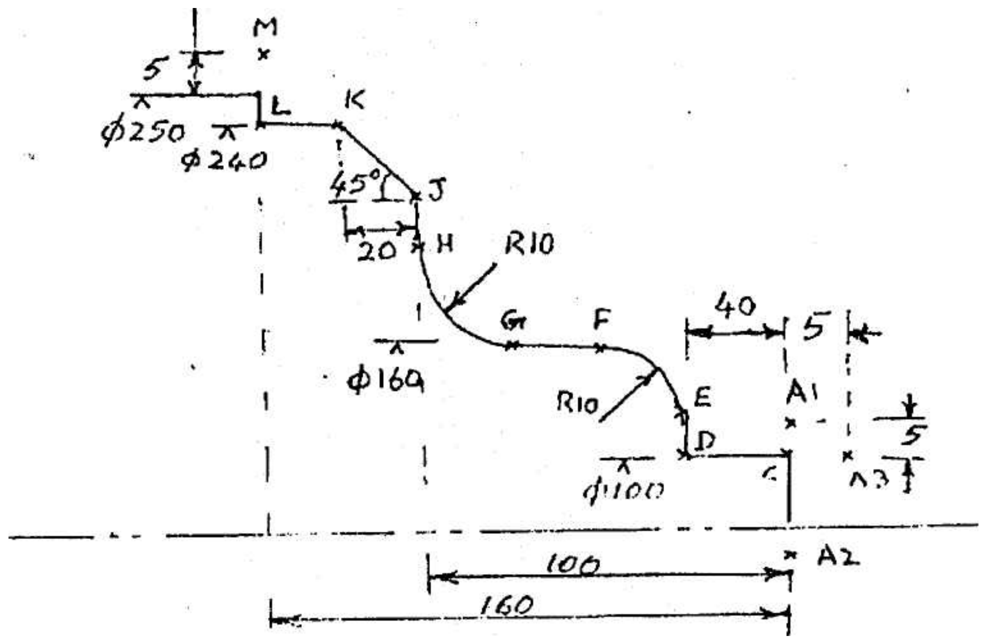
Feed per Rev. : G 95



COORDINATE / WORK ZERO

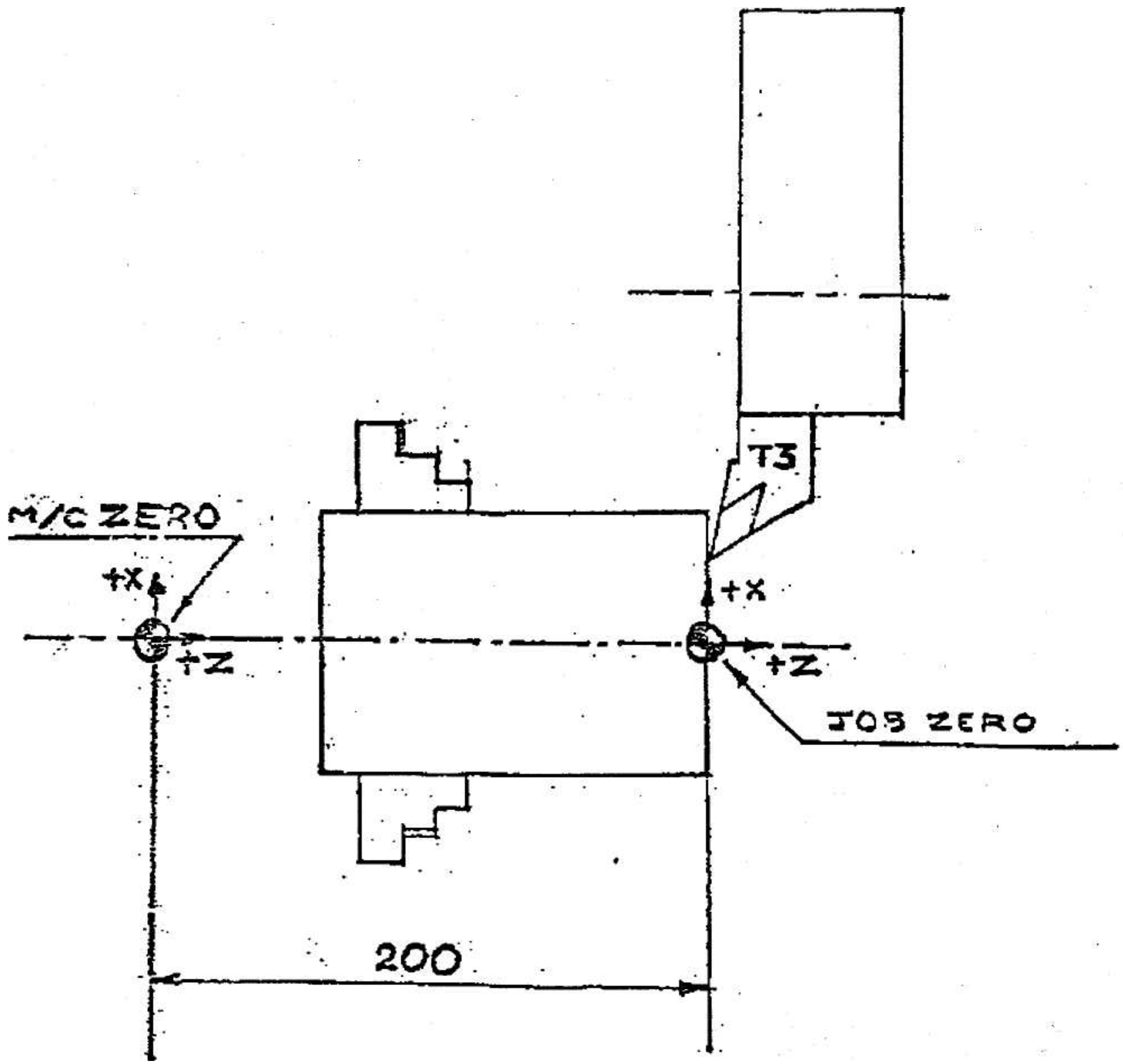






	X	Z	Program	Comment
			%200*	Program Number/start
	X	Z	N10G90G71 G54 G40 G95*	Data for control
	-------------------		N20 M 27*	Speed range selection
T.C.	300	100	N30G0X300 Z100*	
			N40M04 G96 S140 T0303 *	Spindle Rotation
			N50 G92 S800*	Max. RPM control
A1	100+10	0	N60G0X110Z0*	
A2	-1	0	N70G1X-1Z0 F0.3 M08 *	
A3	100	5	N80 G1 X100 Z5 F10 G42*	
D	100	-40	N90G1 X100Z-40F0.3*	
E	140	-40	N100G1 X140 Z-40 *	
F	160	-50	N110 G3 X 160 Z-50 I0K-10*	
G	160	-90	N120G1X160Z-90	
H	180	-100	N130 G2X180Z-100 I10K0*	
J	200	-100	N140 G1 X200 Z-100*	
K '	240	-120	N150G1 X240Z-120*	
L	240	-160	N160 G1 X240 Z-160*	
M	260	-160	N 170 G1 X260 Z-160 M09*	
T.C.	300	100	N180G0 X300 Z100 G40 M05*	

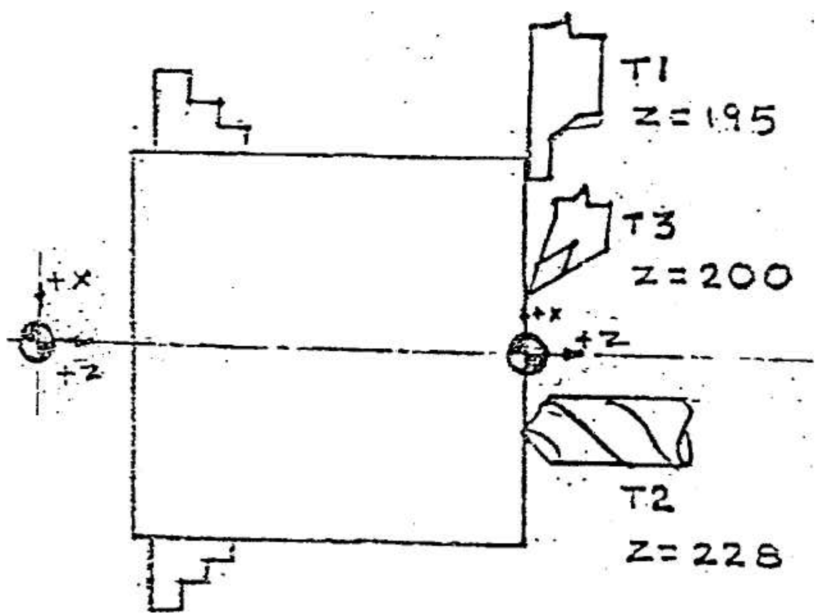
PROGRAM END N190 M30*



ZERO OFFSET

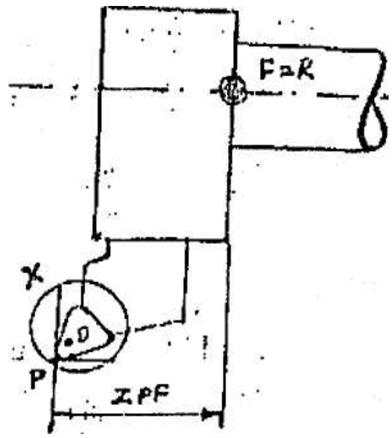
1. X 0 = 200

Z OFFSETS



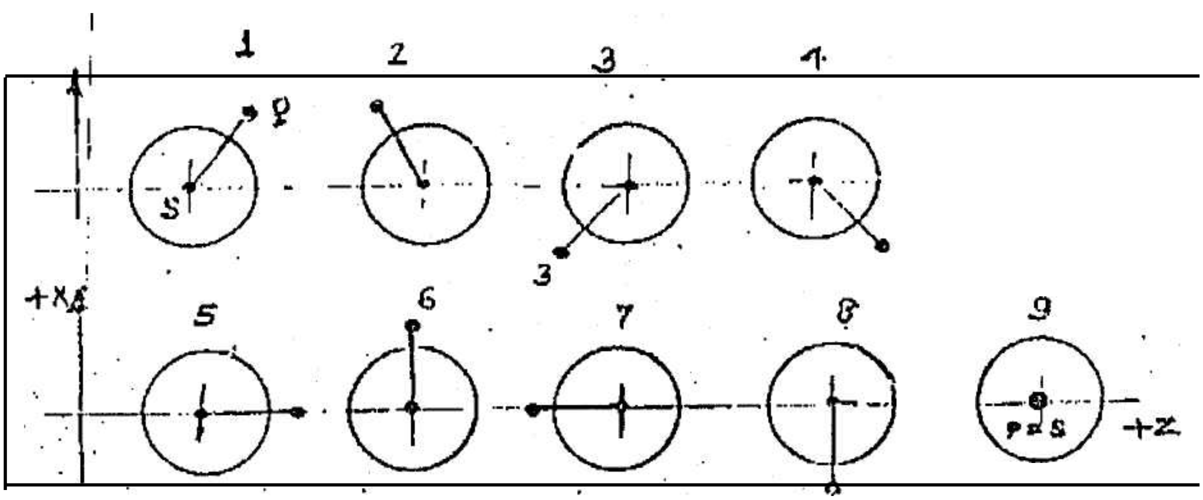
TOOL NO	DISPLAY VALUE	Z OFFSET	
T1	195	-5	(195-200=-5)
T2	228	28	(228-200=28)
T3	200	0	(200-200=0)
T4			

TOOL OFFSET



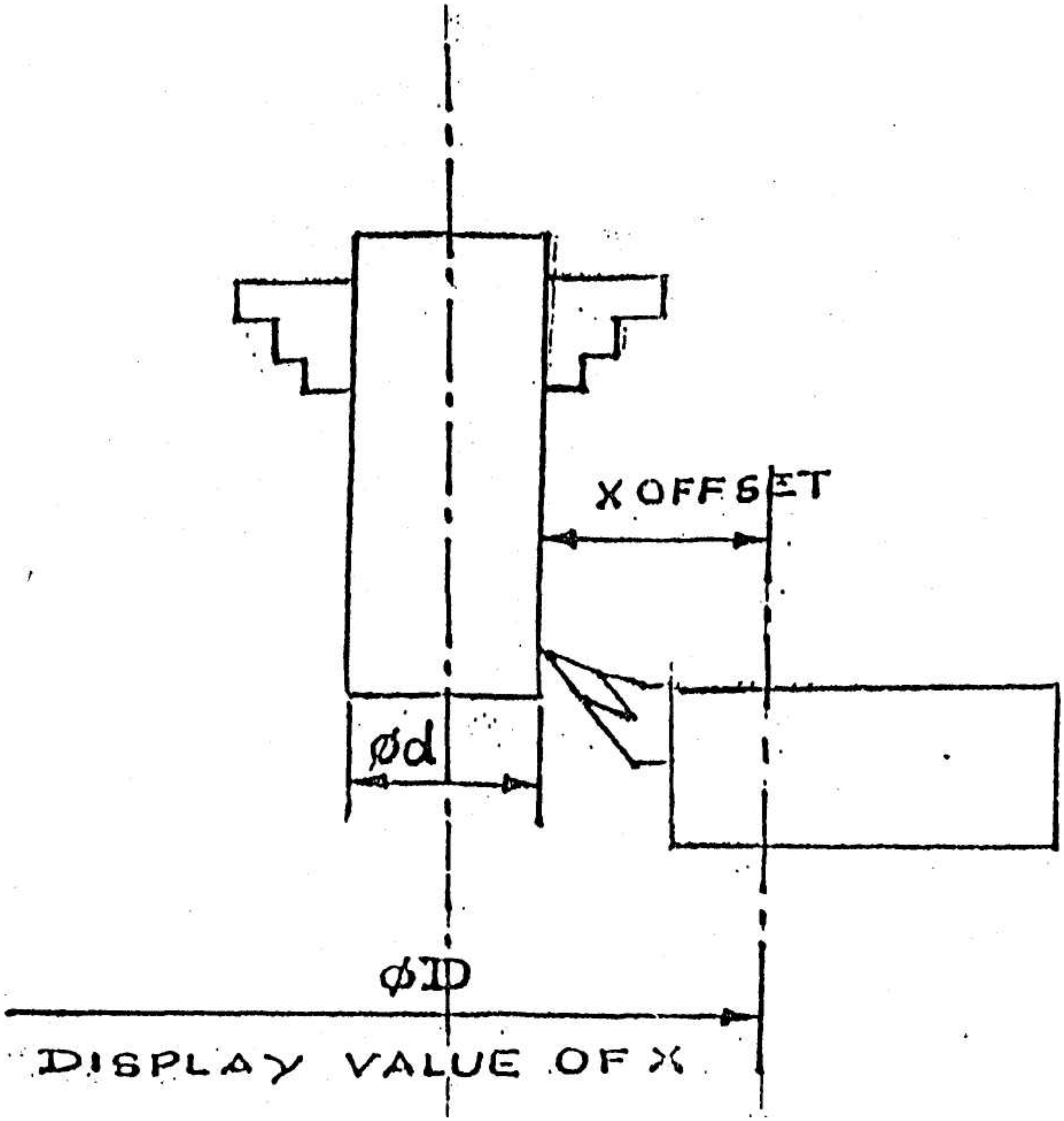
DETAIL X

P = Theoretical Tip Point
S = Toot tip Radius Centre Point
RS = Tool Tip Radius



TOOLOFFSET					
I	1	X+	0	Z+	0
		B+	0	A+	0
T	2	X+	0	Z+	0
		B+	0	A+	0
J	3	X+	250.00	Z1	175.55
		B+	0.80	A	3

X OFFSET = D – (d / 2)



SECTION 5

PARAMETRIC PROGRAMMING

PARAMETRIC PROGRAMMING

R-PARAMETERS

Parametric programming provides more flexibility for the programmer.

Part programs may be programmed with parameters instead of numerical values for the dimension data. A parameter is set equal to a numerical value in the part program or in subroutine.

Number of R parameters provided depends on CNC system. Refer to word address format of the Sinumerik system 3T for R parameter - R2 RL+053. R parameter must always be programmed as 2-digit number. A maximum of 10 parameters are allowed in a program block. Parameter R00 thru R99 can be assigned to all addresses except N.

Example:

```
L 6100 LF               parameters R01, R07,
N1 X-R07 Y R01 LF      and R30 are used in
N2 Z R30 LF             in this subroutine.
N3 X R07 Y-R01 LF
N5M17LF
```

Parameters are first defined and then assigned.

Example:

```
   %2364 LF
   N1 M26 LF
   N2 G96 S180 M04 T0303 LF
»  N3 RIO 200 R03 5 LF        DEFINING
   N4 GOO X RIO Z R03 LF    ASSIGNING
```

DEFINITION

By definition, individual parameters set equal to signed numerical .values. The definition of R parameter can be made in the part program and or subroutine.

Programmed	**Results**
R01 20.55	R01 = +20.55
R02 -35	R02 = -35
R03 10	R03=10

ASSIGNMENT

R parameter can be assigned in two ways

* Direct assignment
* Additive assignment

In direct assignment, an address is assigned directly to the value defined for an R parameter

Programmed	Results
X R01	X = +20.5
Z R02	Z = -35
X-R03	X = -10

In additive assignment, the defined value of an R parameter with the correct sign is assignment to the digital value of an address.

Programmed	**Results**	
X 30.5-R01	X = 10	The sequence of address.
Z 50-R02	Z = 85	numeric value, parameter
X 20 R03	X = 30	must be observed.

PARAMETER LINKING

Arithmatic sign

+ Addition
- Subtraction

* Multiplication /

Division

Assign N11 RIO 20 R13 5 LF

Add N12RI0R13LF

Adding R13 value to R10 value and changing the R10 value to the new value, R13 value remaining the same.

R10 20 *
R13 5
25 NowR10 = 25andR13 = 5

Sub N13 R10-R13 LF

R10 25
R13 5
20 Now R 10 = 20 and R 13= 5

Multip N14R10 R13LF

R10 20
R13 x 5
100 NowR10 = 100andR13 = 5

Div. N15R10/R13LF

R10 100
R13 -i-5
20 NowR10 = 20andR13 = 5

Define & Add

N16 R20 3 R72 18 LF
N17R20 12R72LF In block N17, R20 value is redefined to new value and added R72. Now R 20 = 30 and R72 = 18

PARAMETER CHAINING

N99 R01 20 R02 5 R03 4 R04 2 LF
N100 L100 03*

L100*
N1 G X 20 R01 - R02 *R03 / R04 LF

	X	**R01**	**R02**	**R03**	**R04**
N99	20	20	5	4	2
1st	40	15	20	2	2
2nd	35	- 5	40	1	2
3rd	15	-45	40	5	2

Maximum possible chaining four fold.

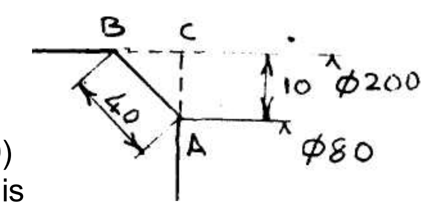
@10 SQUARE ROOT

Application : Extract square root

N10 R 10 25 LF	R10 is loaded with 25
N15 @10 R10 LF	Extract square root of value defined in R10
Now R10=5	Result to be loaded back in R10

Example:

N 100 G1X180LF	R12 10R1310LF
N101 R10 40R1140	(40X40 = 1600)
N102 R1O*R11LF N	(10X10 = 100)
103 R12*R13LF	(1600-100)
N104 R10-R12LF	(Square root of 1500)
N105@ 10R10LF	(R 10 value at N106 is
N106G1 X200Z-R10LF	the value obtained at N105)



@ 15 SINE

Application : Calculate sine of an angle.

. N10 R17 45 LF

- N15 @15 R17LF Now R17 = 0.7071067

sine value of 45 deg.

@18 ARC TAN

Application : Determination of an angle using the Arc tan function.

N10 R10 20R11 30LF
N11 @ 18 R1O LF   20 Divided by 30, then tan inverse value is loaded in R10.

@00   UNCONTIONAL JUMP

Application : Program sections may be omitted using the jump function.
Omitted blocks are not processed.

©00 + 1234

@00   Operation unconditional jump.
Branch destination is located before (-) or after (+) the branch operation is carried-out according to the given direction.

1234   Block No. of the branch destination.

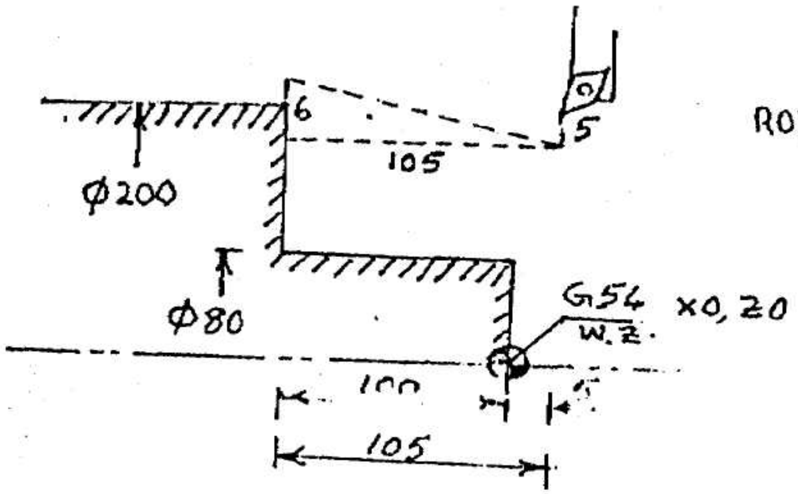
@01,@02, @03   CONDITIONAL JUMP

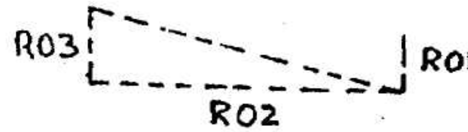
Application   Branch operations are executed according to the following conditions

@01   Equal to
1 st Parameter   =   2nd Parameter

@02   Greater then
1 st Parameter   =   2nd parameter

@03   Greater then or equal to
1st Parameter   >   2nd parameter

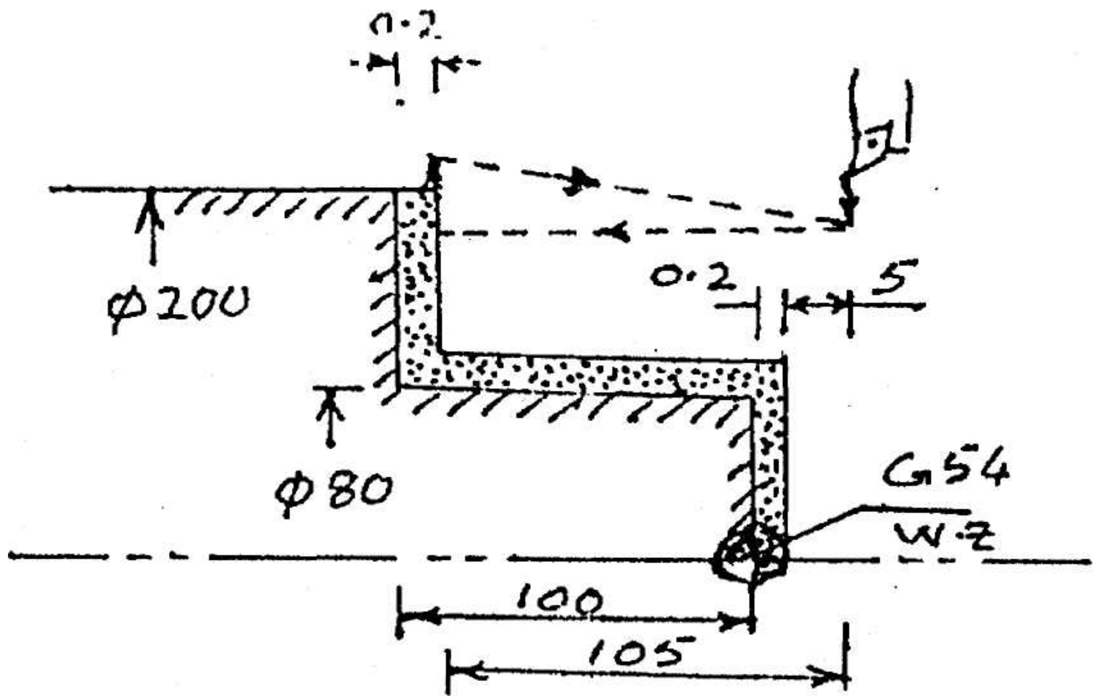




Subprogram	**Alternate subprogam**
L111*	L111*
N1 G 91 G X-5 *	N1 G91 G X-R01 *
N2 G1 Z-105*	N2G1 Z-R02 *
N3 X6 *	N3 X R03 *
N4GX-6Z105*	N4 G X-R03 Z R02 *
N5 G90 M17*	N5G90 M 17*

NOTE : Subprogram has been made in INCREMENTAL SYSTEM.
Main program has been made in ABSOLUTE SYSTEM.
Hence Subprogram must end with Absolute code.

Main program	Alternate Main program
%111*	%111*
N1 G 54 G90 G71 G95 G 40*	N1 G54 G90 G71 G95 G40*
N2 M26*	N2 M26 *
N3 -M04 G96 S180 T0101*	N3 M04 G96 S180 T0101 *
N4 G X200 Z5*	N4 G X200 Z5 *
	N5 R01 5R02 105 R03 6*
N6 L111 12 F 0.5 M8*	N6 L111 12 F0.5 M8*
N7 G X300 Z200 M9 M5*	N7 G X300 Z200 M9 M5*
N8 M30*	N8 M30*



% 112*

N1 M25* Speed range selection.

N2 G96 S180 M4 T0303 (Rough Tool) *

N3 G x200.5 Z5.2 * Position tool to start of cut

N4 R01 5 R02 105 R03 6 R04 12

Depth Z axis Retract No. of cuts

R05 0 R06 1*

Count parameters

N5 G91 G X-R01 F0.5 M8 * 1 st depth of cut N6 G1 Z-R02* 1st cut N7 G X R03* Retract N8 G X-R03 Z R02 * To start of next cut N9 R05 R06* Addition, counting no. of cuts

N10 @01 12R05 R04 * Jump to N12 if R05 = R04 or else continue next block

N11 @00-5* Jump back to N5

N12 G90 GX82* Position to face

N13 Z0.2 * 0.2 Allowance on face
N14 G92 S 800 * Speed control
N15 G1X-2 F0.2 M9 * Facing
N16 G X300 Z200M5* Tool change position
N17 M27 * Speed range selection
N18 G96 S220 T0101 * Finishing tool
N19 GX82 Z0* Position to face
N20 G1 X-2 M8* Facing
N21 G X80 Z2 * Position to turn dia 80
N22 G1 Z-100* Turn to length 100
N23 X 202 M9 *
N24 G X300 Z200 M5*
N25 M30*

SECTION 6

CANNED CYCLES

CANNED CYCLES G81 TO G89

INTRODUCTION :

For standard machining processes which are frequently repeated, machining cycles are provided as permanently stored subroutines by the control system manufacturer in the user memory of the control system.

The canned cycle defines in accordance with DIN 66025, a series of machining sequences for drilling, boring, tapping etc.,

The canned cycles G81 to G89 are stored as subroutines L81 to L89. The parameters R00 to R11 are used by subroutines to define the variable values necessary to correctly execute a fixed cycle.

The fixed cycle call is initiated by G81 to G89, is executed in every NC block till it is cancelled with G80. The selection and cancellation of G81 to G89 may only, be carried out with in a single program level. G81 to G89 cycles are model functions. A boring cycle can be called with L81 to L89, however these are not model and hence performed only once in the NC block in which it is called.

At the end of a fixed cycle execution, the tool is repositioned at the start plane.

The cycles end uniformly with preparatory functions GOO, G60 and G90. Any other G functions required when the program continues must be reprogrammed.

These cycles can be modified if desired to suit the specific machine or tooling requirements, and hence ensure compliance with any additional information provided by the machine tool manufacturer (refer to working instructions / operating and programming instructions of the machine tool manufacturer).

The canned cycles described here-in is based on sinumerik system 3.

R. PARAMETERS USED IN CYCLES L81 TO L89.

R00 Dwell time at the start point (deburr hole).

R01. First depth advance (incremental) entered without sign.

R02 Reference plane or retract plane (absolute).

R03 Final depth (Absolute).

R04 Dwell time at hole bottom (break chips).

R05 Depth advance modifier entered without sign. (Degression).

R06 Reverse spindle rotation direction.

R07 Return to the original spindle rotation direction used in the calling programme (after R06 or M05)

R09 Thread lead dimension.

R10 Retract plane.

R11 Boring axis (axis numbers X=1, Y=2, Z=3).

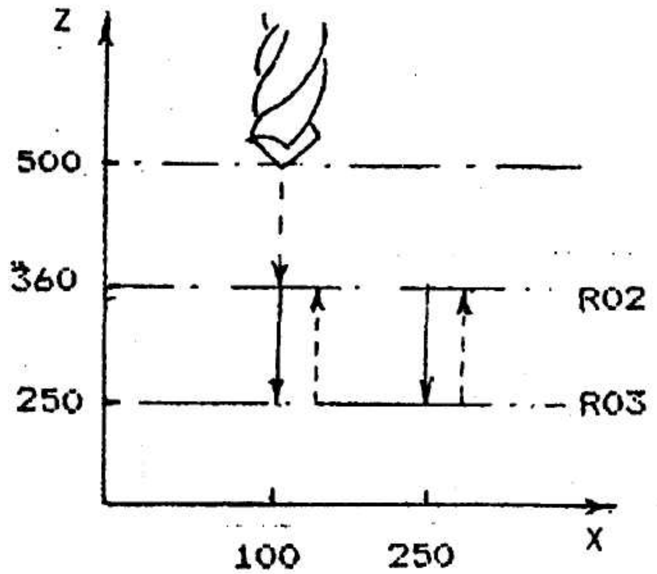
CALLING CYCLE L81 (Drilling, Centering)

The following parameters must be defined prior to calling canned cycle L81.

R02 Reference plane (retract position)

R03 Final hole depth

R11 Drilling axis



CYCLE L81 (Drilling, Centering)

N8101 G90 S48 M03 F460 LF	Spindle on
N8102 GOO D01 Z500 LF	Activate tool off set.
N8103 X100 Y150 LF	First drill position
N8104 G81 R02 360 R03 250 R11 3 LF	Call Cycle.
N8105 X250 Y300 LF	Second drill position and automatic G81 call.
N8106 G80 Z500 LF	Canceling G81 and returning to starting plane.

CALLING CYCLE L82 (Drilling, Counter sinking)

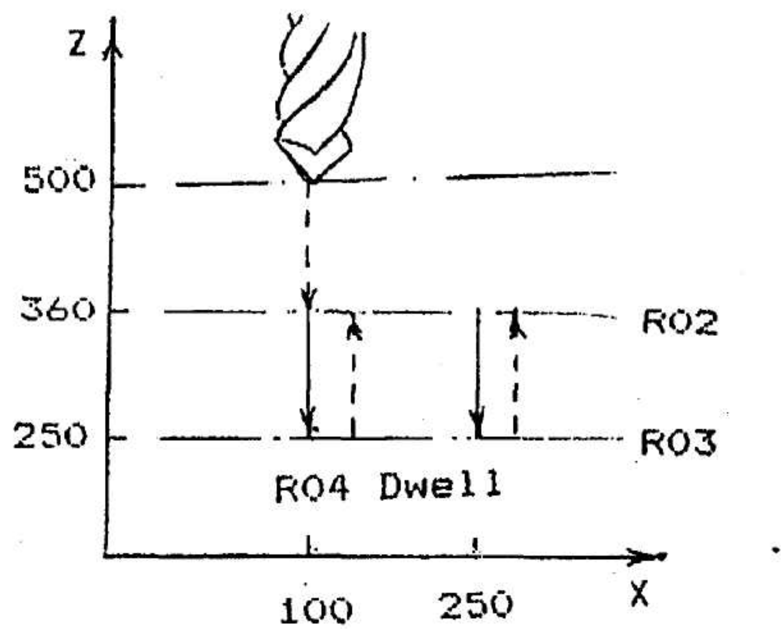
The following parameters must be defined prior to calling canned cycle L82.

R02 Reference plane (retract plane).

R03 Final hole depth.

R04 Dwell time.

R11 Drilling axis.



CYCLE L82 (Drilling, Counter sinking)

N8201 G90 S48 M03 F460 LF

N8202 GOO D01 Z500 LF

N8203X100Y150LF

N8204 G82 R02 360 R03 250 R04 1 R11 3 LF

N8205 X250 Y300 LF

N8206 G80 Z500 LF

CALLING CYCLE L83 (Deep hole drilling)

The following parameters must be defined prior to calling canned cycle L83.

R00 Dwell at starting point.

R01 First drilling depth (incremental)

R02 Reference plane Retract plane (absolute) (Point A)

R03 Final depth of hole (absolute).

R04 Dwell at bottom of hole (chip breaking).

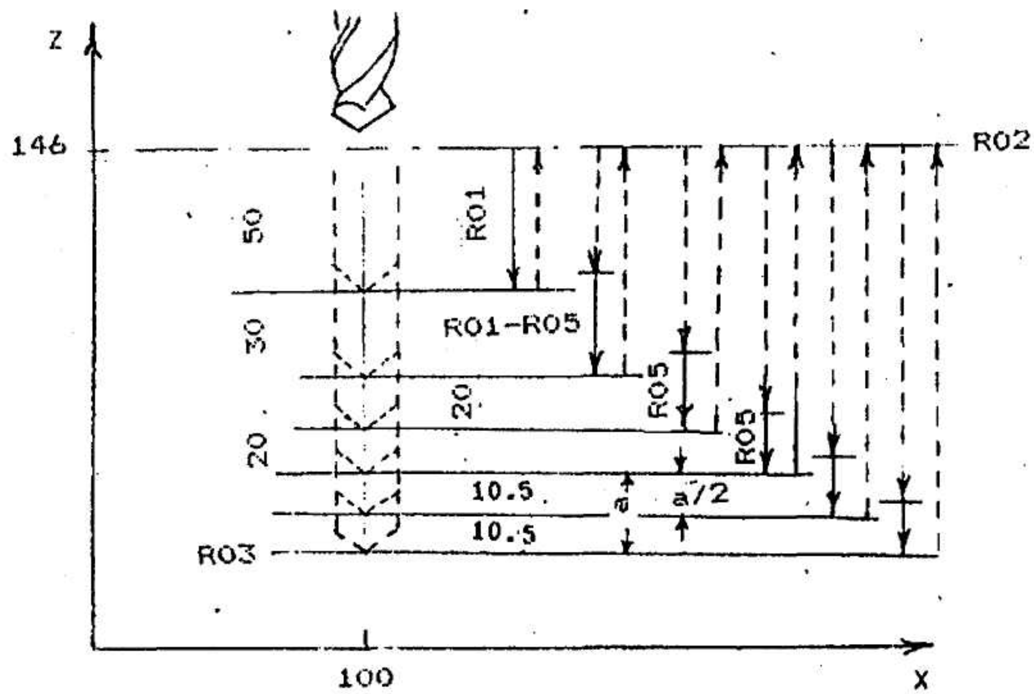
R05 Degression, (incremental depth advance modifier stored as unsigned dimension).

R11 Drilling axis.

First drilling depth 50 mm	R01 50
Reference plane & Retract plane 146 mm	R02 146
Final drilling depth 5mm	R03 5
Dwell at starting point 5 sec	R00 5
Dwell at final depth 1 sec	R04 1
Degression value 20mm	R05 20
Drilling axis (Z axis)	R11 3

R03 Final hole depth :
The respective drilling depth diminishes with each successive drill amount till the final hole depth R03 is reached. If the incremental depth advance modifier, exceeds the actual drill advance, succeeding drill advances will be held constant. At the end of the drilling cycle the drill is brought to point A. If the remaining depth is greater than R05 and less than 2 times R05,,it is divided into 2 drilling strokes.
a = remaining depth.

At the rapid traverse advance with respect to the new drilling depth, a safety distance of 1 mm is kept (to take care of the chips still remaining in the hole). With the inch system (G70) the safety distance must be changed accordingly.



CYCLE L 83 (Deep hole drilling)

N8301 G90 S48 M03 F460 LF

N8302 GOO DO1 Z500 LF

N8303 X100Y150LF

N8304 G83 RO1 50 R02 146 R03 5 ROO 5 R04 1 R05 20 R11 3 LF

N8305 X250 Y300 LF

N8306 G80 Z500 LF

CALLING CYCLE L84 (Tapping with spindle encoder)

The following parameters must be defined prior to calling canned cycle L84.

R02 Reference plane (Retract position).

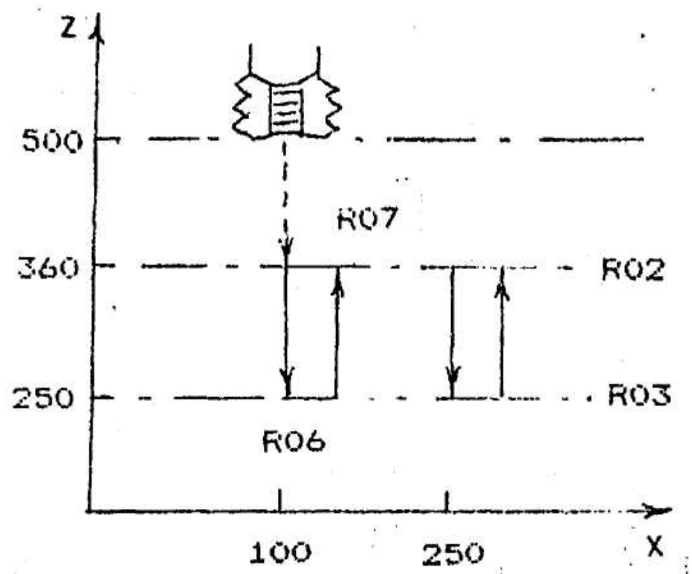
R03 Final depth.

R06 Spindle rotation reversal.

R07 Original spindle rotation direction.

R09 Thread lead dimension.

R11 Boring axis.



CYCLE L84 (Tapping with spindle encoder)

N8401 G90 S48 M03 F460 LF

N8402 GOO D01 Z500 LF

N8403 X100Y150LF

N8404 G84 R02 360 R03 250 R06 04 R07 03 R09 2 R11 3 LF

N8405 X250 Y300 LF

N8406 G80 Z500 LF

CALLING CYCLE L85 (Boring 1)

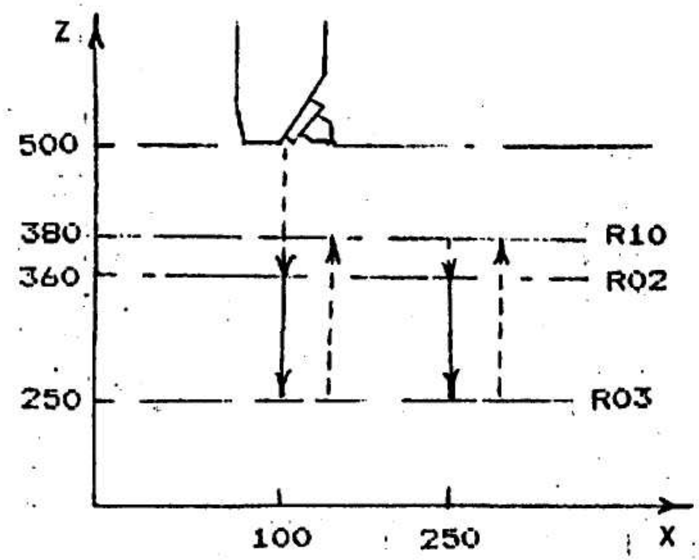
The following parameters must be defined prior to calling canned cycle L85.

R02 Reference plane

R03 Final depth

R10 Retract plane

R11 Boring axis



CYCLE L85 (Boring 1)

N8501 G90 S48 M03 F460 LF

N8502 GOO D01 Z500 LF

N8503 X100Y150LF

N8504 G85 R02 360 R03 250 R10 380 R11 3 LF

N8505 X250Y300LF

N8506 G80 Z500 LF

CALLING CYCLE L86 (BORING 2)

The following parameters must be defined prior to calling canned cycle L86.

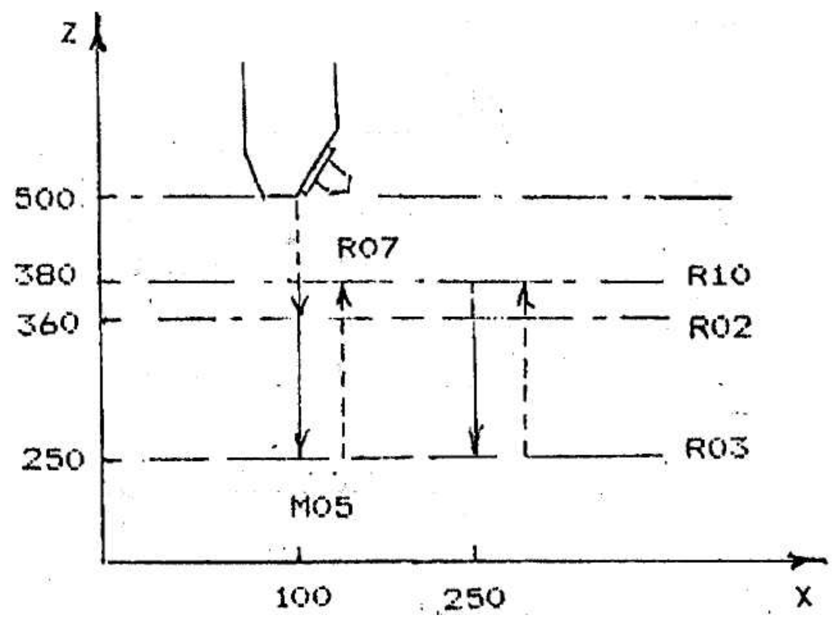
R02 Reference plance

R03 Final Depth

R07 Spindle on (After M05)

R10 Retract plane

R11 Boring axis



CYCLE L86 (Boring 2)

N8601 G90 S48 M03 F460 LF

N8602 GOO D01 Z500 LF

N8603 X100Y150 LF

N8604 G86R02 360 R03 250 R07 03 R10 380 R11 3 LF

N8605 X250 Y300 LF

N8606 G80 Z500 LF

CALLING CYCLE L87 (BORING 3)

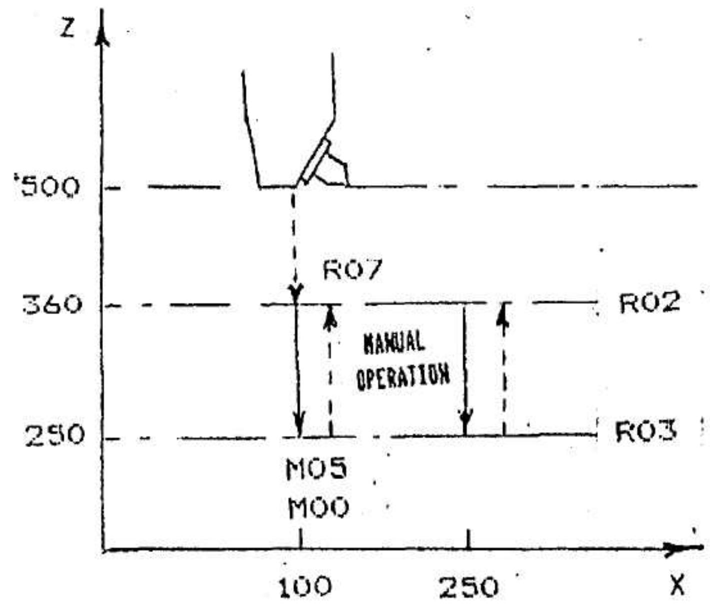
The following parameters must be defined prior to calling canned cycle L87.

R02 Reference plane (Retract position)

R03 Final depth

R07 Spindle on (After M05)

R11 Boring axis



CYCLE L87 (Boring 3)

N8701 G90 S48 M03 F460 LF

N8702 GOO D01 Z500 LF

N8703 X100Y150 LF

N8704 G87 R02 360 R03 250 R07 03 R11 3 LF

N8705 X250 Y300 LF

N8706 G80 Z500 LF

CALLING CYCLE L88 (BORING 4)

The following parameters must be defined prior to calling canned cycle L88.

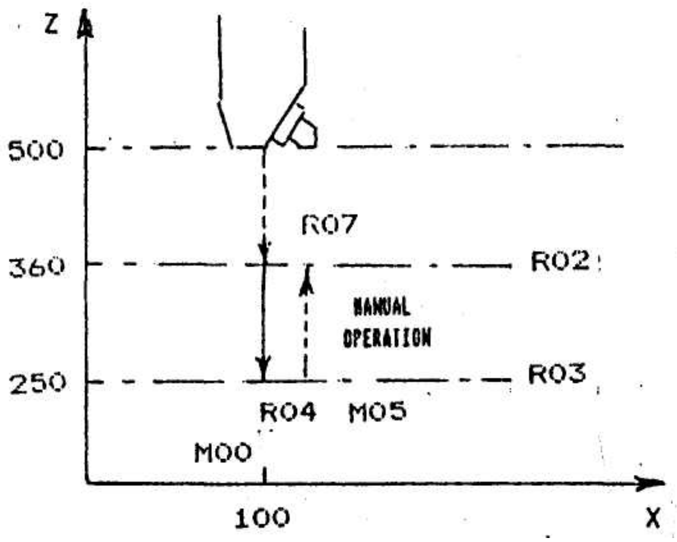
R02 Reference plane (Retract position)

R03 Final depth

R04 Dwell time

R07 Spindle on (After M05)

R11 Boring axis



CYCLE L88 (BORING 4)

```
N8801 G90 S48 M03 F460 LF

N8802 GOO D01 Z500 LF

N8803 X100 Y150 LF

N8804 G88 R02 360 R03 250 R04 1 R07 03 R11 3 LF

N8805 X250 Y300 LF

N8806 G80Z500 LF
```

CALLING CYCLE L89 (BORING 5)

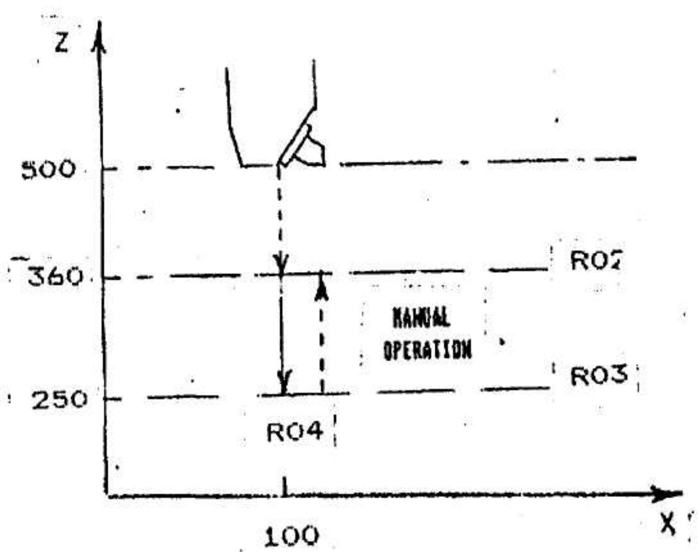
The following parameters must be defined prior to calling canned cycle L89.

R02 Reference plane (Retract position)

R03 Final depth

R04 Dwell time

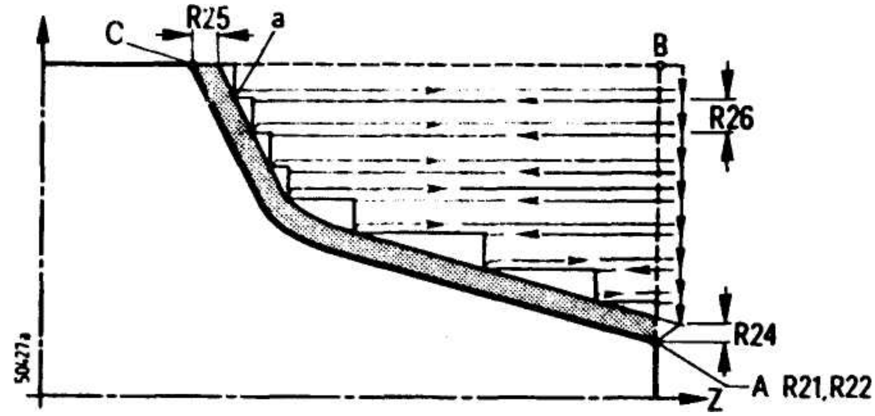
R11 Boring axis



CYCLE L89 (BORING 5)

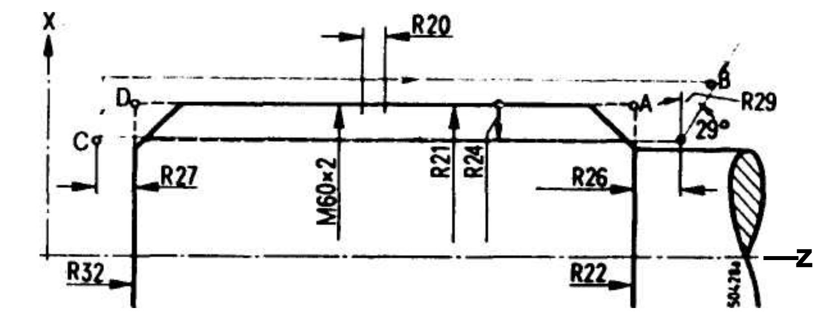
```
N8901 G90 S48 M03 F460 LF

N8902 GOO D01 Z500 LF

N8903 X100 Y150 LF

N8904 G89 R02 360 R03 250 R04 1 R11 3 LF

N8905 X250 Y300 LF

N8906 G80 Z500 LF
```

CONNED CYCLE (L85)



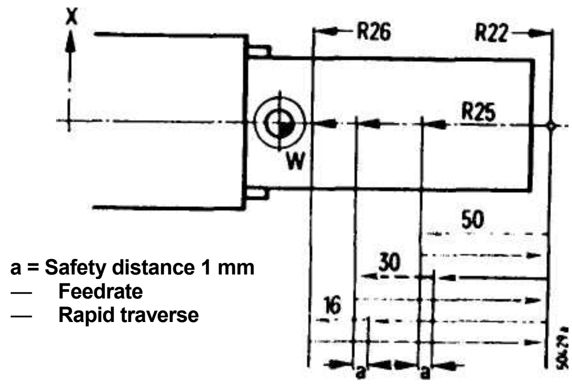
Subroutine number of the contour description	R20
Start position of contour in X	R21
Start position of contour in Z	R22
Finishing stock in X	R24
Finishing stock in Z	R25
Depth of roughing cuts in X and Z	R26
Tool nose radius compensation	R27
Definition of machining operation for roughing and finishing	R29

THREAD CUTTING CYCLE (L97)



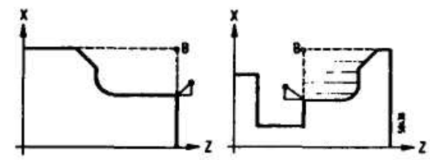
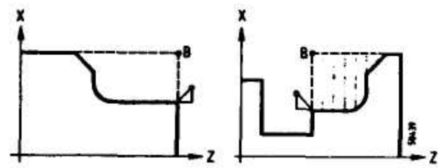
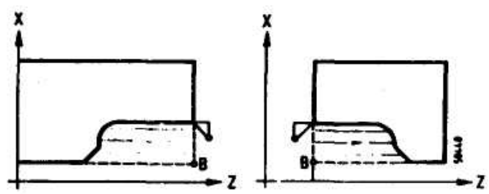
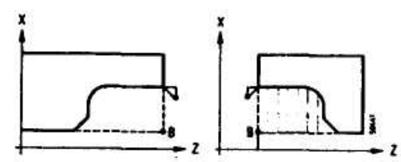
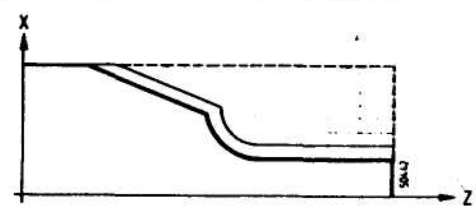
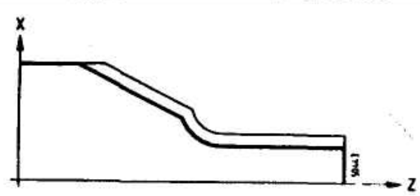
Start position of thread in X	R21
Start position of thread in Z	R22
Number of non-cutting posses	R23
Depth of thread	R24
Finishing stock	R25
Acceleration distance	R26
Overrun distance	R27
Number of roughing cuts	R28
Angle of infeed	R29
End position of thread in X	R31
End position of thread in Z	R32

DEEP HOLE BORING CYCLE (L98)

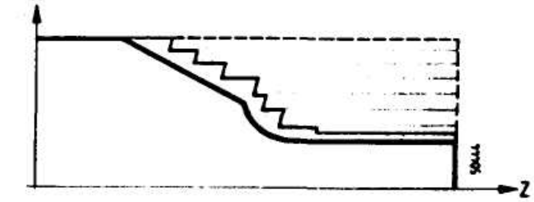
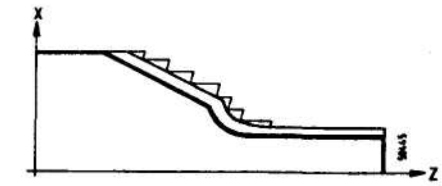
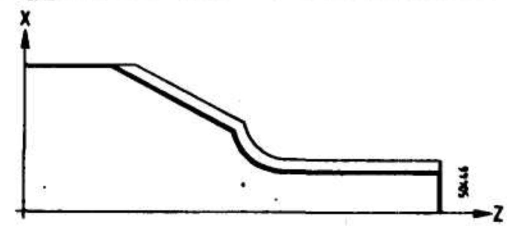


Start position in Z	R22
Amount of degression	R24
First drilling depth	R25
Final drilling depth	R26
Dwell at start position (chip removal)	R27
Dwell at hole bottom (chip breaking)	R28

The parameter R29, states the cutting type
Roughing, finishing, external or internal machining
Type of cut segmentation, longitudinal or face.

Parameter	Machining	Cutting type	Example
R29 11	od longitudinal(Z)	Roughing	 External machining in longitudinal direction (Z)
R29 12	od face (X)		 External machining face roughing (X)
R29 13	Id longitudinal (Z)		 Internal machining in longitudinal direction (Z)
R29 14	Id face (X)		 Internal machining face roughing (X)
R29 21 R29 22 R29 23 R29 24	External External Internal Internal	R29 22, R29 24, roughing Parallel to contour upto finishing stock	 R29 22, roughing parallel to contour upto finishing stock
		R29 21, R29 23 finishing to finishing stock	 R29 21, finishing parallel to contour upto finishing stock

Description and pictorial examples :

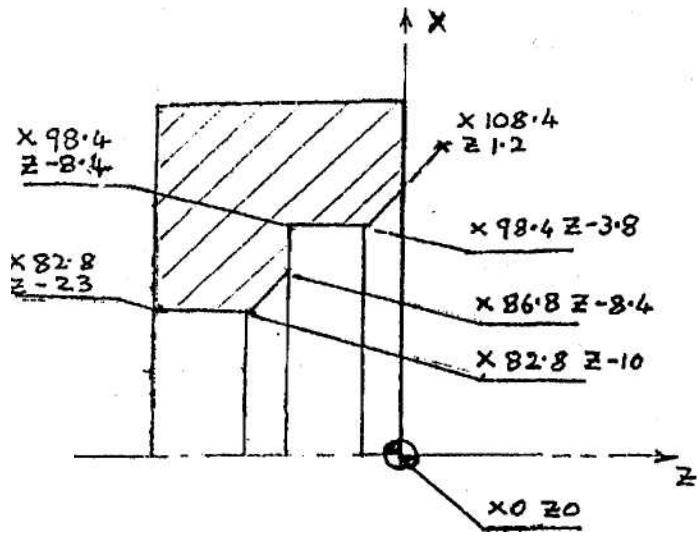
Parameter	Machining	Cutting type	Example
R29 31 R29 32 R29 33 R29 34	od longitudinal (Z) od face (X) id longitudinal (Z) id face (X)	Roughing paraxial then a cut parallel to contour up to finishing stock	 R29 31; R29 41, 1st Paraxial roughing
			 R29 31; R29 41, 2nd Paraxial roughing
R29 41 R29 42 R29 43 R29 44	od longitudinal (Z) od face (X) id longitudional (Z) id face (X)	One cut parallel to contour up to final contour	 R29 41, 3rd cut parallel to contour

EXERCISE FORTURNING CYCLES PROCESS SEQUENCES:

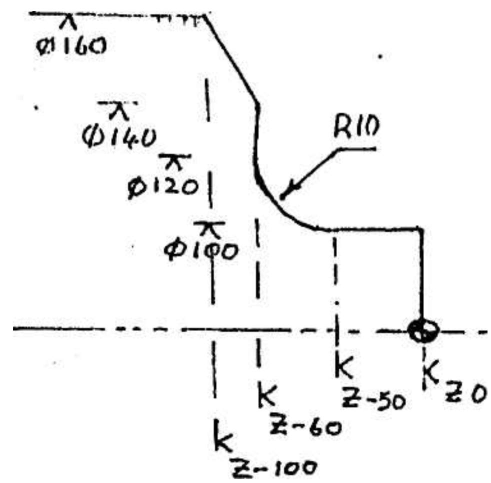
1. Zero off set G54, Abs G90, Metric data G71 Feed mm/rev G95. TNRC cancel G40.
2. Spindle (rpm) range selection M25.
3. RPM control G92 at S500.
4. CSS G96,100 meters / min (S100) & spindle rotation M04.
5. Position tool turret to tool change point X250 Z200.
6. Tool change T0101.
7. Position tool to start of facing cut X150 Z0. Coolant on M08.
8. Facing from dia 150 to dia 80 (X150 to X80) at Feed 0.2.
9. Position tool 2mm away from face (Z2 X80), coolant off M09.
10. Position tool to tool change point (X250 Z200) spindle stop M05.
11. Tool change T1212.
12. CSS G96, S100 spindle rotation M04.
13. Position tool to dia 70 & 2mm away from face (X70 Z2) coolant on M08.
14. Bore dia 70 to depth 23mm (X70 Z-23)
15. Position tool out clear of job (X68 Z2).
16. Position tool to next cut (X73.5)
17. Bore dia 73.5 to depth 23mm (X73.5 Z-23).
18. Position tool out clear of job (X70 Z2).
19. Position tool to next cut (X77 Z2).
20. Bore dia 77 to depth 23mm (X77 Z-23)
21. Position tool out clear of job (X76 Z2)
22. Position tool to X109 Z1.5
23. Define R-Parameters of L95 Stock removing cycle.
24. Call L95 cycle with feed value.
25. Position tool out of bore clear of job (X80 Z5). Coolant off M09.
26. Position tool to tool change point X250 Z200 spindle stop M05.
27. Speed range change (rpm) M25.
28. Tool change T0404 & coolant on M08.
29. Spindle rotation S400 & spindle rotation M03.
30. Position tool to X80.6 Z5.
31. Position tool to Z-6 & Feed 0.2.
32. Define R-Parameters of L97 Threading cycle.
33. Position tool clear of job X78 & coolant stop M09.
34. Position tool out clear of job Z5 and stop spindle M05.
35. Position tool to tool change point X250 Z200.
36. Program end M30.

EXERCISE FORTURNING CYCLES

```
% 2649 *
(HTM COMPONENT 765-50-2659 STC25) *
N1   G54G90 G71 G95 G40*
N2   M25 *
N3   G92 S500*
N4   G96S100M04*
N5   GOO X250 Z 200 *
N6   T0101 *
N7   G00X150 ZM08*
N8   G01 X80 F0.2 *
N9   Z2 F5 M09 *
N10  GOO X250 Z200 M05 *
N11  T1212*
N12  G96 S100 M04 *
N13  GOO X70 Z2 M08 *
N14  G01 Z-23F0.2*
N15  X68 Z2 F5 *
N16  X73.5*
N17  Z-23 F0.2 *
N18  X70 Z2 F5 *
N19  X77*
N20  Z-23 FO. 2 *
N21  X76 Z2 F5 *
N22  X109 Z1.5 *                          R24 0      R25  0
N23  R20 26    R21  108    R22

      R26 1.5    R27 41      R29 33*

N24  L95 FO . 2 *
N25  G01 X80 Z5 F5 M09 *
N26  GOO X250 Z200 M05 *
N27  M25 *
N28  TO 404 M08 *
N29  G95 S400 M03 *
N30  GOO X80.6 Z5 *
N31  G01 Z-6 F0.2 *

. N32  R20 2     R21  82.8   R22 -6     R23 4

      R24 1.3    R25 0.05   R26 2      R27 2

      R28 20     R29 30     R31  82.8  R32 23     L97 *

N33   G01 X78 F3 M09 *
```



```
N34     Z5 M05 *
N35     GOO X250 Z200 *
N36     M30 *

L2600 *
(SUBPROGRAM FOR 765-50-2649 - PART PROG NO 2649)
N1 G90 G01 X108.4Z1.2F0.2 *
N2 X98.4 Z-3. 8 *
N3 Z-8. 4 *
N4 X86.8 *
N5 X82.5 Z-10 *
N6 Z-23 *
N7 X0 *
N8 M17 *

L10
N1      G1 X100 Z-50 *
N2      G2X120 Z-60 110 KO *
N3      GI X140 *
N4      X160 Z-100 *
N5      M17 *
```




```
%10
N1      G90 G71 G54 G40 G95
N2      M25 *
N3      G92 S800 *
N4      G96 S180 M04 T0303 *
N5      GO X160 Z10 M08 *
N6      R20 10            R21   100     R22   5        R24   0.3      R25 0.3
        Sub Prog. No.     S. P    X     S.R   Z        FINISH ALLOW X & Z*

        R26 5             R 27   42     R29 11  *
        Rough cut depth   TNRC          Type of cutting

N7      L95 F0.5 *
N8      R24 1 R25   1 R27 42    R29   21  L95*

N9      R24 0. 5 R25 0.5 R27 42 R29 21 L96 *
N10     R24 0.3 R25 0.3 R27 42 R29 21  L95 *
N11     GO X300 Z200 M05 *
N12     M27 *
N13     S200 T0101  M04 *
N14     GO X100Z10 M08 *
N15     R24 0 R25 0 R27 42 R29 21  L95 F0.15 *
N16     GO X300 Z200 MO9 *
N17     M30 *
```

IF R29 31 IS GIVEN AT BLOCK N6, BLOCKS N8TO N10 ARE NOT REQUIRED
IF R29 41 IS GIVEN AT BLOCK N6, BLOCKS N8 TO N15 ARE NOT REQUIRED

FANUC SYSTEM
STOCK REMOVAL CYCLE G73 & FINISHING CYCLE G72

```
0 10; (PROGRAM NO.)
N1      G90 G21 (ORG71) G40 G95;
N2      G00 G53 XO. Z-500. (POSITION TOOL IN M/C COORDINATE SYSTEM,
                          XO = HOME POSITION, Z= - 500MM FROM HOME POSITION;
N3      M01; (CONTROL CONDITION, THIS IS NEEDED IN SOME CONTROLS) N4
T0303;
N5      G92 X300. Z200. S800; (SETTING WORK ZERO W.R.T TOOL NOSE)
N6      G96S140 M04;
N7      GOO X160 Z10. M08;
N8      G73     P9            Q14        U0.3        W0.3        D5         F0.5;
        Cycle   Start         End        Finish cut depth        Depth      Feed
        call    Blocks of                X axis      Z axis      of cut
                Finish profile                                   Roughing
N9      GOO X100.; (FINISH PROFILE 1ST BLOCK) N10     G01 Z-50. G42 F0.15; (G42
& F0.15 WILL NOT BE ACTIVE DURING ROUGHING
        CYCLE RUN *N8 BLOCK)
N11     G02 X120.Z-60. 110. JO;
N12     G01 X140.;
N13     X160.Z-100.; (FINISH PROFILE LAST BLOCK)
N14     GOO X 300. Z200. M09; N15     TO M08; (TOOL
OFFSETS CANCELL) N16     M01;
N17     T0101; (FINISH TOOL)
N18     G92 X295.Z195.S800; (SETTING WORK ZERO W.R.T TOOL NOSE)
N19     G96S200M04; N20     G00X160. Z10. M08; N21     G72 P9 Q14;
(FINISHING CYCLE G72 CALL)
        (IF A FEED VALUE GIVEN IN THIS BLOCK THEN FEED GIVEN IN BLOCK
        N10 WILL NOT BE ACTIVE)
N22     G00X300. Z200. M09; N23
TO M05; N24     M30;
```

THREAD CUTTING - L97

PITCH	R20 = 2
THREAD DEPTH	R24=1.3
START POINT X	R21 =60
START POINT Z	R 22 = 150
END POINT X	R31 = 60
END POINT Z	R32 = 95
NO OF IDLE PASSES	R23 = 2
FINISHING CUT DEPTH	R25 = 0
NO OF ROUGHING CUTS	R28 = 7
INFEED ANGLE	R29 = 29 DEG

N140 R20 2 R21 60 R22 150 R23 2R24 -1.3
Pitch S.R X S.P Z Idle passes Thread depth (min

R25 0 R26 10 R27 8 R28 7 R29 29*
Finish depth Approach path Run out No of rough cuts Infeed angle

N150 R31 60 R32 95 L97 *
E.P X E.P Z

N160 GO X.... Z.... M09 *(MOVE TO TOOL CHANGE POSITION)

N170.............

SECTION 7

CONTOUR DEFINITION

CONTOUR DEFINITION

1. BLUEPRINT PROGRAMMING :

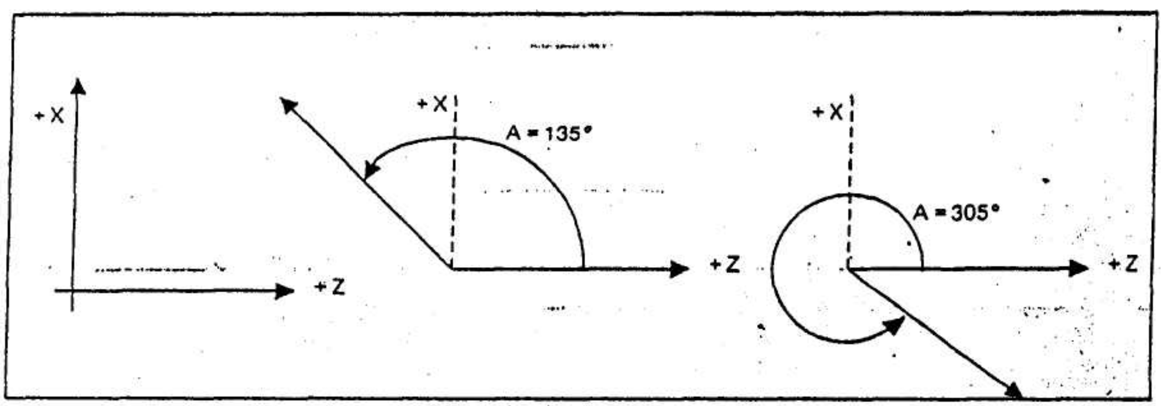
Multi-point definitions for direct programming in accordance with the workpiece drawing are provided for blueprint programming. The points of intersection of the straight lines are specified as coordinate values or by means of angles.

The various straight lines can be joined together directly in the form of a corner, rounded via radii or chamfered. Chamfer and transition radii are specified only by their size. The geometrical calculation is performed by the control. The end position coordinates can be programmed using either absolute or incremental position data.

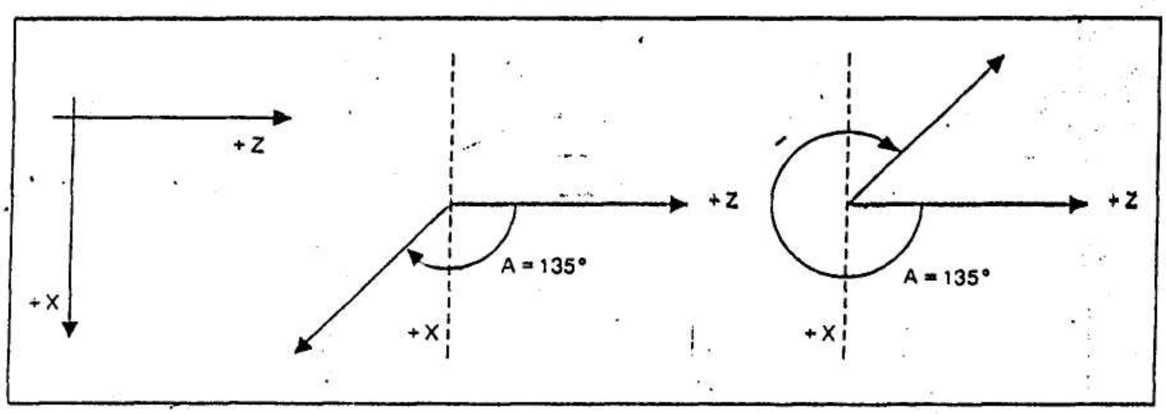
Angle (A):
Input resolution 0.00001 corresponds to $10^{5°}$.

In the clockwise coordinate system the angle (max. 359.99999°) is always measured from the horizontal axis direction to the vertical axis direction.

Turning Machine

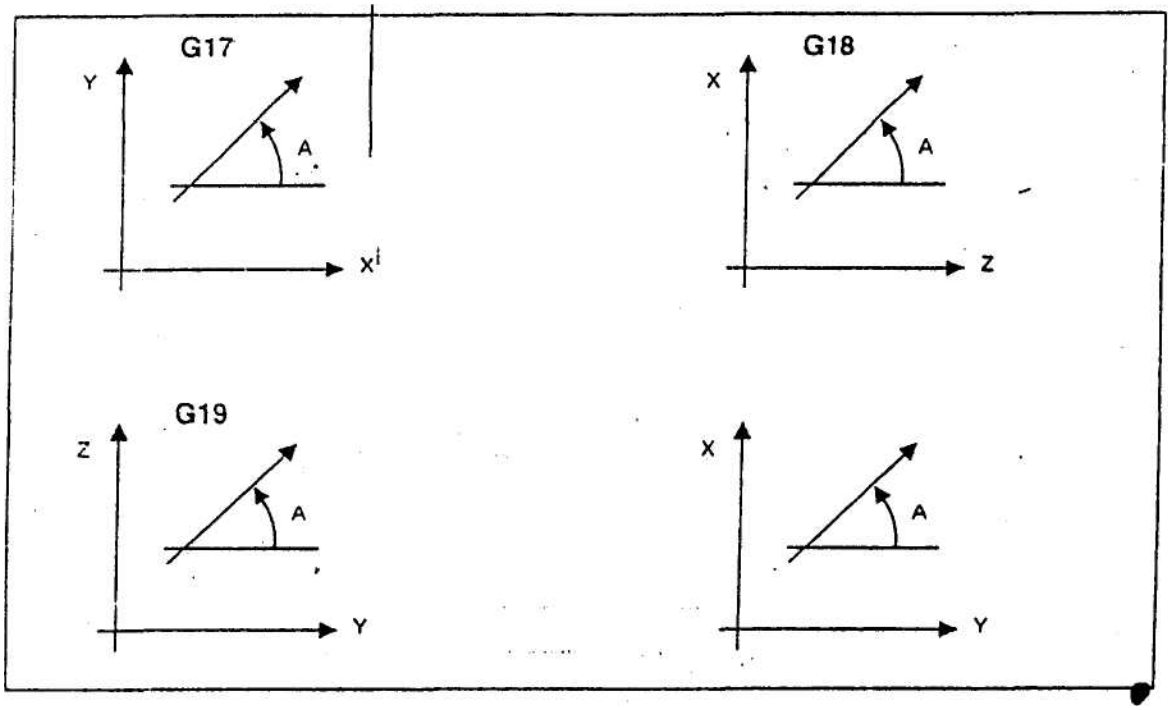


Clockwise system and operating area after turning centre



MILLING MACHINE

Plane selection: The required plane is selected with G17, G18 or G19.



Plane selection:

If the plane is freely selected (G16) the plane is specified by means of the programmed axe! The first axis programmed is the reference axis. The angle in the clockwise coordinate system is always referred to the reference axis.

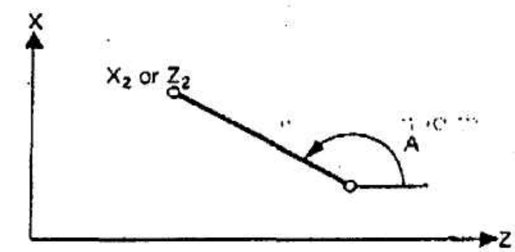
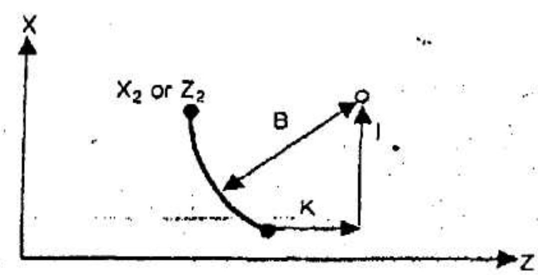
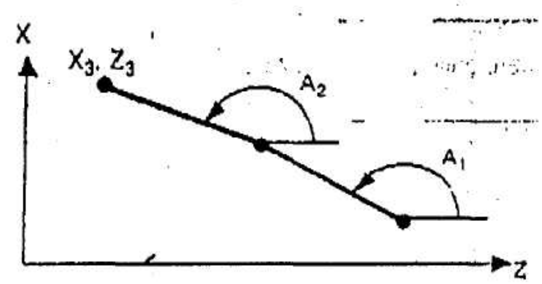
Blueprint programming is only permissible in the selected plan.
3D machining is not possible.

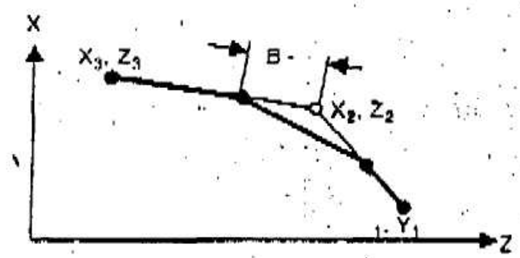
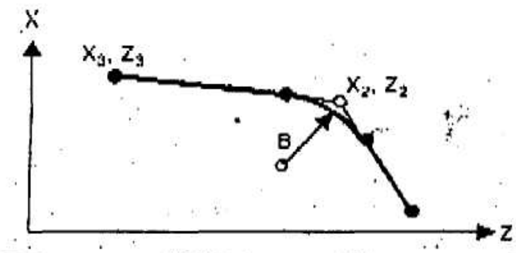
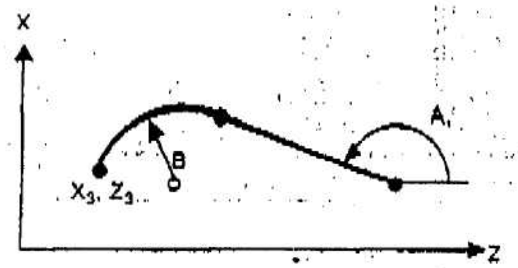
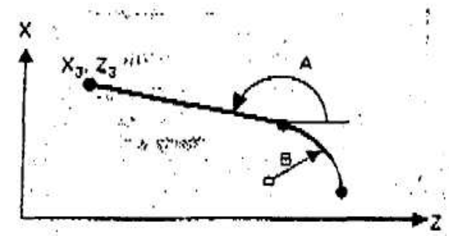
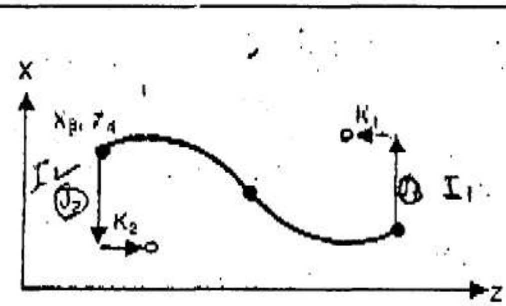
---

2. CONTOUR DEFINITION PROGRAMMING :

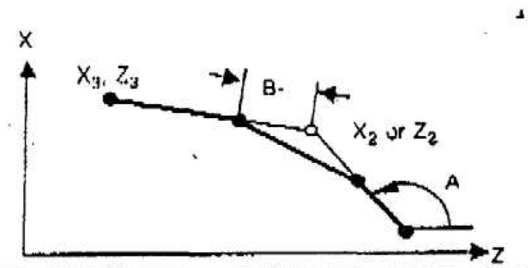
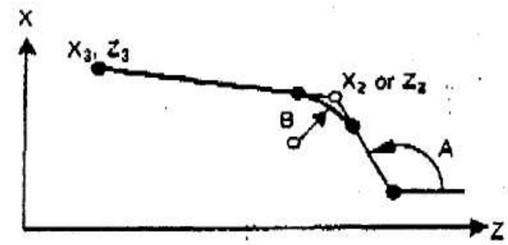
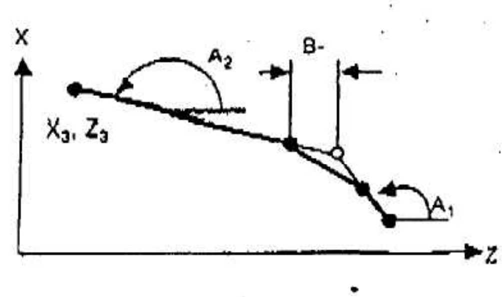
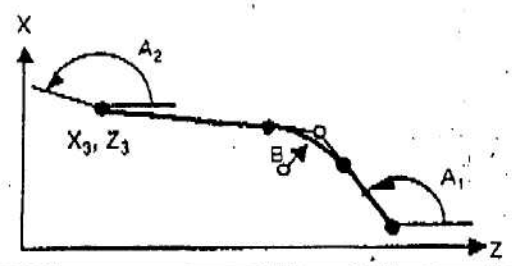
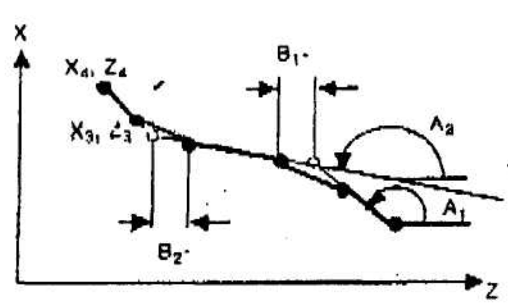
The elements described are valid for a turning machine with an operating area after the turning centre and for a milling machine in the selected plane Z-X (G18).

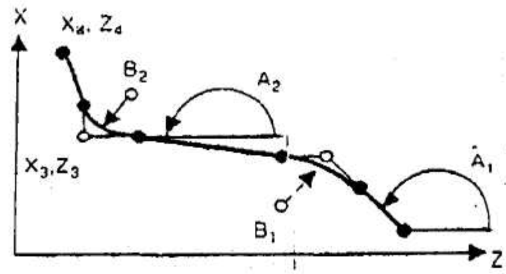
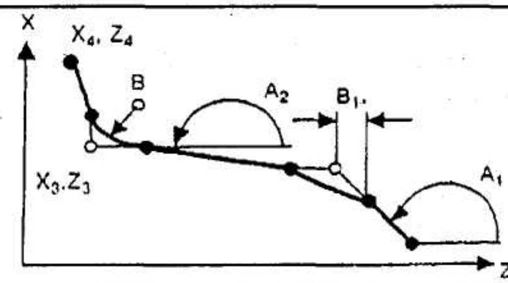
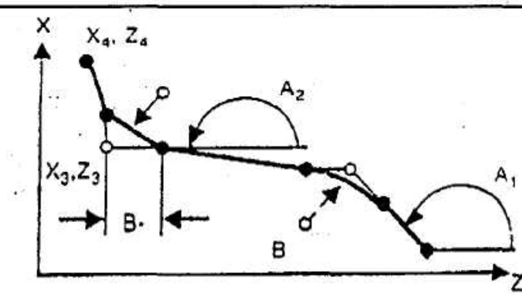
Examples 1 to 8 represent the basic elements of contour definition programming. These contour elements can be combined in a number of ways. The addresses for the angle (in this case A) and the radius (in this case B) are freely selectable in the control. The addresses must not be allocated-more than once.

Function	Programming	Example
(1) 2-point definition	N..... A.... X2... (or Z2) LF The second end coordinate is circulated by the control.	
(2) Circular arc	N.... G02, (or G03) I...K...B...X2... (or Z2) LF The circular arc is limited to one quardant. The seond end position coordinate is calculated by the control. In the contour definitions parameters I and K must both be programmed, even if one of the values is zero.	
(3) 3-point defintion	N... A1.... A2.... X3... Z3.... LF The control calculates the coordinates of the vertex and generates 2 blocks. Angle A2 is referred to the second straight line.	

Function	Programming	Example
(4) chamfer	N X2... Z2... B- LF N...X3...Z3...LF (1) B- means insert a chamfer B means insert a radius (The minus character is not a sign here; instead it is a special identifier of B = chamfer)	
(5) Radius	N.... X2..... Z2.... B.... N.... X3..... Z3... LF The inserted radius must not be larger than the smaller of the two paths	
(6) Straight line circular arc (tangent)	N G02 (or G03) A... B... X3...Z3... LF The circular arc must not exceed 180. The sequence A (angle) followed by B (radius) must be used.	
(7) Circular arc-straight line (tangent)	N... G02 (or G03) B...A...X3...Z3... LF The circular arc must not exceed 180^0. The sequence B, A must be used. A radius must not be inserted in X3, Z3.	
(8) Circular arc circular arc tlangent	N...G02 (or G03) L1...K1...I2...K2..X3 Z3... LF circle The preparatory function is programmed for the first circular arc. The second preparatory function is always the opposite of the first one and is not programmed. The interpolation parameters of the second circle are referred to the end position of this circle. Both interpolation parameters must be programmed, even if one of the values is zero.	

(1) Second block can also be a contour definition

Function	Programming	Example
(1) + (4) 2-point definition + chamfer	N.... A... X2... (or Z2....) B - LF N.... X3...X3... LF (1)	
(1) + (5) 2-point definition + radius	N..... A.... X2 (or Z2...) B... LF N.... X3... Z3... LF The inserted radius must not be larger than the smaller of the two paths.	
(3) + (4) 3-point definition + chamfer	N... A1.... A2... X3... Z3... B- Lf	
(3) + (5) 3-point definition + radius	N... A1...A2...X3... Z3... B... LF	
(3) + (4) + (4) 3-point definition + chamfer + chamfer	N...A1...A2...X3...Z3...B1-...B2-.... LF N...X4...Z4... LF Addition of a second chamfer at end position (X3, Z3)	

Function	Programming	Example
(3) + (5) + (5) 3-point definition + radius + radius	N....A1...A2...X3...Z3...B1...B2...LF N...X4...Z4...LF Addition of a second radius at end position (X3, Z3).	
(3) + (4) + (5) 3-point definition + chamfer + radius	N....A1...A2...X3...Z3...B1 - B...LF N...X4...Z4...LF Addition of a radius at end position X3, Z3. The next block is always taken into consideration automatically.	
(3) + (5) + (4) 3-point definition + radius + chamfer	N....A1...A2...X3...Z3....B - B...LF N...X4...Z4...LF Addition of a chamber B - at the end position	

BO must be programmed for corners where no chamfer or radius is to be inserted if a furthe radius or chamfer follows in the contour definition.

When this is programmed, the control generates a block with a distance of 0. This must be noted if TNRC/CRC is active. B-0 is interpreted as BO. A radius or chamfer can only be inserted between two linear blocks.

The sequence of addresses A, X, Z, B, F etc. is freely selectable; angles and radii must how ever be entered in the sequence described above (first angle before second angle, first ra dius before second radius in machining direction).

3. OPERATION OF FUNCTIONS G09, F, S,T, H, M in contour definition :

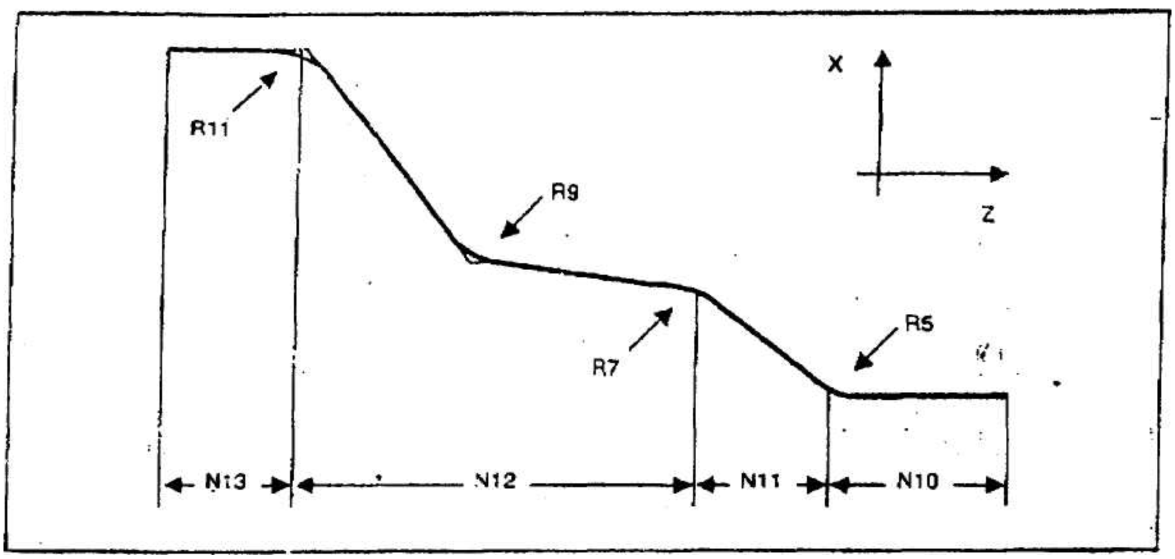
If G09 is programmed in a contour definition block, it is not active until the end of the block, i.e. when the end position is reached. G09 is automatically generated by the control at irregular points (corners, edges) in the contour definition.

If F, S, T, H or M are programmed in a contour definition block, they are active at the start of the block; MOO, M01, M02, M17 and M30 are active at the end of the block.

4. CHAINGING OF BLOCKS :

It is possible to chain blocks with or without angle inputs and with inserted radii or chamfers in any sequence.

Example

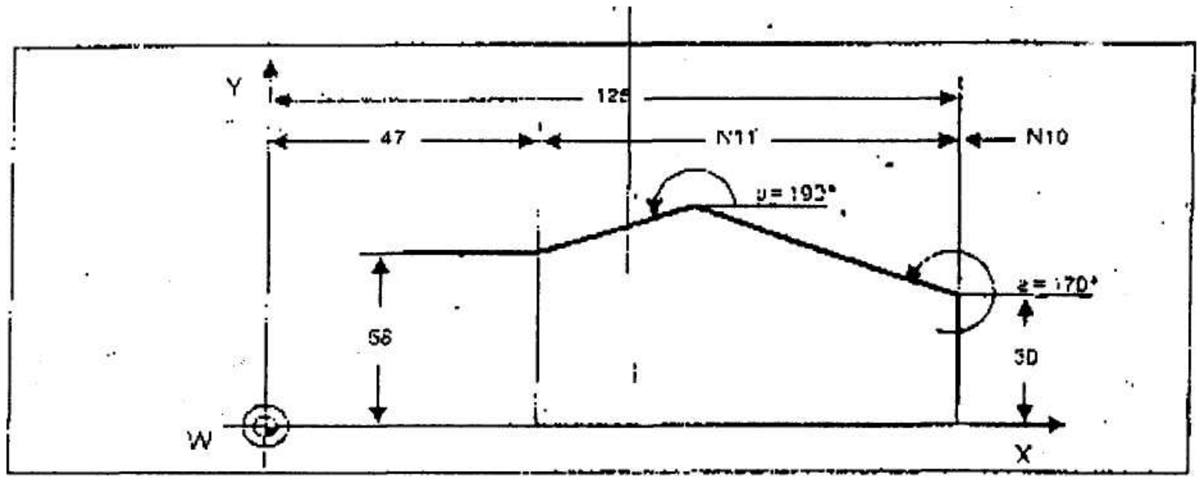


Chaining of blocks

N10 Z….B5LF
N11 A....X....B7.LF
N12 A...A...X...Z...89 B11. LF
N13 Z... LF

5. EXAMPLE MILLING MACHINE :

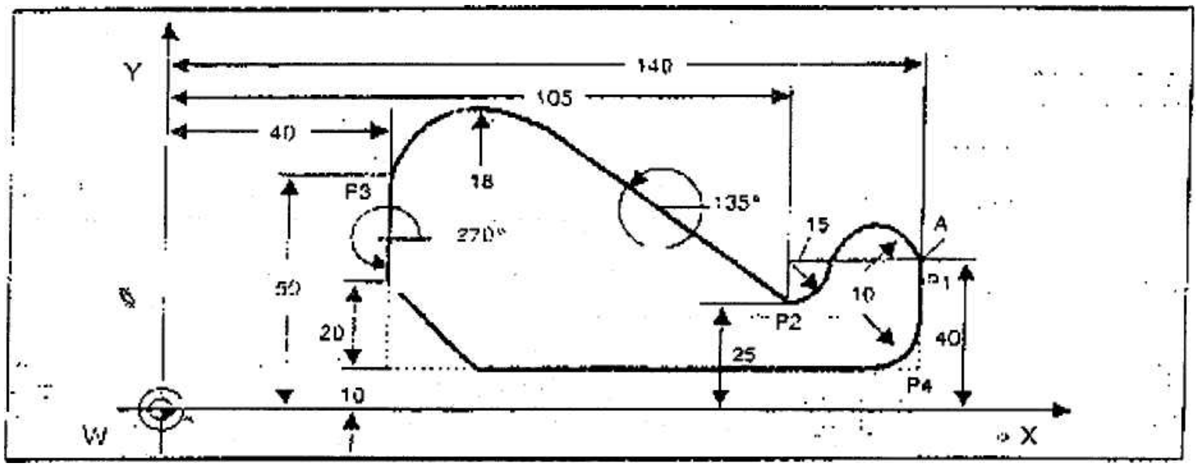
Angle a refers to the starting point; angle b refers to the missing vertex. The end point can be programmed using absolute point data G90 or incremental position data G91. Both end point coordinates must be specified. The control determines the vertex from the known starting point, the two angles and the end point.



N10 GOO G90 X125. Y50. LF
N11 G01 A170 A190. X47. Y58. F....LF

Examples: Contour definition programming for milling machine

In the example described below the following contour definitions are used: Circular arc circular arc, straight line - circular arc, 3-point definition + chamfer + radius.



A = Starting point

L 168
N1 G90 G03 1-10. JO. 10. J15. X105. Y25. LF (P2)
N2 G03 A135. U18. X40. Y50. LF (P3)
N3 G01 A270. AO. X140. Y10. U-20. U10. LF (P4)
N4 Y40. LF(P1)
N5 M17 LF

SECTION 8

NC TOOLING & QUALIFIED TOOLS

NCTOOLING & QUALIFIEDTOOLS

In any machining operation the tooling plays an important part in determining the results in terms of metal removal, surface finish and economy. The cost of any CNC machine is so high and hence its effective utilization should be of paramount importance. In general the selection of any tooling depends on -

1. Power and condition of machine.

2. Surface finish requirement.

3. Spindle and work piece rigidity.

4. Cutting speed and feed.

5. Economic tool life.

The urge for use of cutting tool materials like cemented carbide, coated carbide, ceramics, cubic boron nitride (C B N), Poly crystalline diamond, etc., followed by HSS increased due to the varying machine ability of new and better materials in work pieces and also to reduce the production costs by increase of cutting speeds & feeds.

While aiming at increased production rates the economic tool life is required to be seen as the tooling costs are high, consideration has to be paid to the cutting depth and feed should be chosen giving due consideration to stability, surface finish, chip formation, etc., After this the cutting speed can be selected in such a way that gives economic tool life.

The objective of using any tooling should be to increase production rate at minimum cost.

In conventional machines, the cutting tool cuts metal for about 25% of the total machining time where as CNC machine tools are expected to cut metal for 70 to 80% of the time.

NC machines have been designed to achieve high power, high speeds better rigidity, closer accuracies and multi operational capabilities. In order to utilize the high capability and high cost NC machines, proper selection of tooling is more important other wise improper selection of tooling will affect the utilization of machine resulting heavy financial loss.

The NC toolings should therefore have -

1. More rigidity to withstand high metal removal rate for full utilization of spindle power.

2. Presetting and resetting in the shortest possible time.

3. Accurate enough for repetitive accuracy.

NC toolings are broadly classified into:

1. Cutting tools

2. Tool holding devices

3. Work-holding devices

1. Cutting Tools:
Cutting tools may be subdivided into

(i) Stationary tools (ii) Rotary tools

(i) Stationary tools: Both inside and outside tools. These are the tools used in centre lathes turret lathes, capstan lathes, turning centres.

Besides HSS tools, tools with inserts are used.

Presetting of tools:

In order to reduce idle time of machines, the adjustment of tools on machine should be mini mum. The length, width, tip radius, etc are to be measured with a device called tool presetter and the reading may be entered in the tool offset page of CNC control.

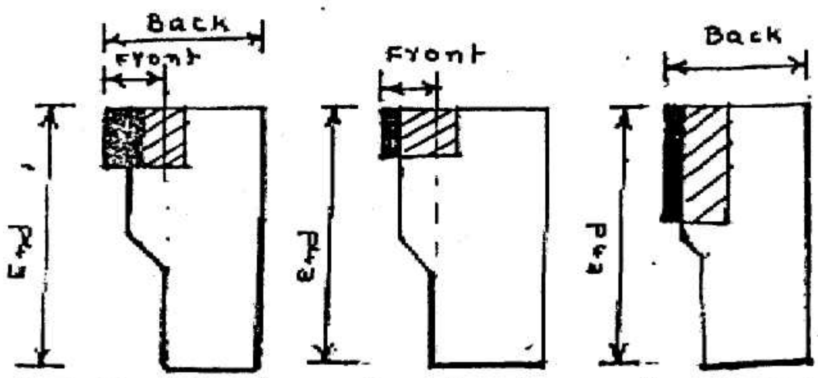
The presetting of tools can be planned and carried out in advance so that tools are available to ensure continuity of production and minimize down time due to tool set up on job changes Special presetting devices are available for the resetting of tools.

The various tool presetters available are simple mechanical type, optical type having micron reading and digital displays.

Presenting is done with the tool held in the tool holder so that the assembly i.e. the tool holder and cutting tool can be straight away fitted on the machine.

Qualified tools:

Some times presenting does not ensure required accuracy due to variation in job material and rigidity of set up. In such cases the qualified tools are of immense help and economical. Qualified tools are of tools having known dimensions with tolerances written on the tool shank. The position of the cutting edge is guaranteed within close limits of accuracy from a specified datum on the tool holder in these tools. Hard metal inserts are ideally suited for qualified tooling.



Back, Front & end Qualified tool | Front & end qualified tool | Back & End Qualified tool

(ii) Rotary tools:

These are the tools used mostly in milling, boring, machining centres, etc. They are drills, taps, reamers, end mills, milling cutters etc. In order to make these tools suitable for high metal removal chip breakers, coolant fed drills, etc are developed. Straight fluted taps and helical fluted taps are recommended.

Inserts and cartridges of various shapes are developed.

Presetting:

Rotary tools require presetting for length. This can be done by height master, height guage and universal presetting devices. The presetting on length need not be very accurate since tool length compensations are normally available on NC machines.

2. Tool holding devices:

Tool holding devices should be of rigid and flexible type. Whether rotary or stationary tool holders must be quick changing type and should be able to preset. Since in the modern CNC machines, there is a provision for automatic tool changing, the quick changing type tool holders can reduce the tool changing time to about 3 to 5 seconds.

On turning centre tool turrets are provided with 8-12 tools.

On machining centres, automatic tool changer consists of a tool magazine for storing the tools and a tool change unit for transferring the tool from tool magazine to spindle. Tool magazines are provided with as many as 60 tools. The tool change cycle consists of tool selection and tool transfer.

The various tool holding devices are -

(i) Turning tool holders

(ii) Milling tool holders.

(iii) Collect chucks
(iv) Drill chuck adaptor
(v) Arbors
(vi) Adaptors
(vii) Boring bars
(viii) Tapping attachment
(ix) Special tool holding device

T max tooling systems, block tool systems for various purposes are available.

Indexable inserts of various shapes are available which make tool-tip replacement easy am quick and also more cutting edges are available.

3. Work holding Devices:

The work holding device on CNC machines should facilitate to hold the component in such a way that maximum sides are exposed for machining without changing or repositioning of clamps. To reduce the clamping / unclamping time, hydraulic and pneumatic actuation is widely used in work holding devices.

The collets and jaw chucks in NC machines should be rigid and accurate enough to ensure repeated accuracy and concentricity. To reduce idle time they should be power operated with sufficient clamping force. They are generally hydraulic operated and ensures quick action.

The latest concept is modular tooling where in the standard elements like base plates, angle plates, V blocks, holding devices, clamps, screws, etc are used to assemble into any desired fixture to suit the job requirement in a short time.

Automatic tool changers (ATC):

The CNC machines are designed to perform a number of operations in a single setting of the work piece. To reduce the down time in change over from once operation to the next, the CNC machines are equipped with automatic tool change facility. The tool is automatically selected and changed based on the tool control function (T-word) in the part programme.

Multi pallet Machines:

To further reduce the non productive time, the CNC machines are provided with automatic pallet change systems. Twin pallet CNC machines are very common but machines with up to 5 pallets are also being used. The multi pallet system enables the operator to load the work piece on one pallet while machining of work piece on second pallet is going on. This helps is rapid change of work piece, there by reducing the idle time of the machine.

The ISO code keys for certain turning and milling tools & inserts are given in separate sheet

ISO carbide grades are given below for different materials. These grades are to be careful selected keeping in view the type of material to be machined and the inserts wear resistance and toughness properties.

1 P01, 10, 20, 30, 40 & 50 — Suitable for steel, cast steel, stainless steel & long Chipping malleable iron

←——————————→
Wear resistance Toughness .

2.- M 10, 20, 30 & 40 — Suitable for steel, cast steel, manganese Steel, Alloy cast iron, Austenitic stainless steel castings, malleable iron & free cutting steel.

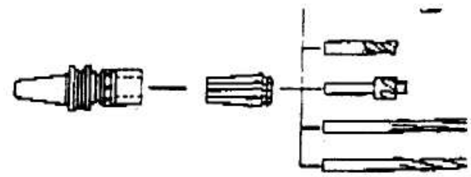
←------------------------→
Wear resistance toughness

3.-K 01, 10, 20 & 30 — Suitable for cast iron, chilled cast iron short chipping malleable iron, hardened steel, Non ferrous metals, plastics & wood

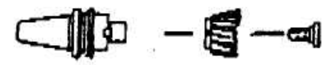
←----------------------→

CNC
TOOLING SYSTEM

COLLETS
COLLET CHUCK

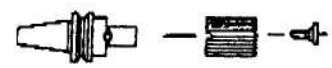


FACE MILL ARBOR - FMB

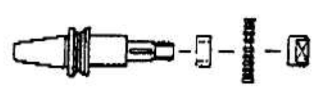


SHOULDER CUTTER ARBOR-FMC

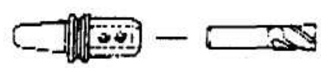
SHELL END MILL ARBOR-SMS



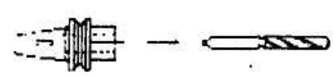
SIDE AND FACE MILL ARBOR-SCA
(STUB ARBOR)



SIDE LOCK HOLDER-SL/SLA FOR
"WELDON"TYPE END MILLS



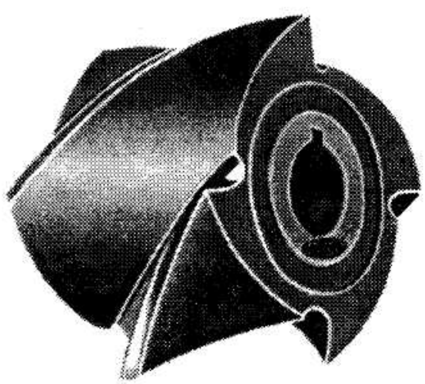
MORSE TAPER ADAPTOR - MTA

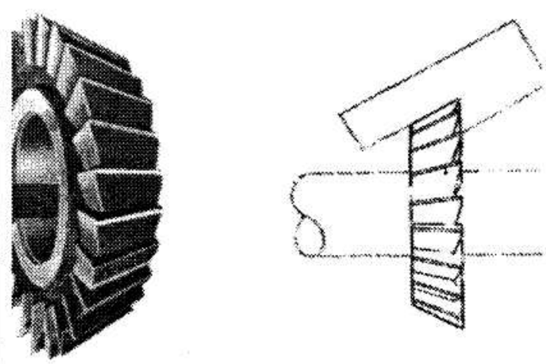


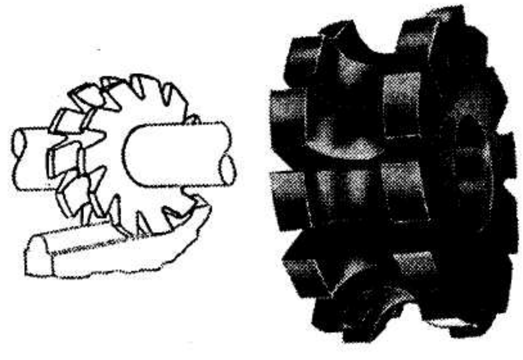
DRILL CHUCK ADAPTOR - JTA/B

METAL CUTTING TOOLS

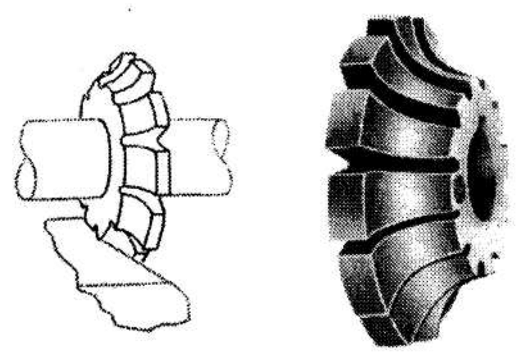

Cylindrical Cutter


Helical Cylindrical Cutter

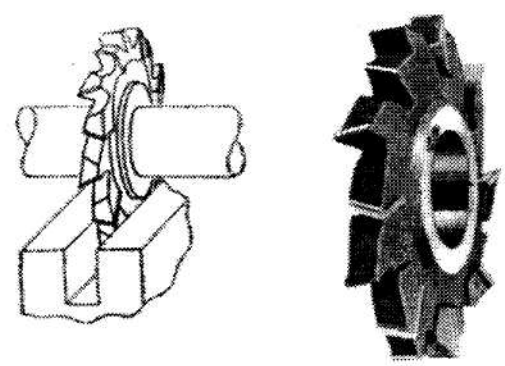

Single Angle Cutter



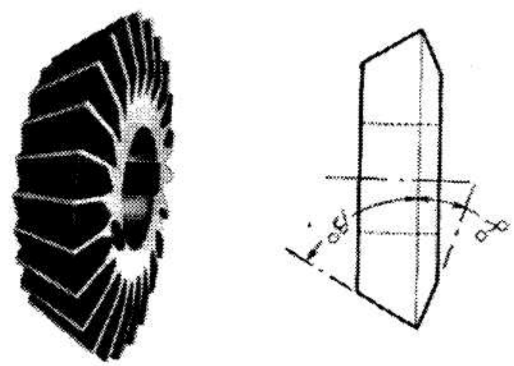
Concave Cutter



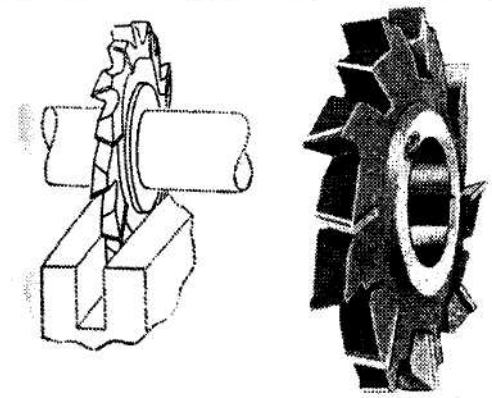
Single Corner Rounding Cutter



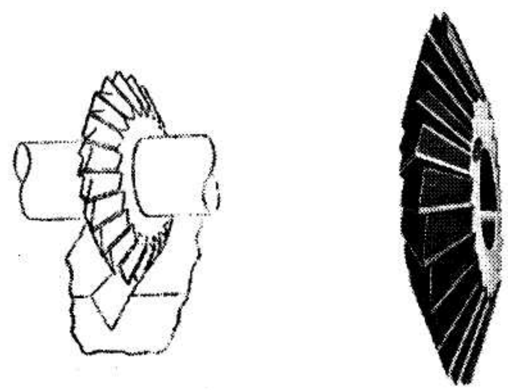
Side and Face Cutter



Double Angle Cutter

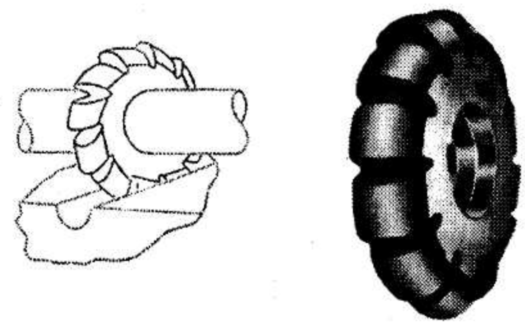


Staggered Tooth side & Face cutter



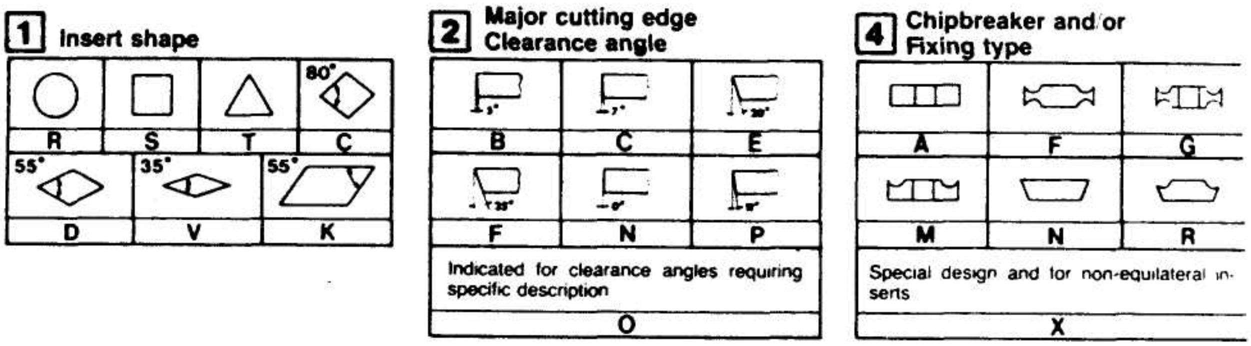
Equal Angle Cutter

Slotting Cutter



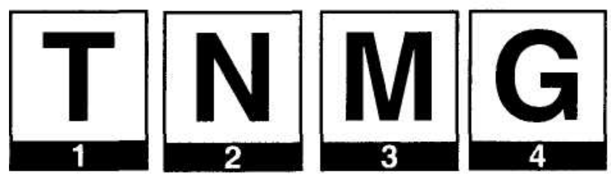
Convex Cutter

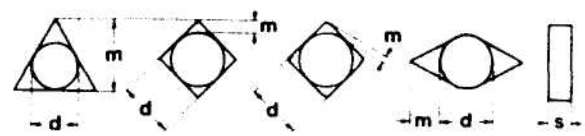
ISO code key - indexable inserts



3. TOLERANCES, mm

Class	m	S	d
A. C.H E G J K L M U	± 0.005 ±0.013 ±0.013 ±0.25 ±0.25 ± 0.025 ±0.013 ± 0.025 ± 0.08 - ± 0.20 ±0.13-±0.38	± 0.025 ± 0.025 ± 0.025 ± 0.025 ±0.13 ± 0.025 ± 0.025 ± 0.025 ±0.13 ±0.13	±0.025 ± 0.025 ±0.013 ± 0.025 ±0.025 ±0.05- ±0.15 ±0.05- ±0.15 ±0.05- ±0.15 ±0.05- ±0.15 ± 0.08 - ± 0.25

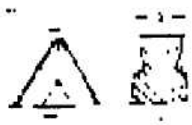




Vanes depending on the insert size for insert shapes R,S,T,C,D and V see tables below.

d	m				d	•
	Class M Insert shapes S.T.C	Insert shape D	Insert shape V	Class U Insert shapes S,T	Classes J,K,L,M Insert shapes S,T,C,R	Class U Insert shapes S,T
4.76	±0.08	±0.11	±0.015	±0.13	±0.05	±0.08
5.0	± 0.08	±0.11		±0.13	±0.05	±0.08
5.56	±0.08	±0.11		±0.13	±0.05	±0.08
6.0	±0.08	±0.11		±0.13	±0.05	±0.08
6.35	±0.08	±0.11		±0.13	±0.05	±0.08
7.94	±0.08	±0.11		±0.13	±0.05	±0.08
8.0	±0.08	±0.15		±0.13	±0.05	±0.08
9.525	±0.08	±0.18		±0.13	±0.05	±0.08
10.0	±0.08	±0.18		±0.13'	±0.05	±0.08
12.0	±0.13			±0.20	±0.08	±0.13
12.7*	±0.13			±0.20 ±	±0.08	±0.13
151875	±0.15			0.27	±0.10	±0.18
16.0	±0.15			±0.27	±0.10	±0.18
19.05	±0.15			±0.27	±0.10	±0.18
20.0	±0.15			±0.27	±0.10	±0.18
25.0	±0.18			±0.38	±0.13	±0.25
25.4	±0.18			±0.38	±0.13	±0.25
11.75	± 0.20			±0.38	±0.15	±0.25
17.0	±0.20			±0.38	±0.15	±0.25

6 Thickness, mm

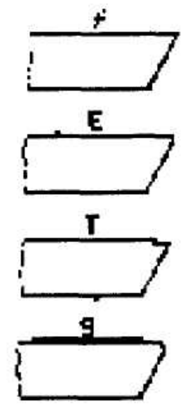


01 s = 1.59
02 s = 2.38
03 s = 3.18
13 s = 3.97
04 s = 4.76
05 s = 5.56
06 s = 6.35
07 s = 7.94
09 s = 9.52

7 Radius, mm

00	Sharp point		
00	Round insert, d - inch basic size converted into mm.		
M0	Round insert, d-mm basic size		
02	r = 0.2	15	r = 1.5
04	r = 0.4	16	r = 1.6
05	r = 0.5	20	r = 2.0
08	r = 0.8	24	r = 2.4
10	r = 1.0	32	r = 3.2
12	r = 1.2	40	r = 4.0

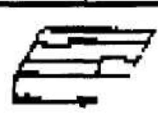
8 Cutting edge condition



11	03	04	T	R	-15
5	6	7	8	9	10

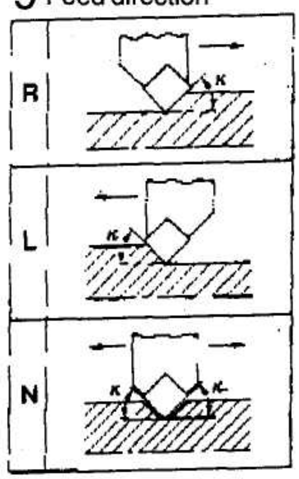
5 Insert size, mm

d	R	S	T	C	D	V
3.97			06			
5.0	05					
5.56			09			
6.0	06					
6.35			11	06	07	
8.0	08					
9.525	09	09	16	09	11	16
10.0	10					
12.0	12					
12.7	12	12	22	12	15	
15.875	15	15	27	16		
16.0	16					
19.05	19	19	33	19		
20.0	20					
25.0	25					
25.4	25	25		25		
31.75	31					
32.0	32					



For insert shape K only the theoretical cutting edge length is indicated Applies to insert type KNUX and KNMX.

9 Feed direction

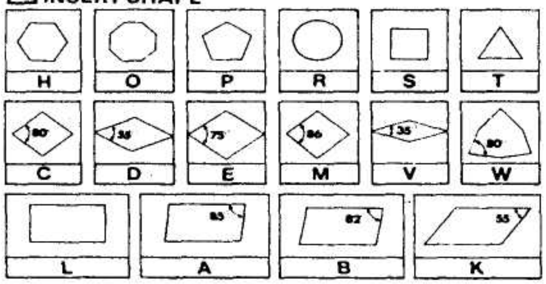


10 **Manufacturers option**

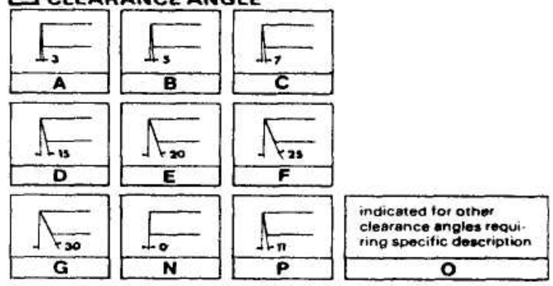
The ISO code includes nine symbols whereof 8 and/or 9 are used only when required. In addition, the manufactured may and further symbols joined to the ISO code through a hyphen (e.g. -15 for chipbreaker design).

ISO CODES — Indexable inserts

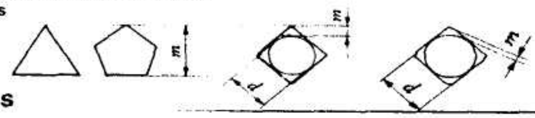
1 INSERT SHAPE

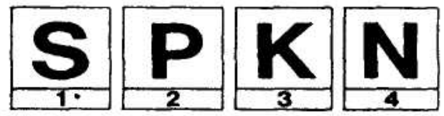


2 MAIN CUTTING EDGE CLEARANCE ANGLE



d : theoretical diameter of inscribed circle
s : insert thickness
m : see fig.





3 TOLERANCES

Letter Symbol	Tolerances in mm			Tolerances in inches		
	m	i	a	m	i	a
A	± 0.005	± 0.025	± 0.025	± 0.0002	± 0.001	± 0.0010
F	± 0.005	± 0.025	± 0.013	± 0.0002	± 0.001	± 0.0005
C	± 0.013	± 0.025	± 0.025	± 0.0005	± 0.001	± 0.0010
H	± 0.013	± 0.025	± 0.013	± 0.0005	± 0.001	± 0.0005
E	± 0.025	± 0.025	± 0.025	± 0.0010	± 0.001	± 0.0010
G	± 0.025	± 0.013	± 0.025	± 0.0010	± 0.005	± 0.0010
J	± 0.005	± 0.025	± 0.05 ± 0.13	± 0.0002	± 0.001	± 0.002 ± 0.005
K	± 0.013	± 0.025	± 0.05 ± 0.13	± 0.0005	± 0.001	± 0.005 ± 0.005
L	± 0.025	± 0.025	± 0.05 ± 0.13	± 0.0010	± 0.001	± 0.002 ± 0.005
M	± 0.08 ± 0.10	± 0.13	± 0.05 ± 0.13	± 0.003 ± 0.002	± 0.005	± 0.002 ± 0.005
U	± 0.13 ± 0.38	± 0.13	± 0.08 ± 0.25	± 0.005 ± 0.015	± 0.005	± 0.003 ± 0.010

i) These tolerances normally apply to inserts with parallel land
m) The tolerance is dependent upon the insert size and should be indicated for each insert according to the standard tolerance for the corresponding size.

For inserts with C,E,H,M,O,P,S,T,W shapes having a point (included plan) angle of 60° and larger, values for tolerance class M and U on m and with M,J,K,L and U on d are indicated in the following table.

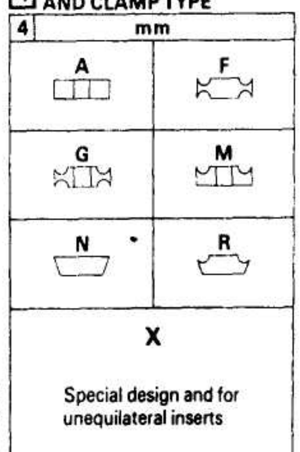
Inscribed circle diameter d		Tolerance for m				Tolerances for d			
		Class M		Class U		Class M,J,K,L		Class U	
mm	in	mm	in	mm	in	mm	in	mm	in
6.35	0.250	± 0.08	± 0.003	± 0.13	± 0.005	± 0.05	± 0.002	± 0.08	± 0.003
9.525	0.375	± 0.08	± 0.003	± 0.13	± 0.005	± 0.05	± 0.002	± 0.08	± 0.003
12.7	0.500	± 0.13	± 0.005	± 0.20	± 0.008	± 0.08	± 0.003	± 0.13	± 0.005
15.875	0.625	± 0.15	± 0.006	± 0.27	± 0.011	± 0.10	± 0.004	± 0.18	± 0.007
19.05	0.750	± 0.15	± 0.006	± 0.27	± 0.011	± 0.10	± 0.005	± 0.18	± 0.007
25.4	1.000	± 0.18	± 0.007	± 0.38	± 0.015	± 0.13	± 0.005	± 0.25	± 0.010

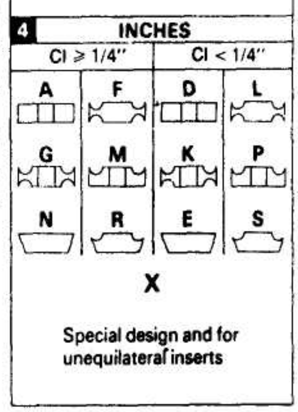
Insert shapes

For thombic inserts having a point angle of 55° (shape D) values for tolerance class m on m and d are indicated in the following table.

Inscribed circle diameter		Tolerance for m		Tolerance for d		Insert shape
mm	m	mm	m	mm	m	
6.35	0.250	± 0.11	± 0.004	± 0.05	± 0.002	
9.525	0.375	± 0.11	± 0.004	± 0.05	± 0.002	
12.20	0.500	± 0.15	± 0.006	± 0.08	± 0.003	D
15.875	0.625	± 0.15	± 0.007	± 0.10	± 0.004	
14.38	0.256	± 0.18	± 0.007	± 0.10	± 0.003	

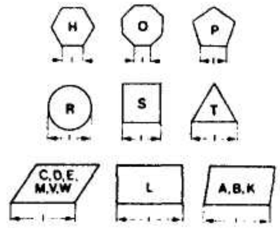
4 CHIPBREAKER AND CLAMP TYPE





5 CUTTING EDGE LENGTH

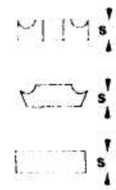
mm



Integers to be preceded by 0, eg. 9.52 mm indicated with 09.

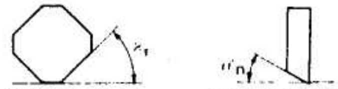
6 THICKNESS

mm



Integers to be preceded by 0, eg. 4.76 indicated with 04.

7 PARALLEL LAND, CLEARANCE ANGLE



$\varkappa_r$	
A	45
D	60
E	75
F	85
P	90

α_n	
A	3
B	5
C	7
D	15
E	20
F	25
G	30
N	0
P	11

Special design

Z	mm	Z	mm
Z			in

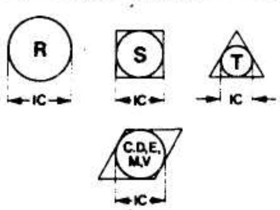
12	03	ED	R
5	6	7	9

The ISO code includes nine symbols whereof 8 and/or 9 are used only when required. In addition, the manufacturer may add further symbols joined to the ISO code through a hyphen (eg. -71 for the chipbreaker design).

3	2
5	6

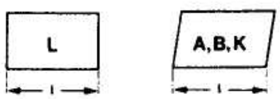
5 INSCRIBED CIRCLE

INCHES



in 1/32" for IC<1/4"
in 1/8" for IC≥1/4"

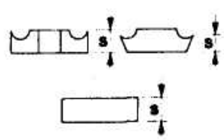
Cutting edge length



For rectangular and rhomboidal inserts, the cutting edge length is indicated in 1/4".

6 THICKNESS

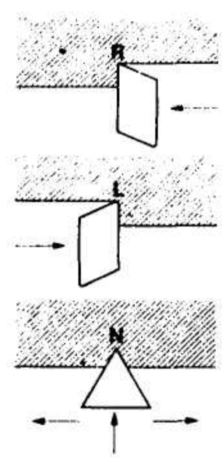
INCHES



indicated in 1/32" for inserts having IC<1/4" and in 1/16" for inserts having IC≥1/4".

For rectangular and rhomboidal inserts, width is used as reference instead of IC.

9 FEED DIRECTION

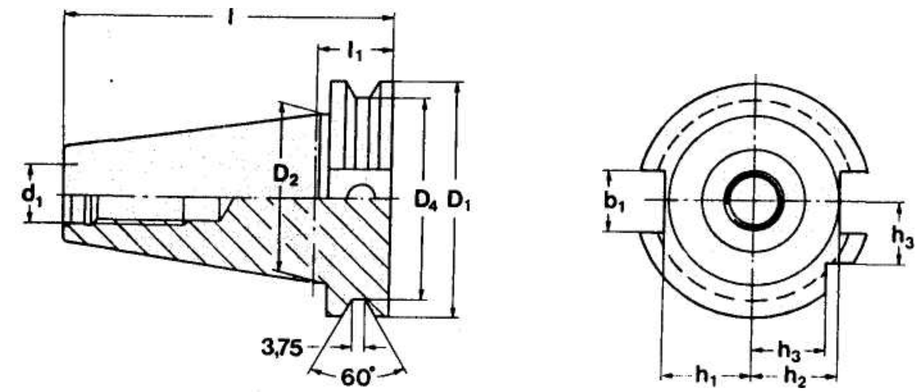


CUTTING EDGE LENGTH in mm (pos. 5) to IC in inches.

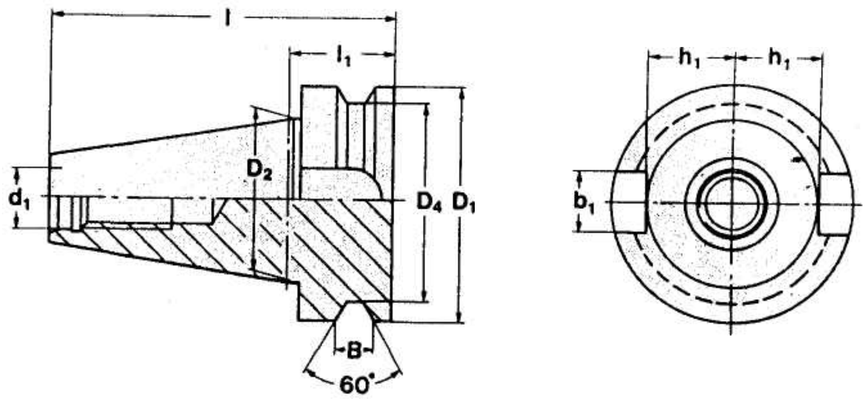
	06	09	11	16	22	27	33	44
				09	12	15	19	25
55°			07	11	15	19	23	31
80°			06	09	12	16	19	25
IC=d	5/32"	7/32"	1/4"	3/8"	1/2"	5/8"	3/4"	1"

SHANK DETAILS

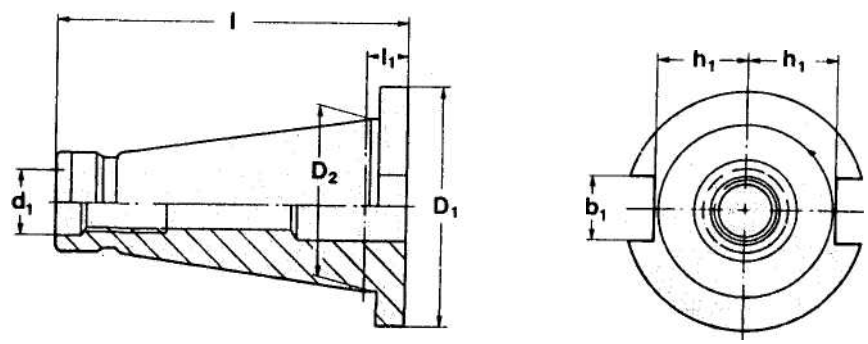
ISO 7388-1
DIN 69871



MAS/BT 403

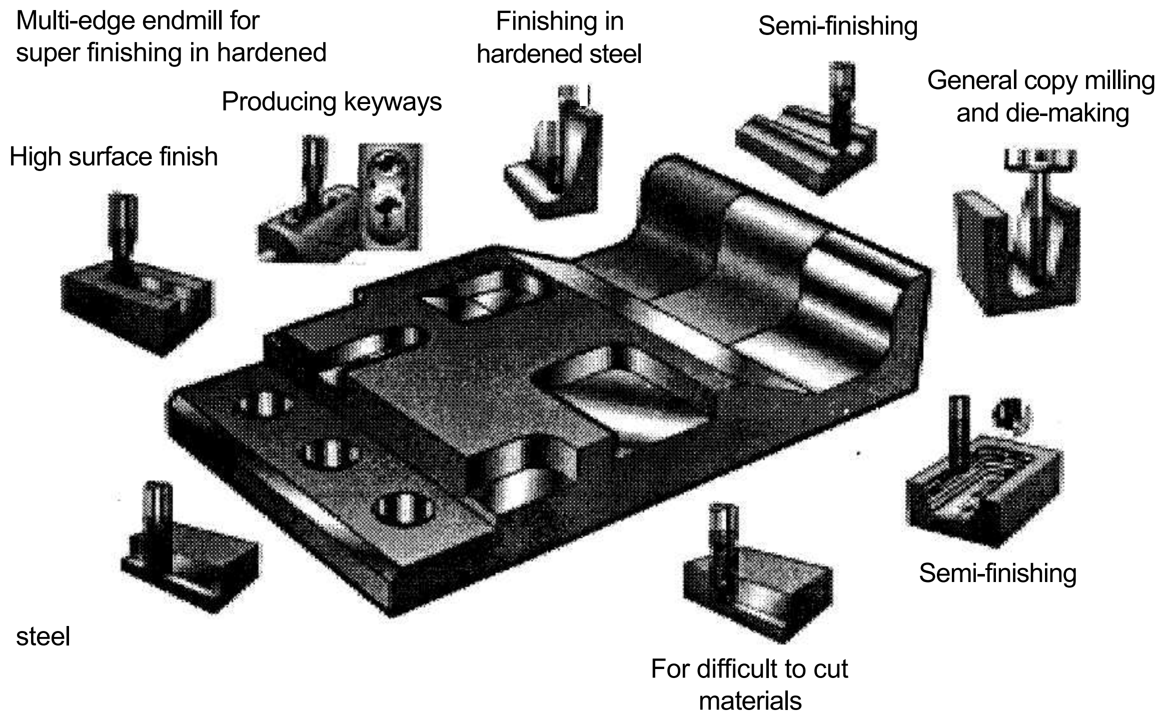


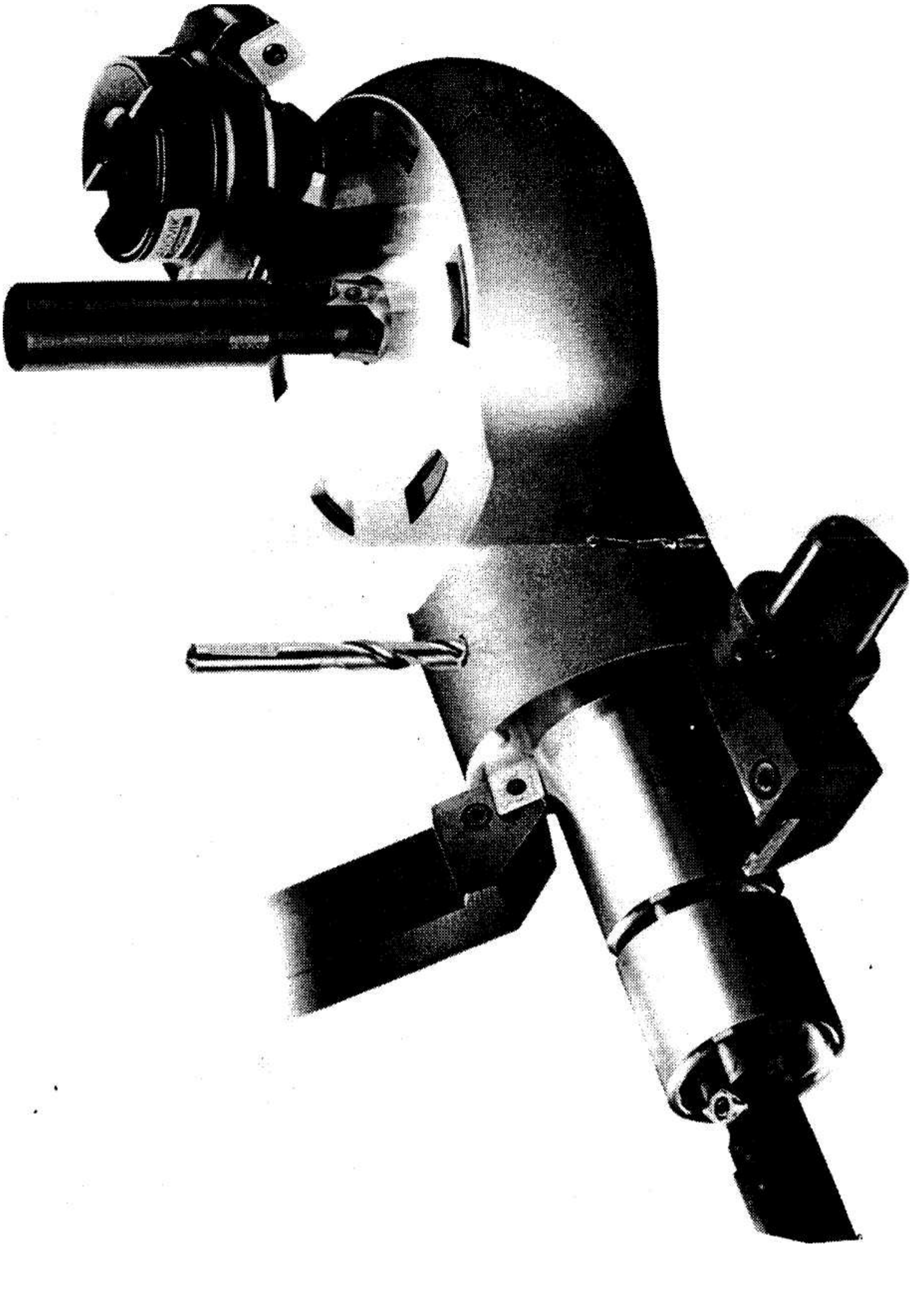
DIN 2080

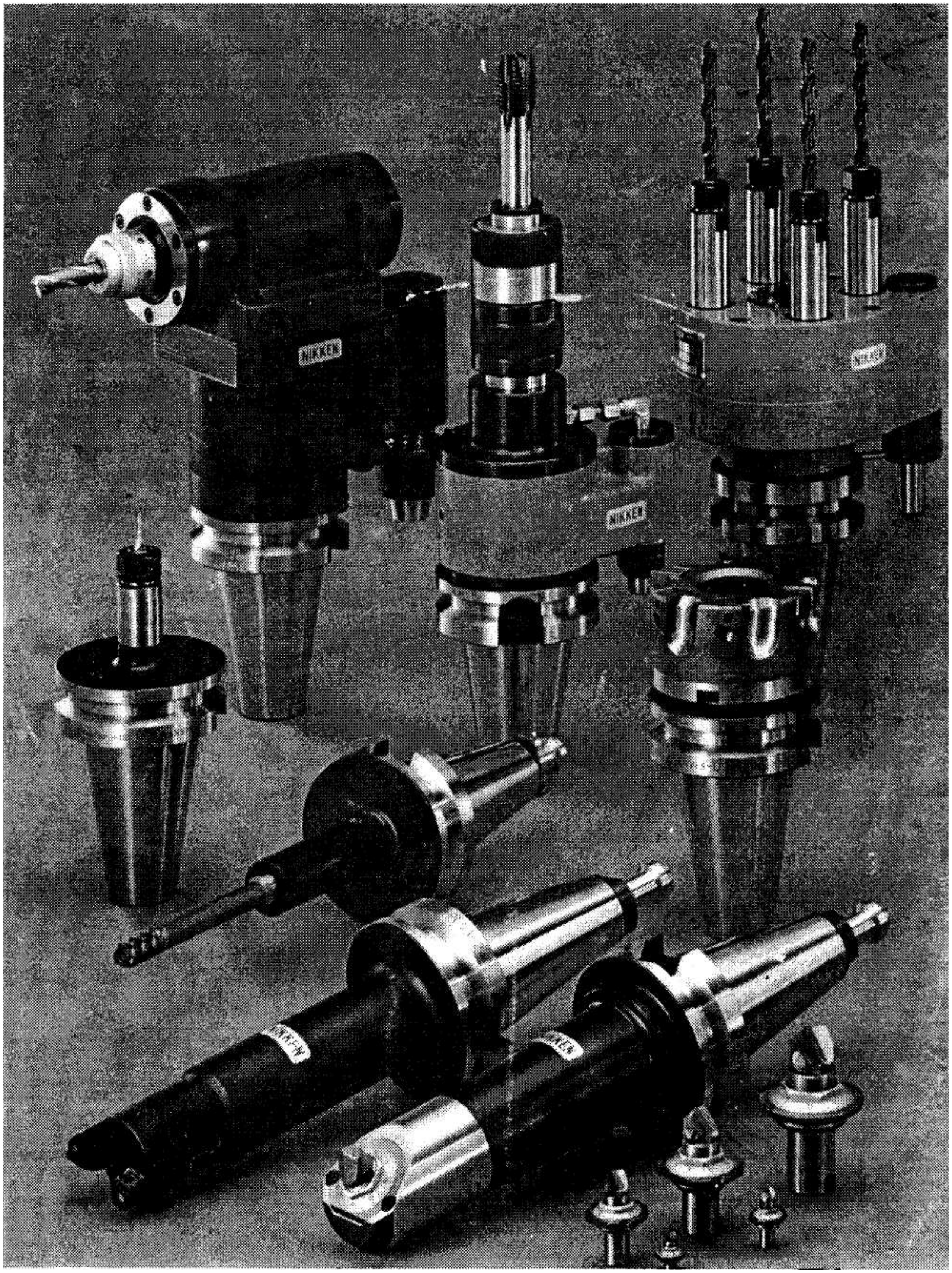


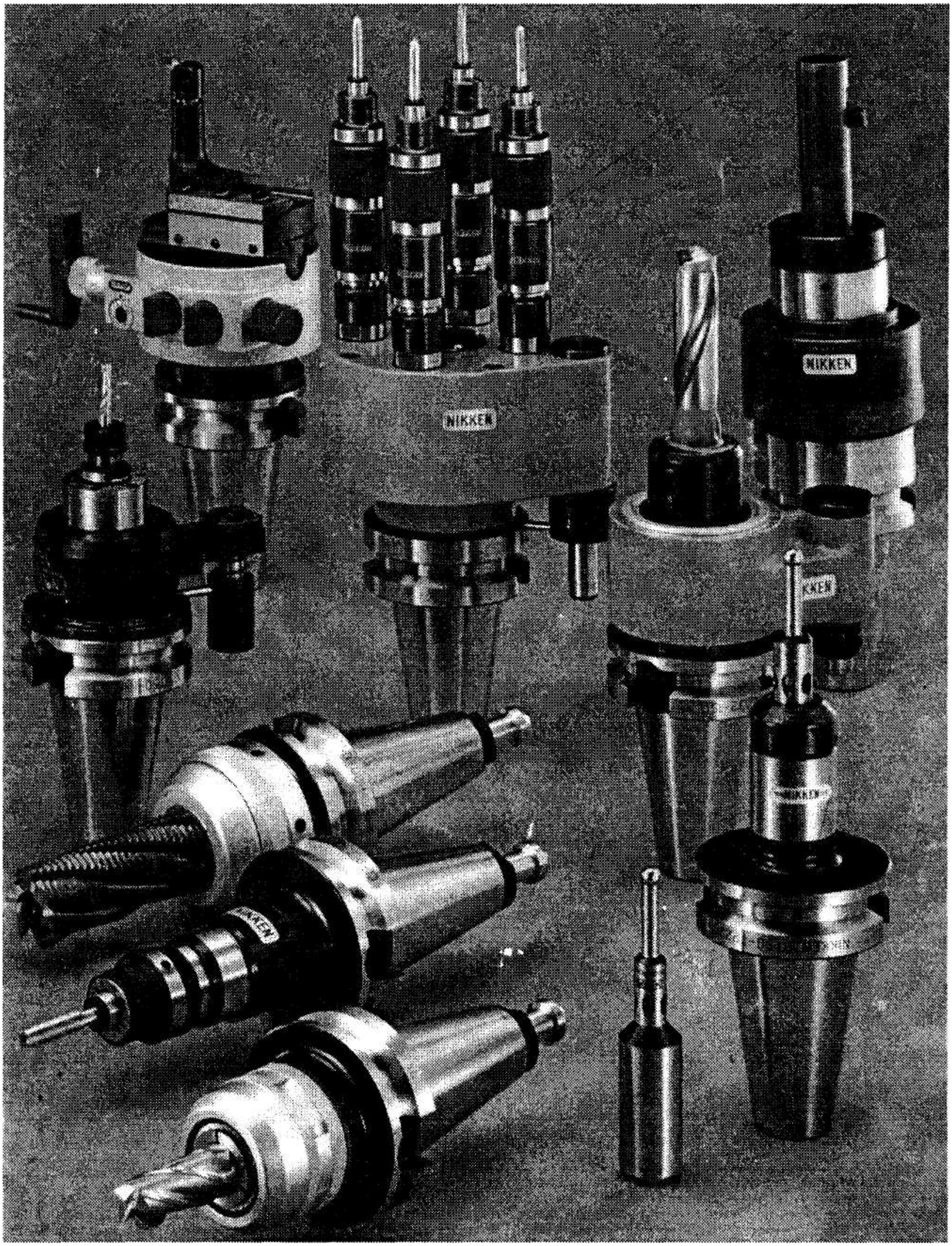
Machine design	Taper	Dimensions b1	B	d1	D1	D2	D4	hi	h2	h3	l	
ISO/DIN	40	16.1	-	16	63.55	44.45	56.25	22.8	25.0	18.5	87.5	19.1
	45	19.3	-	20	82.55	57.15	75.25	29.1	31.3	24.0	101.8	19.1 25.7
	50	-	24	97.5	69.85	91.25	35.5	37.7	30.0	120.85	19.1	
MAS/BT	40	16.1	10	16	63.0	44.45	53.0	22.5	-	-	92.4	27.0 25.7
	50	15	24	100.0	69.85	85.0	35.4	-	-	139.8	38.0	
DIN 2080	40	16.1	-	16	63.0	44.45	-	22.5	-	-	105.0	11.6
	50	25.7	■	24	97.5	69.85	-	35.3	-	-	142.0	15.2

EASY OPERATIONS WITH ENDMILLS






NIKKEN
NIKKEN
NIKKEN


NIKKEN
NIKKEN
NIKKEN

SECTION 9

ECONOMICS OF CNC MACHINES

ECONOMICS OF CNC MACHINES

Once the technical suitability is assessed for processing of components on CNC machines, it is very essential to assess the economic suitability of CNC machines to process such components.

In order to ascertain the assessment criteria and evaluation points, it is important to recognise the specific advantages of NC machines over the conventional ones since the work pieces are primarily suitable for NC machining.

Some of the aspects to be considered and analyzed while calculating the economic suitability of taking up the jobs on CNC machines are:

4. Complex geometric forms related with short tool changing times, minimum requirement of tooling and faster processing.

2. Requirement of higher and uniform accuracies and the net result in rejections and rework.

3. Preparatory and planning time and their cost.

4. Handling costs and reduction of number of processing stages.

5. Batch quantities to be processed.

The total manufacturing cost can be analyzed as follows:

Total Manufacturing Cost

Cost of preparing for Production	Production costs	Other related costs
Planning cost Programming cost Tape preparation cost Jigs & fixtures design, manufacture & proving cost	capital equipment cost Depreciation Interest cost Maintenence cost supervising cost stores and office services cost, etc.,	order scheduling cost Machining set up cost Tooling costs of requisitioning pre-setting, resharpening and restoring.

I. Calculation of Machine hour rate for conventional and NC Machines.

	Particulars	Conventional Machine (Rs.)	NC Machine (Rs.)
01.	Total cost of equipment		
	up to erection	2,20,000	22,00,000
1.	Life of Machine	10 Years	10 Years
2.	Residual value of machine	20,000	2,00,000
3.	Depreciation method	Straight line method	straight line method

Machine hour rate calculation (Assume 2000hrs per annum)

		Expenditure in-curred per annum (Rs.)	
4.	Depreciation	20,000	2,00,000
5.	Interest on capital @ 15%	33,000	3,30,000
6.	Floor area occupation cost	4,000	. 6,000
7.	Power cost	5,000	10,000
8.	Shop supervision	2,000	5,000
9.	Repairs & maintenance	2,000	22,000
10.	Operator wages	18,000	25,000
11.	Over heads	4,000	10,000
1.	consumables	2,000	12,000
		90,000	6,20,000

Machine hour rate 90,000/2000 = Rs. 45/- 6,20,000/2000 = Rs. 310/-

II. Calculation of production cost - conventional and NC machines

Job	:	Rotor shaft
Material	:	Alloy steel
Batch Quantity	:	10
Operations	:	Various turning operations

Machine	Conventional Lathe M/c.			CNC Lathe		
Machine hour cost	Rs. 45			Rs. 310		
Production cost details	Time		cost	Time		Cost
	Hrs- mts		Rs.	Hrs.- mts		Rs.
Planning cost	80	-	2,000	40	-	1,000
Programming cost	.	.	.	60	-	1,500
Toolings design, manufacture						
& proving cost	-	-	5,000	-	-	2,000
Pre production cost/batch	-	-	7,000	-	-	* 4,500
Tool presetting cost	-	-	-	-	30	10
Set up cost	8	-	360	-	30	155
Tape proving cost	-	-	-	-	30	155
Machining cost	12	-	540	1	30	465
Inspection cost	1	-	25	-	30	10
Processing cost/piece			925			795
Total cost / item			1,625			1,245

In other words the economics can also be analyzed in terms of both direct and indirect savings.

Direct savings:

Economic of FTF production time.

(Conventional machining time - NC machining time) x Machine hour rate.

Indirect savings :

	Items to be analyzed	Anticipated savings
1.	Reducing Cutting tool change time	20% of tool allowance
2.	Improved accuracy	5% of direct labour cost
3.	Less inspection	30% of inspection cost
4.	Reduced set up time	80% of set up cost
5.	Reduced scrap	30% of scrap cost
6.	Longer tool life	30% of tool cost
7.	Lower fixture cost	75% of durable fixture cost
8.	Less material handling	5% of material handling cost
9.	Control of cycle time	10% increased production
10.	Reduction of inventory	5% of money value of inventory