GLOBAL ACADEMY OF TECHNOLOGY

Rajarajeshwari Nagar, Bangalore – 560 098

DEPARTMENT OF MECHANICAL ENGINEERING

CNC PROGRAMMING LABORATORY [20MEDL76] MANUAL

NAME					
USN			**YEAR**		
SEMESTER		**SECTION**		**BATCH**	

<u>PREPARED BY</u>

Mr. R. KIRAN Assistant Professor

Dr. BHARAT VINJAMURI Professor

Dr. RAVI KUMAR D.V Professor

Mr. SUDARSHAN M S Lab Instructor

INSTITUTE VISION

Become a premier institution imparting quality education in engineering and management to meet the changing needs of society.

INSTITUTE MISSION

M 1 Create environment conducive for continuous learning through quality teaching and learning processes supported by modern infrastructure.

M 2 Promote Research and Innovation through collaboration with industries.

M 3 Inculcate ethical values and environmental consciousness through holistic education programs.

DEPARTMENT VISION

Become one of the leading providers of education in Mechanical Engineering with emphasis on research, development and innovation for the benefit of society

DEPARTMENTMISSION

M 1 Impart quality technical education in the field of mechanical engineering through excellent teaching-learning process, modern infrastructure and computing tools

M 2 Prepare students for successful careers by providing placements and encouraging research, development and innovation through industry-institute interaction

M 3 Instill professional ethics and environmental consciousness amongst students through inclusive development programs

PROGRAM EDUCATIONAL OBJECTIVES [PEOS]

Mechanical Engineering graduates will be able to:

PEO 1 Analyze, design and evaluate mechanical components and systems using conventional and modern IT tools with due consideration for ethics, public health, safety and environment

PEO 2 Apply the knowledge of mechanical engineering to solve problems of social relevance and/or pursue higher education and research

PEO 3 Work effectively as individuals and as team members in multidisciplinary projects

PEO 4 Engage in lifelong learning, career growth and adapt to changing professional and societal needs

PROGRAM SPECIFIC OUTCOMES [PSO]

After successful completion of Mechanical Engineering Program, the graduates will be able to:

PSO1 Specify, design, and analyze machine elements using CAD / CAE software's.

PSO2 Evaluate thermal performance of Heating, Ventilation & Air-Conditioning systems, Electronic systems, Solar Roof Top Photo-Voltaic systems using experimental approach or /and CFD tools and design these systems for better performance

PSO3 Develop composite materials, manufacturing process and products in an efficient, safe and cost-effective manner

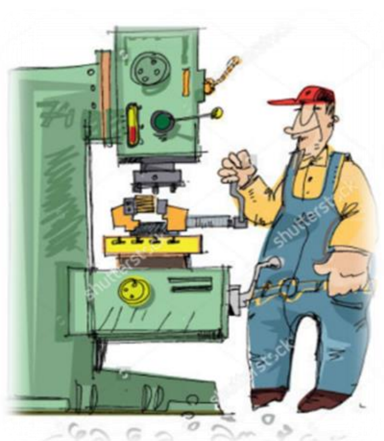
PROGRAM OUTCOMES (POS)

Engineering Graduates will be able to:

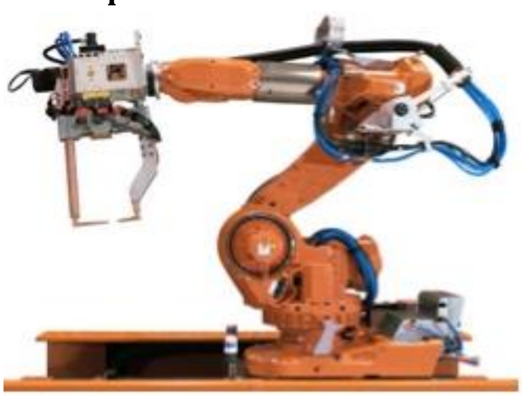
1. **Engineering knowledge**: Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.
2. **Problem analysis**: Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.
3. **Design/development of solutions**: Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.
4. **Conduct investigations of complex problems**: Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.
5. **Modern tool usage**: Create, select, and apply appropriate techniques, resources, and modernengineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.
6. **The engineer and society**: Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.
7. **Environment and sustainability**: Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.
8. **Ethics**: Apply ethical principles and commit to professional ethics and responsibilities and norms ofthe engineering practice.
9. **Individual and team work**: Function effectively as an individual, and as a member or leader indiverse teams, and in multidisciplinary settings.
10. **Communication**: Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.
11. **Project management and finance**: Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in a team, to manage projects and in multidisciplinary environments.
12. **Life-long learning**: Recognize the need for and have the preparation and ability to engage inindependent and life-long learning in the broadest context of technological change.

RELEVANCE OF "CNC LAB" IN MECHANICAL ENGINEERING

Evolution of CIM



During the course of history, people have changed the manufacturing process dramatically. Instead of items being produced by hand, the owners of the facilities created ways to have machines produce the items. The impact of changing the way items were manufactured had a wide reach. Industries such as textile manufacturing, mining, glass making and agriculture all had undergone changes. During Industrial Revolution, changes took place in how goods were produced. Machinery such as the spinning wheel to produce textiles, the water wheel used to power machinery and the steam engine were invented. These inventions aided in speeding up the production of manufactured items. However, with materials now being produced quicker and cheaper, the need for manufactured goods was greater than the supply. These demands became increasingly difficult to achieve and ultimately led to utilize semi-automated machines controlled by computers. As the demand for products were increasing, many production facilities came into existence giving rise to a competitive environment, where quality played an important role. To overcome these demands factories first opened their doors to modern industrial robots. Robots could perform tasks that humans often found dangerous or boring, and it could do them with consistent speed and precision. Due to globalization, there was a necessary to integrate all the various activities in an organization which lead to integration of design, manufacturing and sales with the help of computers thereby giving rise to the concept of "Computer Integrated Manufacturing".



Computer Integrating manufacturing denotes the use of computer systems to design the products, Plan the production, and control the operations to perform various business-related functions needed in a manufacturing firm.

CIMlab includes CNC programs to generate a part and simulate the program to understand how a part looks like and to rectify the errors made while writing a program. These days the CNC machines are found in almost all industries, from a small scale industry to big companies.

Using a computer numerically controlled (CNC) program, the CNC machines can cut, shape, and finish materials such as metals, plastics, or wood into usable components.

Fig: CNC turning center

Applications of CIM Lab

In many industries thin plates like steel plates are required for various purposes, in fabrications industry the machining operations are performed on such plates. In these industries the CNC machines are used for various machining operations like shearing, flame or plasma cutting, punching, laser cutting, forming, and welding and many other applications. To bring the plates to their final shape CNC lasers and CNC plasma cutters are used commonly.

Fig: Robots performing welding, riveting, bonding and installing a component

Fig: CNC laser cutting

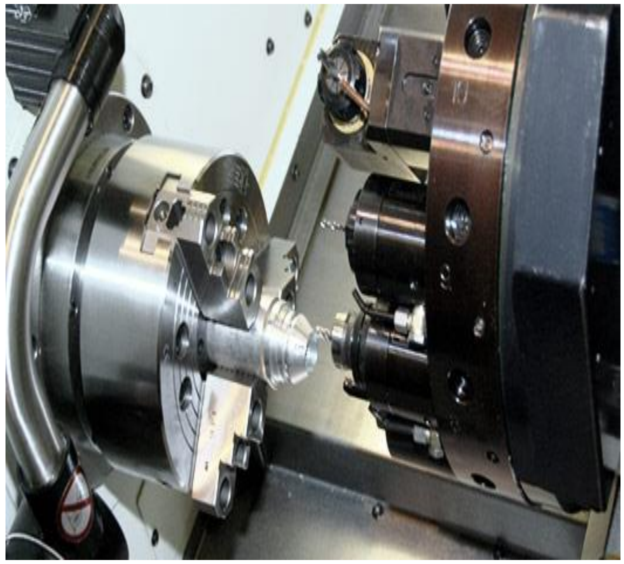


Fig: CNC Turning

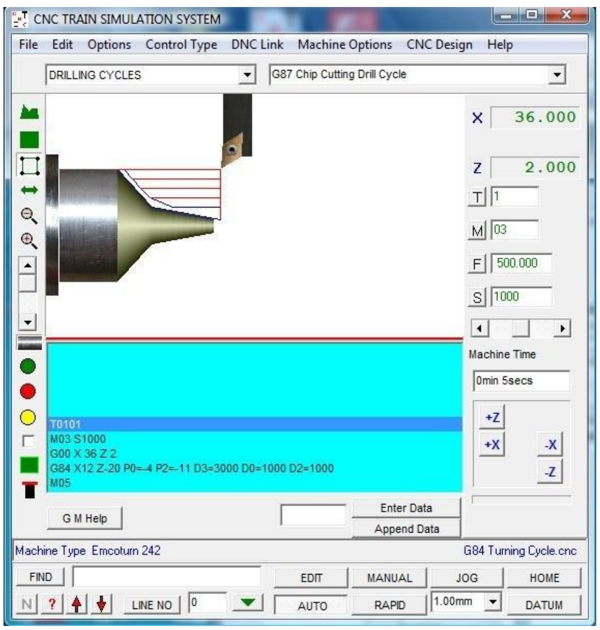


Fig: Simulation of CNC Turning

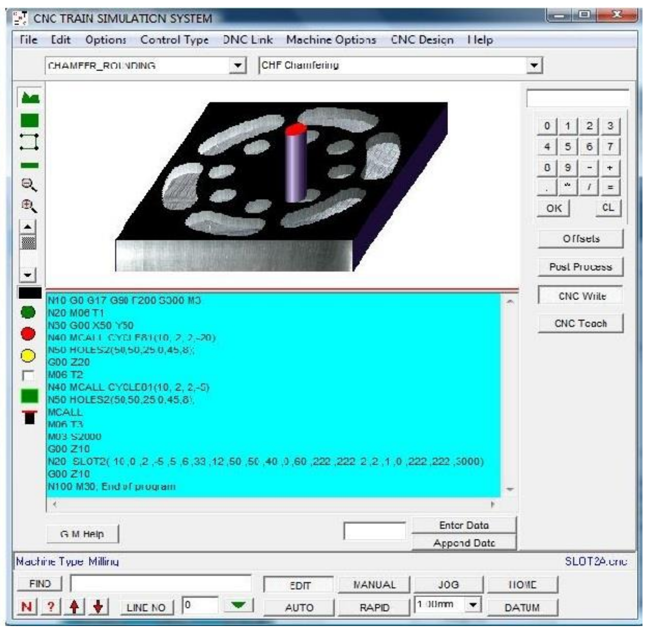


Fig: Simulation of CNC Milling

INTRODUCTION TO CAM

NC Part Programming

NUMERICAL CONTROL PROCEDURE

The following are the basic steps in NC procedure

- ✓ **Process Planning**
- ✓ **Part Programming**
- ✓ **Part Program entry**
- ✓ **Proving the part program**
- ✓ **Production**

A) PROCESS PLANNING

The part programmer will often carry out the task of process planning. Process planning is the procedure of deciding what operations are to be done on the component, in what order, and with what tooling and work holding facilities. Both the process planning and part programming aspects of manufacture occur after the detail drawings of a component have been prepared. The following procedure may be used as a guide to assist the programmer, by describing each step required in preparing the method of production.

PROCESS PLANNING

- ✓ **Receive the part drawing from part drawing information, check suitability of part to be machined against the machine capacity.**
- ✓ **Determine a method of driving the component (chuck type, chuck size, type of jaw) and the method of machining.**
- ✓ **Determine the tooling required to suit the method of machining and utilize as much as possible the tools which are permanently in the turret set upon the machine.**
- ✓ **Determine the order of machining and the tooling stations.**
- ✓ **Determine planned stops for checking dimensional sizes where required by operator**
- ✓ **Determine cutting speeds based on**
    - **Component material, method of driving, rigidity of component**
    - **Tooling selected for roughing and finishing**
- ✓ **Determine the depths of cut and feeds for roughing operations**
- ✓ **Determine surface finish requirements, the cutter nose radius most suited for finishing operations and determine feed rates.**
- ✓ **Allocates tool offsets as required**
- ✓ **Complete planning sheet**

B) PART PROGRAMMING

- ✓ **After completing the planning sheet, draw the component showing the cutter paths (a simple sketch is sufficient for simple components)**
- ✓ **Select a component datum and carryout the necessary calculations at slopes and arcs.**
- ✓ **Prepare tooling layout sheet showing tools to be used in the program and indicate the station number for each tool.**

- ✓ **Indicate the ordering code for each tool and grade and type of inserts to be used.**
- ✓ **Write the part program according to the sequence of operations.**

C) PART PROGRAM ENTRY (OR) TAPE PREPARATION

The part program is prepared / punched on a 25 mm wide paper tape with 8 tracks and is fed to MCU in order to produce a component of interest on machine tool. Other forms of input media include, punched cards, magnetic tape etc. The input to the NC system can be in two ways:
1. Manual data input
2. Direct Numerical control.

1) Direct Data Input (MDI): Complete part programs are entered into CNC control unit via the console keyboard. It is suited only for relatively simple jobs. The most common application for MDI is the editing of part programs already resident in controller's memory.
One variation of MDI is a concept called "Conversational Programming". CNC machines are programmed via a question and answer technique whereby a resident software program asks the operator a series of questions. In response to the operators input, and by accessing a preprogrammeddata file, the computer control can.

- ✓ **Select numerical values for use within machining calculations**
- ✓ **Perform calculations to optimize machining conditions**
- ✓ **Identify standard tools and coordinates**
- ✓ **Calculate cutter paths and coordinates**
- ✓ **Generate the part program to machine the component**

A typical dialogue from the machine would be as follows for the operator to identify such things as:

- ✓ **Material to be cut**
- ✓ **Surface roughness tolerance**
- ✓ **Machined shape required**
- ✓ **Size of the raw material blank**
- ✓ **Machining allowances**
- ✓ **Tools and tool detail etc.**

The operator may then examine and prove the program via computer graphics simulation on the console VDU. After this, the program is stored or punched on tape. Although there is some sacrifice in machine utilization, actual programming time is minimal and much tedious production engineering work is eliminated.
2) Direct Numerical Control: The process of transferring part programs into memory of a CNC machine tool from a host computer is called Direct Numerical Control or DNC

D) PROVING PART PROGRAMS

It is safe practice to check the programmed path for any interference between the tool and the work before using the part program for production. The proving part program is done by:

- ✓ **Visual inspection**
- ✓ **Single step execution**
- ✓ **Dry run**
- ✓ **Graphical simulation.**

Visual Inspection: It represents the method of checking visually the program present in the memory of the CNC machine. In this, actual program is run and the programmed movements in all axes are to be checked along with ensuring the tool offset and cutter compensation feature. This method represents the least form of verification and should not be relied up on entirely.

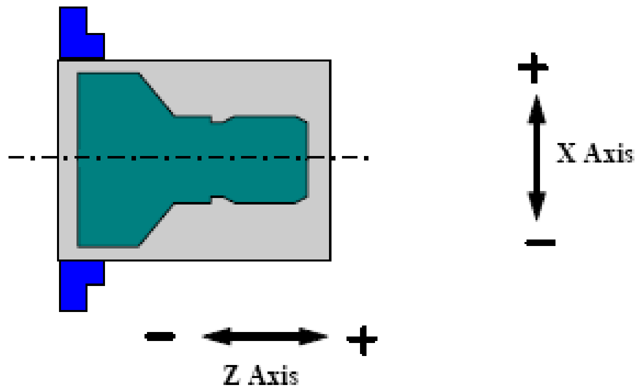
Single Step Execution: Before auto-running the part program should be executed in a step mode i.e. block by block. During this execution, spindle speed and feed rate override facilities are to be used so that axes movement can be easily monitored. This operation may be carried out with or without mounting the component on the machine.

Dry run: A dry run consists of running the part program in auto-mode. During this, the component is not installed on the machine table and the cutting is done in air. The purpose of this run is to verify the programmed path of the tool under continuous operation and to check whether adequate clearance exist between the clamping arrangement and other projections within the set up. Feed rate override facilities are used to slow down the speed of execution of the program.

Graphical simulation: A graphical simulation package emulates the machine tool and, using computer graphics, plots out the machine movements on a VDU screen. Machine movement often takes the form a cutting tool shape moving around the screen according to the programmed movements. When the tool shape passes over a shaded representation of the component, it erases that part of the component. The resulting shape after the execution represents the shape of the finished component. Any gross deviations from the intended tool path can be observed and any potential interference can be highlighted.

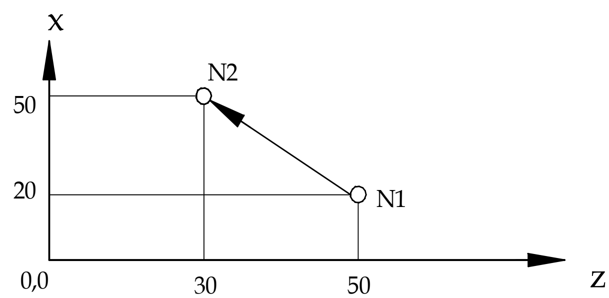
Programming for the Fanuc 0T controller

Axes convention



Absolute and Incremental co-ordinates

In the absolute programming, the end point of a motion is programmed with reference to the program zero point. In incremental programming, the end point is specified with reference to the current tool position.



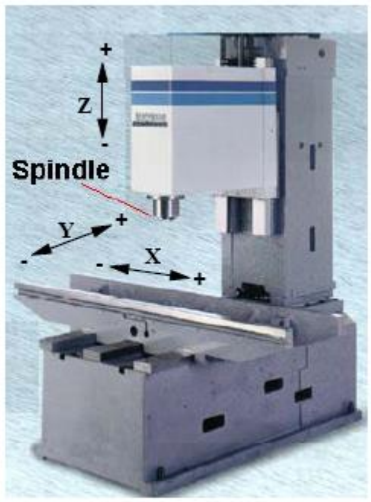
Absolute traverse to N1, then to N2
X20.0 Z50.0
X50.0 Z30.0
Absolute traverse to N1, incremental to N2
X20.0 Z50.0
U30.0 W-20.0

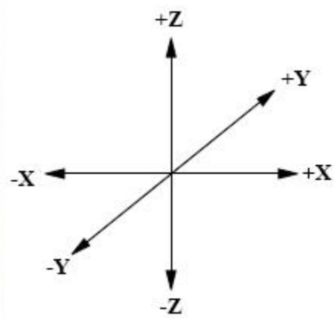
Programming for the Fanuc 0M controller

Axes convention

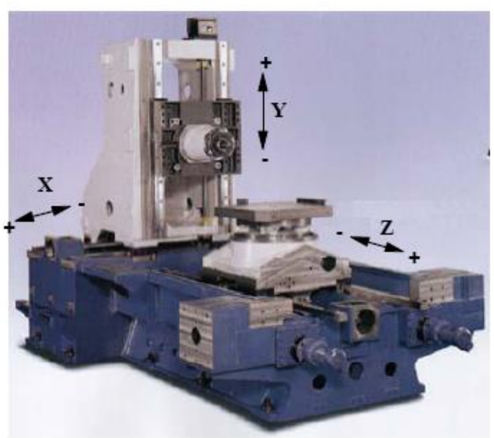
The tool can be moved to any position in a 3 dimensional Cartesian co-ordinate system. The Z axis is along the spindle axis. The X and Y axes are perpendicular to Z.

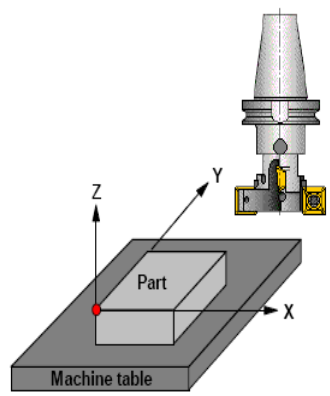
VMC (Vertical Machining Center)





HMC (Horizontal Machining Center)





List of G-codes for Milling

G-code	Function	G-code	Function
G00	Positioning rapid traverse	G58	Work co-ordinate system 5 selection
G01	Linear interpolation (feed)	G59	Work co-ordinate system 6 selection
G02	Circular interpolation CW	G74	Left hand tapping cycle
G03	Circular interpolation CCW	G76	Fine boring cycle
G04	Dwell	G80	Canned cycle cancel
G20	Inch unit	G81	Drilling cycle
G21	Metric unit	G82	Drilling cycle with dwell
G28	Automatic zero return	G83	Peck drilling cycle / deep drill
G30	2nd reference point return	G84	Tapping cycle
G40	Tool nose radius compensation cancel	G85	Boring / Reaming cycle
G41	Tool nose radius compensation left	G86	Boring cycle
G42	Tool nose radius compensation right	G87	Back boring cycle
G43	Tool length compensation	G90	Absolute command
G52	Local co-ordinate system	G91	Incremental command
G54	Work co-ordinate system 1 selection	G94	Feed per minute
G55	Work co-ordinate system 2 selection	G95	Feed per revolution
G56	Work co-ordinate system 3 selection	G98	Return to initial point in canned cycle
G57	Work co-ordinate system 4 selection	G99	Return to R point in canned cycle

List of M - codes for Milling

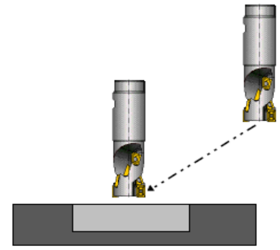
M codes vary from machine to machine depending on the functions available on it. They are decided by the manufacturer of the machine. The M codes listed below are the common ones.

M-codes	Function	M-codes	Function
M00	Optional program stop automatic	M07	Mist coolant ON (coolant 1 ON)
M01	Optional program stop request	M08	Flood coolant ON (coolant 2 ON)
M02	Program end	M09	Coolant OFF
M03	Spindle ON clock wise (CW)	M19	Spindle orientation
M04	Spindle ON counter clock wise (CCW)	M30	End of program, Reset to start
M05	Spindle stop	M98	Sub program call
M06	Tool change	M99	Sub program end

There are other M-codes for functions like gear change, pallet change, pallet clamp / unclamp, door open / close etc.

G00 Rapid traverse

When the tool being positioned at a point preparatory to a cutting motion, to save time it is moved along a straight line at Rapid traverse, at a fixed traverse rate which is pre-programmed into the machine's control system. Typical rapid traverse rates are 10 to 25 m /min., but can be as high as 80 m/min.



Format

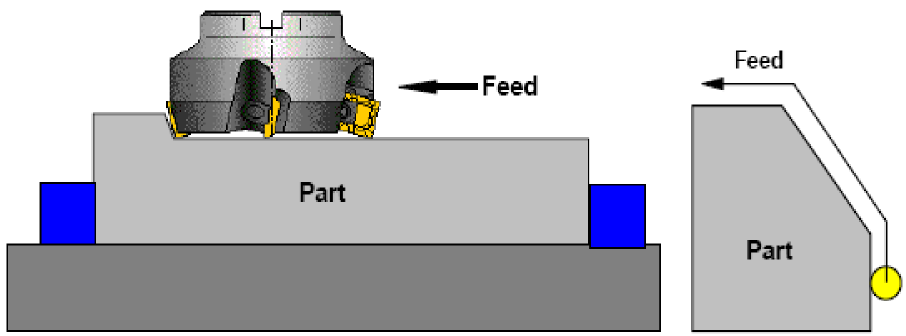
N_ G00 X__ Y__Z__

G01 Linear interpolation (feed traverse)

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

Format

N__ G01 X__ Y__Z__ F__



G02/03 Circular interpolation

Format

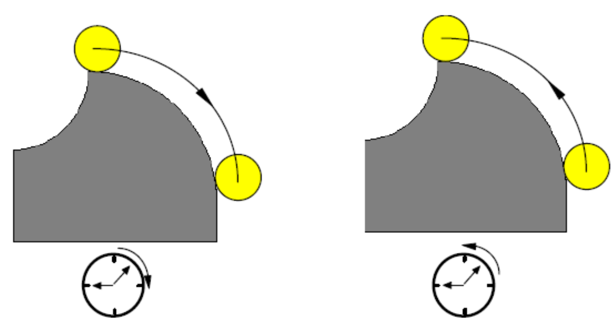
N__ G02/03 X__ Y__Z__ I__ J__K__ F__ using the arc center

or

N__ G02/03 X__ Y__Z__ R__ F__ using the arc radius

G02 moves along a CW arc

G03 moves along a CCW arc

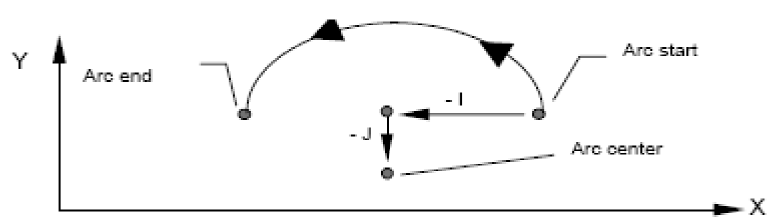


Arc center

The arc center is specified by addresses I, J and K. I, J and K are the X, Y and Z co-ordinates of the arc center with reference to the arc start point.

I = X coord. of center - X coord. of start point

J = Y coord. of center - Y coord. of start point

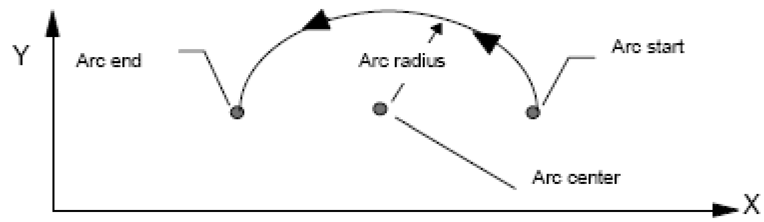


K = Z coord. of center - Z coord. of start point
I, J and K must be written with their signs.

Arc radius

The radius is specified with address R.

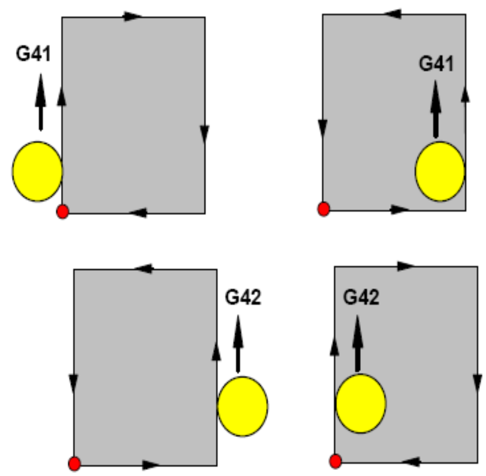
N__ G02 X__Y__ Z__ R__ F__
N__ G03 X__Y__ Z__ R__ F__



Block format

Cutter radius compensation (CRC)

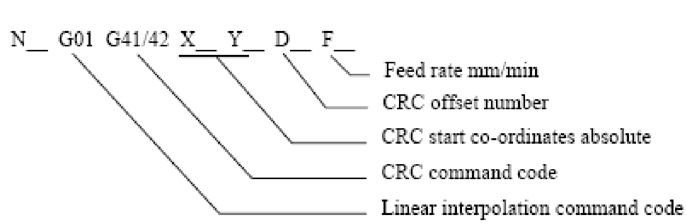
Cutter radius compensation is one of the CNC machining center programmer's most helpful programming tools when it is properly applied. It keeps the programmer from having to calculate the tool's center line coordinates, it allows the easy specification of roughing commands, and it allows a variety of cutter sizes to be used.



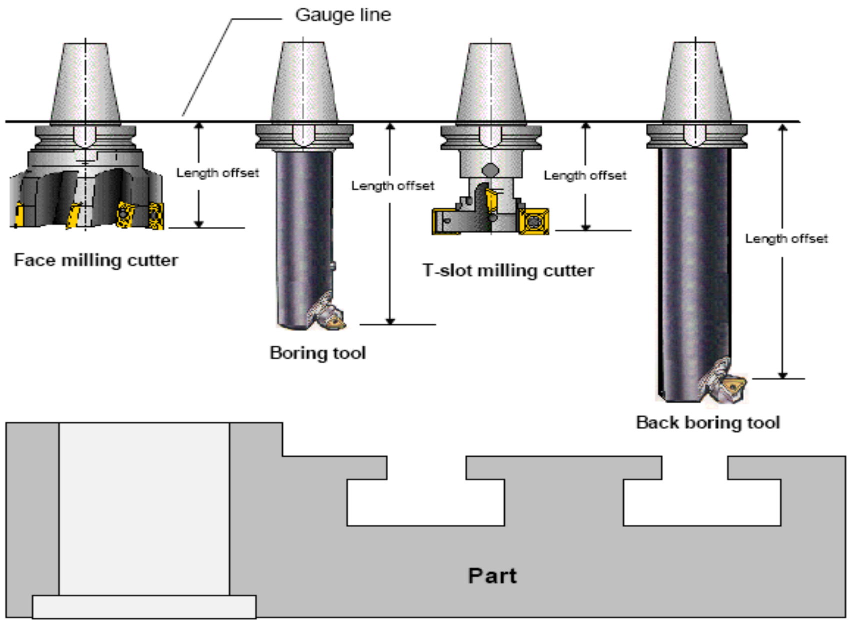
G40 Tool nose radius compensation Cancel
G41 Tool nose radius compensation Left
G42 Tool nose radius compensation Right

Necessity of using CRC

- Difficult to calculate cutter center co-ordinates
- If cutter center co-ordinates are used
- Same diameter of cutter is required to be used &Cutter wear cannotbecompensated



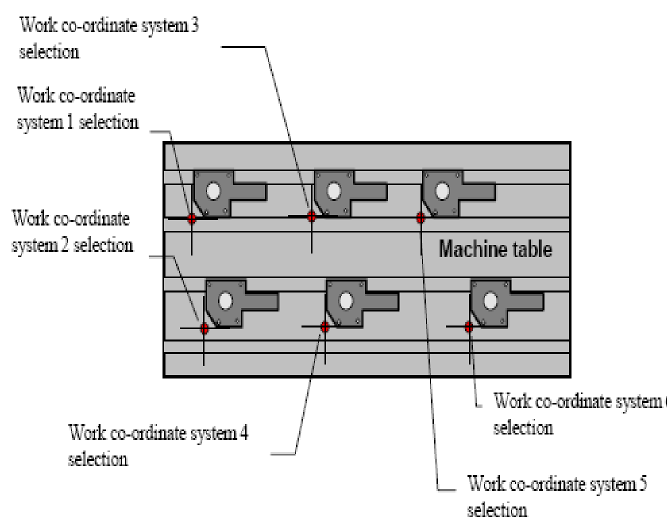
Tool length compensation (G43)



Different tools of different lengths are used in machining any part. The lengths of the tools are not considered in the part program. They are entered in the machine's memory, and are considered automatically for each motion in the program depending on the tool that is being used. The tool lengths in the Z direction are called the Tool length offsets.

Setting work co-ordinate system (G54 - G59)

G55 Work co-ordinate system 2 selection
G56 Work co-ordinate system 3 selection
G57 Work co-ordinate system 4 selection
G58 Work co-ordinate system 5 selection
G59 Work co-ordinate system 6 selection

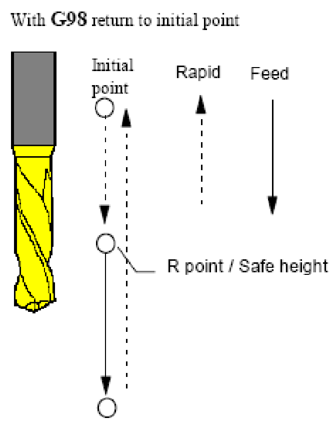


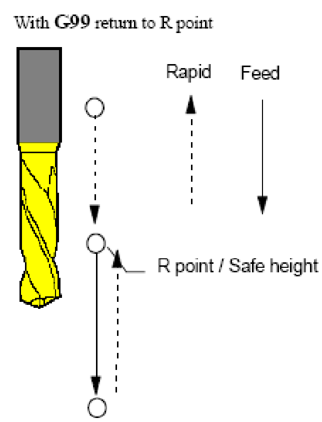
Canned cycles

Canned or fixed cycles are programming aids that simplify programming. Canned cycles combine many programming operations and are designed to shorten the program length, minimize mathematical calculations, and use minimal tool motions.

Examples: drilling, peck drilling, tapping, boring, back spot facing.

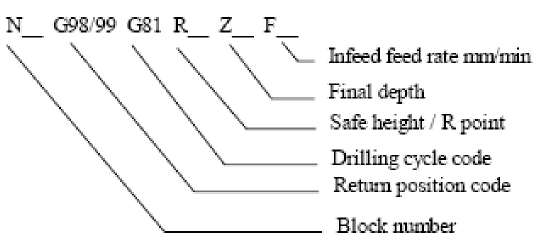
- **G81 Drilling cycle**
- **G82 Drilling cycle with dwell (Counter bore cycle)**
- **G83 Peck drilling cycle / deep drill**
- **G84 Right hand tapping cycle**
- **G85 Boring / Reaming cycle**
- **G86 Boring cycle**
- **G87 Back boring cycle**
- **G74 Left hand tapping cycle**
- **G76 Fine boring cycle**

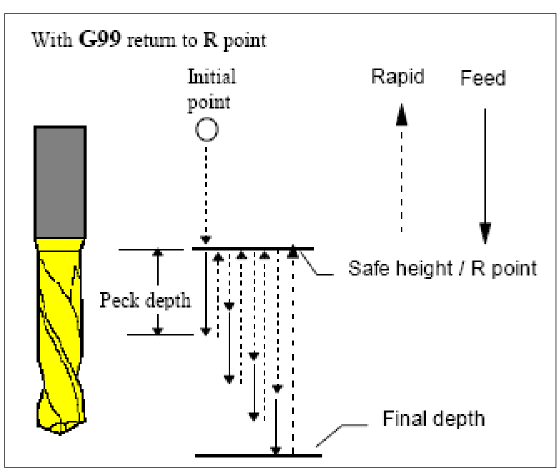




Drilling cycle (G81)

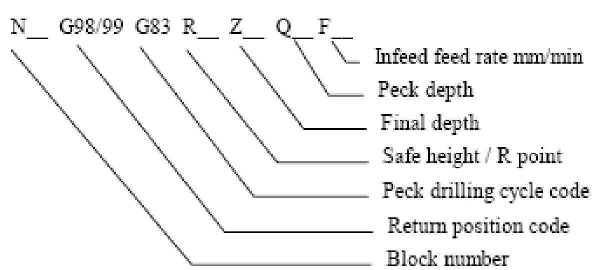
Format

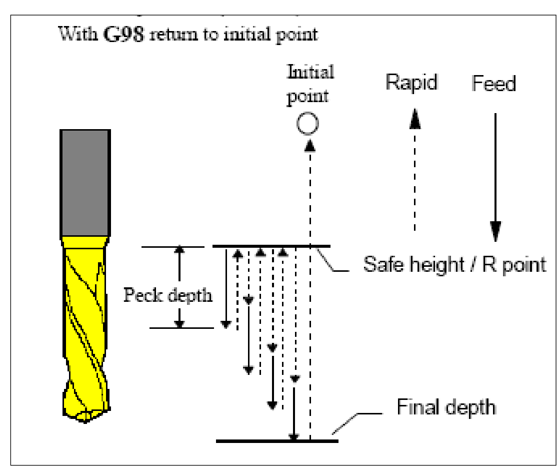




Peck drill cycle (G83)

Format





List of G-codes for Turning

G-code	Function
G00	Positioning rapid traverse
G01	Linear interpolation (feed)
G02	Circular interpolation CW
G03	Circular interpolation CCW
G04	Dwell
G20	Inch unit
G21	Metric unit
G28	Automatic zero return
G30	2nd reference point return
G32 / G33	Thread cutting (single motion)
G40	Tool nose radius compensation cancel
G41	Tool nose radius compensation left
G42	Tool nose radius compensation right
G50 / G92	Local co-ordinate system setting, max. spindle speed setting
G70	Finishing cycle
G71	Stock removal in turning (Multi pass Turning Cycle)
G72	Stock removal in facing
G73	Pattern repeating
G74	Peck drilling on Z axis / Face grooving
G75	Peck drilling on X axis / Int-Ext grooving
G76	Multiple threading cycle
G94 / G98	Feed per minute
G95 / G99	Feed per revolution
G96	Constant surface speed control
G97	Constant surface speed control cancel

List of M codes for Turning

M codes vary from machine to machine depending on the functions available on it and the manufacturer of the machine decides them. The M codes listed below are the common ones.

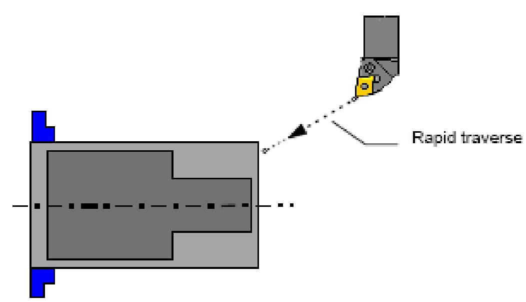
M-codes	Function	M-codes	Function
M00	Optional program stop automatic	M07	Mist coolant ON (coolant 1 ON)
M01	Optional program stop request	M08	Flood coolant ON (coolant 2 ON)
M02	Program end	M09	Coolant OFF
M03	Spindle ON clock wise (CW)	M19	Spindle orientation
M04	Spindle ON counter clock wise (CCW)	M30	End of program, Reset to start
M05	Spindle stop	M98	Sub program call
M06	Tool change	M99	Sub program end

G00 Rapid traverse

When the tool being positioned at a point preparatory to a cutting motion, to save time it is moved along a straight line at Rapid traverse, at a fixed traverse rate which is pre-programmed into the machine's control system. Typical rapid traverse rates are 10 to 25 m /min., but can be as high as 80 m/min.

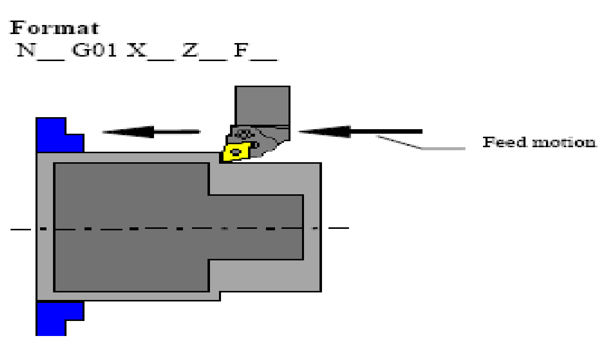
Format

N_ G00 X_ Z_

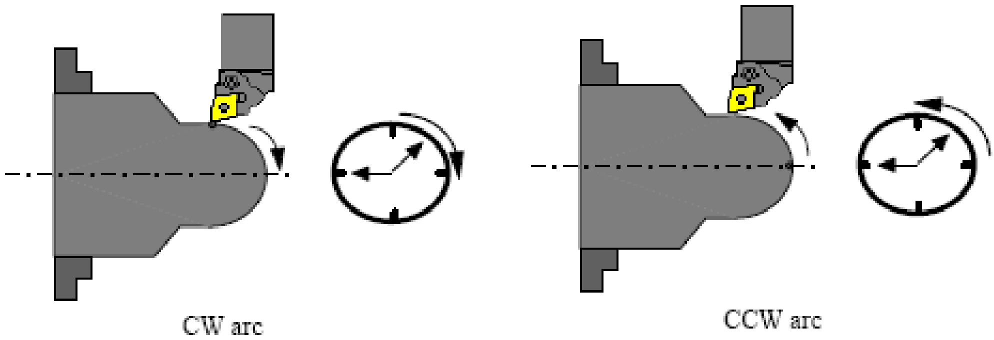


G01 Linear interpolation (feed traverse)

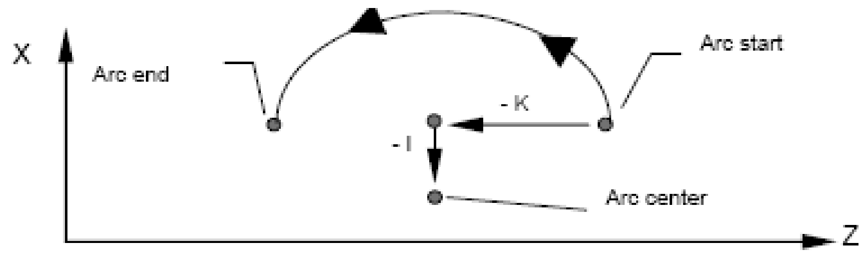
The tool moves along a straight line in one or two axis simultaneously at a programmed linear speed, the feed rate.



G02/03 Circular interpolation

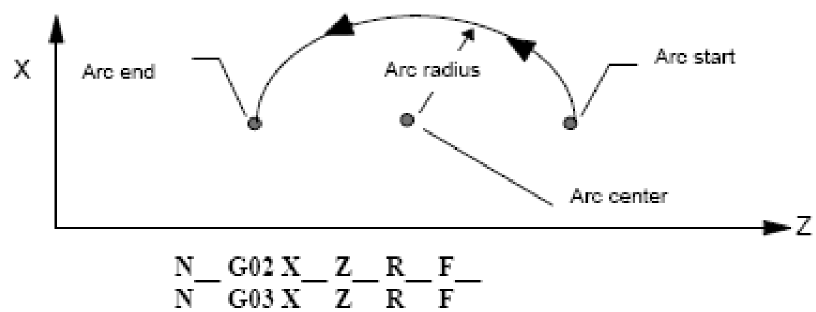


Format
N__ G02/03 X__ Z__ I__ K__ F__ using the arc center
Or
N__ G02/03 X__ Z__ R__ F__ using the arc radius
G02 moves along a CW arc
G03 moves along a CCW arc



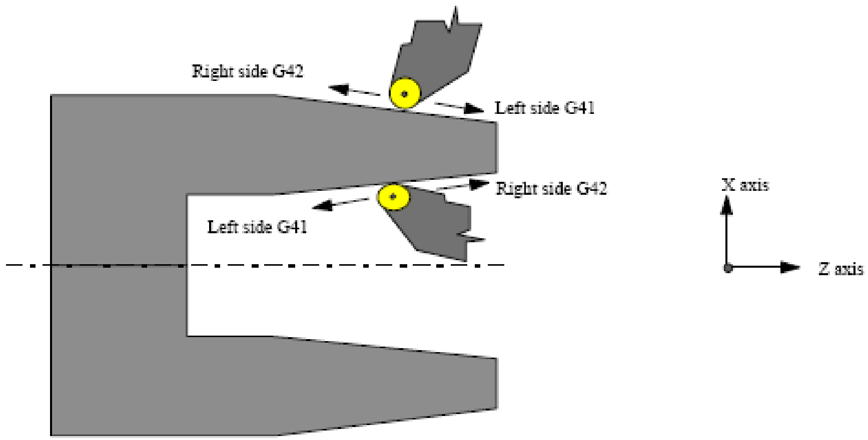
Arc center
The arc center is specified by addresses I and K. I and K are the X and Z co-ordinates of the arc center with reference to the arc start point.
I = (X co-ord. of center - X co-ord. of start point)/2
K = Z co-ord. of center - Z co-ord. of start point
I and K must be written with their signs.

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



Tool nose radius compensation

G40 Tool nose radius compensation Cancel
G41 Tool nose radius compensation Left
G42 Tool nose radius compensation Right



Canned cycles

A canned cycle simplifies a program by using a few blocks containing G-codes functions to specify the machining operations usually specified in several blocks.

Turning Cycle - G71

Format

> **G71 U (d) R (e)**
> **G71 P (n) Q(n) U(u) W(w) F(f) S(s) T(t)**
> **N (n) _ _ _ _**
> **_ _ _ _ _ _ _**
> **_ _ _ _ _ _ _**
> **N (n)_ _ _ _**
> **d = Depth of cut**
> **e = Retract amount**
> **n = Number of the first block of the shape**
> **n = Number of the last last block of the shape**
> **u = Finishing allowance in X**
> **w =Finishing allowance in Z**
> **f = Feed rate**
> **s = Spindle speed**
> **t = Tool number**

Multipass Turning Cycle - G73

Format

G73 U (i) W (k) R (d)
G73 P (n) Q (n) U (u) W (w) F (f) S(s) T (t)
N (n)_ _ _ _ _
_ _ _ _ _ _ _ _
_ _ _ _ _ _ _ _
N (n)_ _ _ _ _
i = Relief in the X axes direction
k = Relief in the Z axis direction
d = Number of cuts
n = Number of first block of the shape
n = Number of the last block of the shape
u = Finishing allowance in X
w = Finishing allowance in Z
f = Feed rate
s = Spindle speed ,
t = Tool number

Axial Drilling Cycle -G74
Format

G74R (e)
G74X (U)_ Z (W)_ P(i) Q (k) R(d) F(f)
e = Retract amount
X = X component of point B
U = Incremental amount from A to B
Z = Z component of point C
W = Increment amount from A to C
i = Movement amount in X direction
k = Depth of cut in Z direction
d = Relief Amount of the tool at the cutting bottom
f = Feed rate

Radial Grooving Cycle - G75

Format

G75R (e)

G75X (U)_ Z (W)_ P(i) Q (k) R(d) F(f)

e = Retract amount

X = X component of point B

U = Incremental amount from A to B

Z = Z component of point C

W = Increment amount from A to C

i = Movement amount in X direction

k = Depth of cut in Z direction

d = Relief Amount of the tool at the cutting bottom

f = Feed rate

Thread Cutting Cycle - G76

G76 P (m) (r) (a) Q(d min) R (d)

G76 X__ Z__ P__ Q__ F__

Format

m = No.of idle passes for finishing operation

r = Chamfering amount or pull out angle

a = Angle of thread, deg

d min = Minimum thread depth in microns

d = Finishing allowance in mm

X=Minor/major diameter

Z=Length of thread

P= Height of thread

Q = Depth of cut in first pass

F= Pitch of thread

G76 Threading Cycle Example

N5 G76 P010060 Q100 R0.05

N6 G76 X30 Z-20 P1024 Q200 F2

First block of the G76 Threading cycle

G76 : G code for threading cycle.

P : P actually consists of multiple values which control the thread behavior,

01: Number of spring passes or spring cuts.

00: Thread run out at 45 degree

60: Flank angle or Infeed angle

Q : Depth of normal cut (these values are given in hundreds, so the depth of cut will be 0.1).
R : Depth of Last or Finish cut

Second block of the G76 Threading cycle

G76 :G-code of the threading cycle.
X : The end value in x-axis.
Z : The end value in z-axis.
P : Thread depth (as radius value).
Q : Depth of first cut.
F : Thread Pitch

PROGRAMMING STRUCTURE

Main Programs are written using I.S.O. address codes listed below:
Addresses –

A FANUC compatible program number line is written in the following format:OØØØØ
Where, O is the Address Code. ØØØØ is the four digit program number

N	**refers to the block number.**
G	**refers to the G code (Preparatory function).**
X	**refers to the absolute/incremental distance travelled by the slide tool in the X axis direction.**
Y	**refers to the absolute/incremental distance travelled by the slide tool in the Y axis direction.**
Z	**refers to the absolute/incremental distance travelled by the slide tool in the Z axis direction.**
F	**refers to the feed rate.**
M	**refers to the M code (Miscellaneous function).**
S	**refers to the spindle speed.**
T	**refers to the tooling management.**

CNC MILLING CENTER AT GAT

TECHNICAL SPECIFICATIONS OF CNC MILLING CENTER

SPARK

DRILL TAP MACHINING CENTERS

AMS®

In Pursuit of Excellence

Technical Specifications

Capacity	Unit	SPARK
Table longitudinal travel (X axis)	mm	300
Table cross travel (Y axis)	mm	250
Headstock travel (Z axis)	mm	250
Spindle nose face to table top	mm	150 - 400
Table height from floor	mm	840
Table		
Table size	mm x mm	500 x 250
T-slot (No.xsizexpitch)		3 x 14 x 80
Max. load on table	kgf	150
Spindle & axes		
Spindle taper		7 / 24 No.30
Spindle speed - std.	rpm	60 - 6000
Spindle speed - opt.	rpm	80 - 8000
Spindle power	kW	5.5 / 3.7
Rapindtraaverse X/Y/Z	m /min	25 / 25 / 18
Guide ways type		LM
Accuracy - As per ISO 230-2		
Positioning accuracy	mm	0.010
Repeatability	mm	± 0.003
Automatic Tool Changer		
Tool change system		Disc Armless

*All specifications are subject to change without prior notice

CNC TURNING CENTER AT GAT

Built suited for educational institutions for training needs

Machine Features

- Cost effective CNC turning machine
- Rigid & true slant bed for easy chip disposal
- Precision LM guideways on both the axes
- Cartridge type high speed spindle
- Bi-directional tool turret (8 station)
- Tailstock with hydraulic quill actuation
- Rapid rate of 15 m/min on both the axes
- Compact foot print
- Maintenance friendly

Standard Accessories

- Automatic centralized oil lubrication
- Hydraulic power operated chuck
- Basic hydraulic system
- OD tool holder clamping block (8 nos)
- Boring bar holder (4 nos)
- Facing tool holder (2 nos)
- One set of sleeves

Optional Accessories

- Programmable quill
- 3 tier lamp
- Voltage stabilizer - 15 kVA
- Chip conveyor rear
- TPM - Trak productivity monitoring system

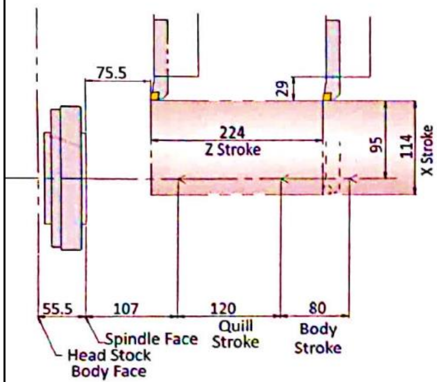
Fast on Site: All India Post Sales Service

Ace Micromatic Group

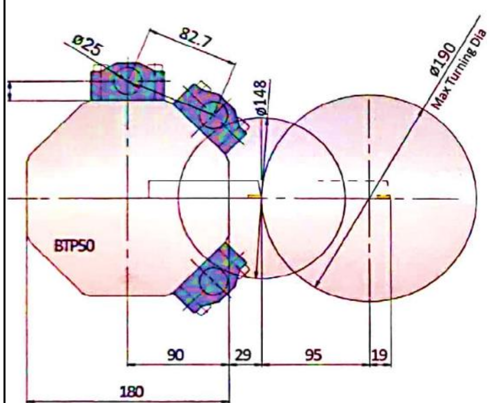
THE LARGEST MACHINE TOOL GROUP IN INDIA

TECHNICAL SPECIFICATIONS OF TURNING CENTER

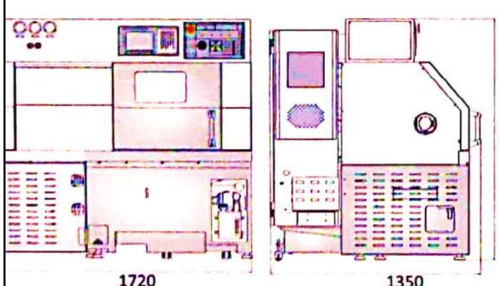
Machining Range



Interference Diagram



Overall View



Manufactured by:

All Dimensions are in mm

Specifications

Specifications		Tutor
CONTROL SYSTEM	Units	Fanuc / Siemens
CAPACITY		
Swing Over Bed	mm	425
Distance Between Centers[1]	mm	250
Maximum Turning Dia	mm	190
Between Centre Turning Dia	mm	165
Maximum Turning Length[2]	mm	224
SPINDLE		
Spindle Nose		Flat 110
Bore Through Spindle	mm	36
Front Bearing Bore	mm	60
Maximum Spindle speed[3]	rpm	4000
Spindle Motor Power (F/S) 15 min / S6-40% Rating	kw	5.5
Spindle Motor Power (Continuous Rating)	kw	3.7
Full Power Range (F/S)	rpm	1333-2333 / 1000-4000
Standard Chuck Size	mm	135
AXES		
Type of Guideways		Linear motion Guideways
X-Axis Stroke	mm	114
Z-Axis Stroke	mm	224
X & Z Axes Rapid Rate	m/min.	15
X Axis Ball Screw Dia And Pitch	mm	25 & 5
Z Axis Ball Screw Dia And Pitch	mm	25 & 5
X & Z Axes Motor Torque	Nm	2.6
TOOLING		
Number of Tools Maximum		8
Boring Bar Dia Maximum	mm	25
OD Turning Tool Size	mm	16 X 16
TAILSTOCK		
Tailstock Base Travel	mm	80
Tailstock Quill Travel	mm	120
Tailstock Quill Dia.	mm	50
Quill Taper		MT-4
Tailstock rotating centre type		Add-on
Tailstock Thrust (Max)(@ 15 Bar)	kgf	150
Coolant Tank Capacity	litre	60
HYDRAULIC SYSTEM		
Hydraulic Pump Capacity	lpm	12
Hydraulic Power Pack Tank Capacity	litre	20
System Pressure	Kg/cm²	30
Overall Machine Dimensions (LxWxH)	mm	1720 X 1350 X 1700
Overall Weight (Without Packing)	kg	~1850

Specifications correct at the time of print. Owing to continuous up-gradation of our products, the specification may change and to be reconfirmed at the time of ordering.

1 Refer machining range diagram   2 Refer Interference Diagram

3 With Appropriate Work holding.   (F/S) - Fanuc / Siemens

Note: Features shown may not be part of the standard equipment.

DESIGNERS

ACE DESIGNERS LTD.

Plant I - Plot No.7 & 8, II Phase, Peenya Industrial Area, Bangalore - 560 058, India

Tel : +91-80-2218 6700, Fax : +91-80-2218 6723, E-mail : info@acedesigners.co.in

MACHINING DATA FOR TURNING

Machining data for Turning					
Cutting conditions.					
Work material	Tool material	Cutting speed,m/min			
		Depth, mm			
		5 to 10	2 to 5	0.5 to 2	0.1 to 0.5
		Feed mm/rev			
		0.4-0.6	0.25-0.5	0.2-0.3	0.05-0.2
Free-machining steels	HSS	20-40	40-70	40-110	50-120
	Carbide	90-150	120-180	150-250	200-500
Mild steels	HSS	25-35	30-50	30-60	40-80
	Carbide	60-120	80-150	120-200	150-450
Medium carbon steels	HSS	15-25	25-45	25-50	30-70
	Carbide	50-110	60-120	90-150	120-300
Alloy steels	HSS	10-15	15-25	15-35	20-45
	Carbide	30-65	40-80	60-100	80-180
Tool steels	HSS	15-20	20-25	20-30	30-60
	Carbide	50-110	60-120	90-150	120-300
Stainless steels	HSS	15-20	15-25	15-30	20-50
	Carbide	40-60	40-70	50-80	50-90
Cast iron: Grey, Ductile Malleable	HSS	20-25	25-30	35-45	40-60
	Carbide	60-90	70-100	90-120	100-200
Aluminium alloys	HSS	40-70	70-100	90-120	100-200
	Carbide	60-150	80-180	90-450	150-600
Copper alloys	HSS	40-60	60-100	90-120	100-200
	Carbide	50-110	60-150	90-180	120-310
Magnesium alloys	HSS	40-70	70-100	90-120	100-200
	Carbide	60-150	80-180	90-450	150-600
Titanium alloys	HSS	10-15	15-30	30-50	50-90
	Carbide	15-30	30-50	50-90	60-120

MACHINING DATA FOR MILLING

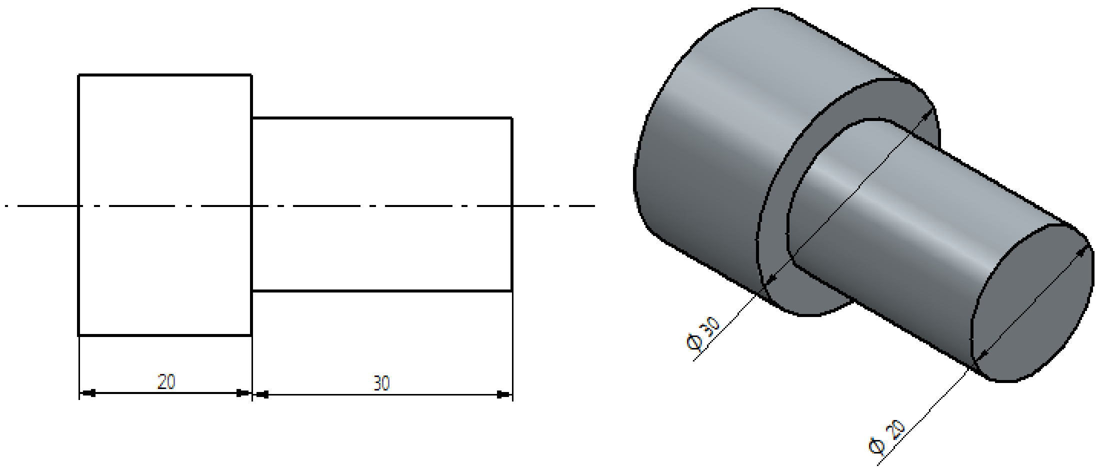
Machining data conditions for Milling							
Work material	**Tool material**	**Speed, m/min**	**Feed, mm per tooth**				
			Face mills	**Slab mills**	**Slotting & side mills**	**End mills**	**Form cutters**
Free-machining steels	**HSS**	**30-40**	**0.3**	**0.25**	**0.175**	**0.15**	**0.1**
	Carbide	**100-200**					
Mild steels	**HSS**	**25-40**	**0.25**	**0.2**	**0.15**	**0.125**	**0.1**
	Carbide	**90-130**					
Medium carbon steels	**HSS**	**20-30**	**0.2**	**0.15**	**0.125**	**0.1**	**0.075**
	Carbide	**60-90**					
Alloy steels	**HSS**	**10-20**	**0.15**	**0.1**	**0.075**	**0.06**	**0.05**
	Carbide	**40-55**					
Tool steels	**HSS**	**15-25**	**0.2**	**0.15**	**0.1**	**0.075**	**0.05**
	Carbide	**60-80**					
Stainless steels	**HSS**	**15-20**	**0.15**	**0.1**	**0.1**	**0.075**	**0.05**
	Carbide	**30-60**					
Cast iron: Grey, Ductile, Malleable	**HSS**	**20-30**	**0.35**	**0.3**	**0.2**	**0.175**	**0.1**
	Carbide	**70-100**					
Aluminium alloys	**HSS**	**60-100**	**0.5**	**0.4**	**0.3**	**0.25**	**0.175**
	Carbide	**60-180**					
Copper alloys	**HSS**	**40-75**	**0.3**	**0.25**	**0.2**	**0.175**	**0.15**
	Carbide	**60-100**					
Magnesium alloys	**HSS**	**60-100**	**0.5**	**0.4**	**0.25**	**0.2**	**0.175**
	Carbide	**60-180**					
Titanium alloys	**HSS**	**10-30**	**0.15**	**0.125**	**0.1**	**0.075**	**0.05**
	Carbide	30-50					

PROCEDURE FOR WRITING THE PROGRAM IN RECORD

Step Turning using G90 and G71 Cycles

Aim: To develop a Part program for the given component and to simulate using SeeNC Turn Software

Diagram:



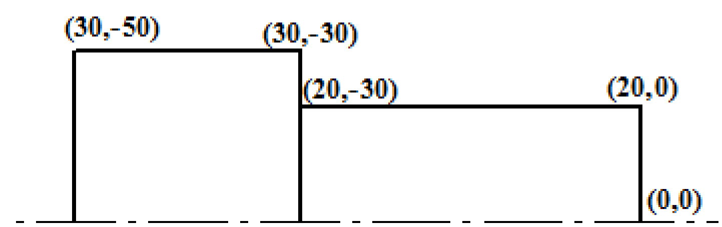


Fig: Absolute Dimensioning

All dimensions are in mm

Route Sheet				
Prog. No: 01	**Blank Size:** Ø 30 mm X 50 mm		**Programmer :**ME , GAT	
Dwg. No.: 01	**Material :**Mild Steel		**Approved by : _______________;** **Date:_________**	
Sl.No	**Operation**	**Tool type**	**Speed**	**Feed**
1	Step turning	Single Point Cutting tool PDJNL 2020 K 11 R 0.4	1200 RPM	30 mm/min

Part program

BOX TURNING CYCLE [G90]	
Part Programming	**Description**
N010 G21 G98;	**Program is set with Metric units and in Absolute mode**
N020 G28 U0 W0;	**To start the program from the reference position**
N030 M06 T0101;	**Select Single point cutting tool**
N040 M03 S1200;	**Start spindle at 1200 rpm.**
N050 G00 X30 Z2;	**Positions the tool at the location (30,2)**
N060 G90 X30 Z-30 F30;	Initiates box turning cycle up to 30 mm length.
N070 X28;	Reduction in diameter by 2 mm
N080 X26;	Reduction in diameter by 2 mm
N090 X24;	Reduction in diameter by 2 mm
N100 X22;	Reduction in diameter by 2 mm
N110 X20;	Reduction in diameter by 2 mm
N120 G28 U0 W0;	Go to home position
N130 M05;	Stop the spindle
N140 M30;	Stop the program.
TURNING CYCLE [G71]	
Part Programming	**Description**
N010 G21 G98;	Program is set with Metric units and in Absolute mode
N020 G28 U0 W0;	To start the program from the reference position
N030 M06 T0101;	Select Single point cutting tool
N040 M03 S1200;	Start spindle at 1200 rpm.
N050 G00 X30 Z2;	Positions the tool at the location (30,2)
N060 G71 U0.5 R1;	Depth of cut as 0.5 mm and retardation as 1 mm
N070 G71 P080 Q120 U0 W0 F30;	Turning cycle to follow the path mentioned with in P & Q
N080 G01 X0 Z0;	Path - 5
N090 G01 X20 Z0;	Path – 4
N100 G01 X20 Z-30;	Path – 3
N110 G01 X30 Z-30;	Path – 2
N120 G01 X30 Z-50;	Path – 1
N130 G28 U0 W0;	Go to home position
N140 M05;	Stop the spindle
N150 M30;	Stop the program.

List of G – Codes and M – Codes used

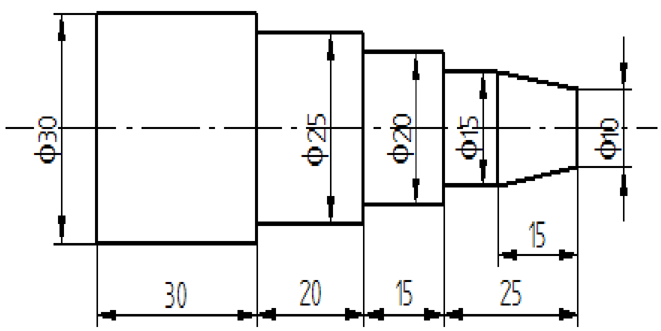
G - Codes	Description	M - Codes	Description
G00	Rapid traverse	M03	Spindle Rotation – CW
G21	Metric units	M05	Spindle Stop
G28	Reference position	M06	Tool Selection
G71	Multi Pass turning cycle (Canned cycle)	M30	Program End
G90	Box Turning Cycle		
G98	Absolute Programming Mode		

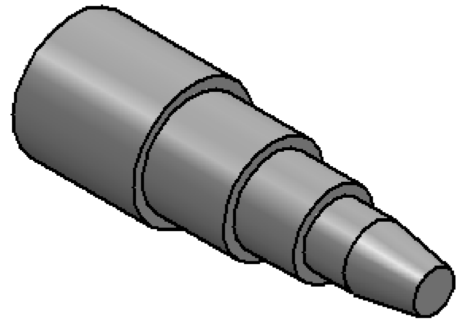
Result:

Part program using word address format was developed and simulated using SeeNC Turn software.

SEENC – TURN
PROGRAM 1

Write a program to machine the geometry for the following sketch using SeeNC-Turn. [Facing,





```
O001 (Program using G90-Box turning cycle)
N010 G21 G98
N020 G28 U0 W0
N030 M06 T0101
N040 M03 S1200
N050 G00 X30 Z1
N060 G90 X30 Z-60 F30
N070 X28
N080 X26
N090 X25
N100 G00 X25 Z1
N110 G90 X25 Z-40 F30
N120 X23
N130 X21N130 X20 Z-40
N140 X20N140 X25 Z-40
N150 G00 X20 Z1N150 X25 Z-60
N160 G90 X20 Z-25 F30N160 X30 Z-60
N170 X18
N180 X16
N190 X15
N210 G90 X15 Z-15 R0 F30
N220 R-1
N230 R-2
N240 R-2.5
N250 G28 U0 W0
N260 M05
N270 M30
```

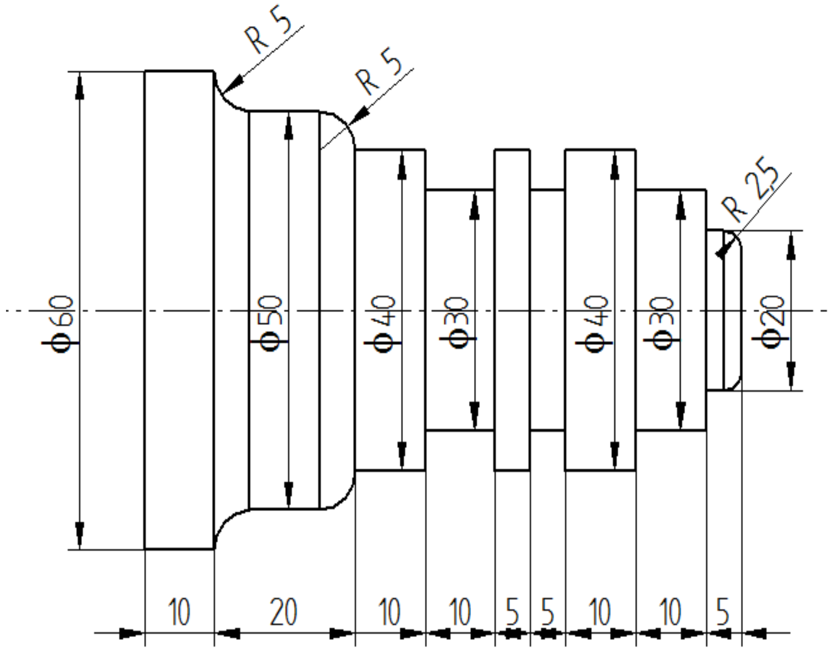
Problem 1 (Program using G71 cycle)

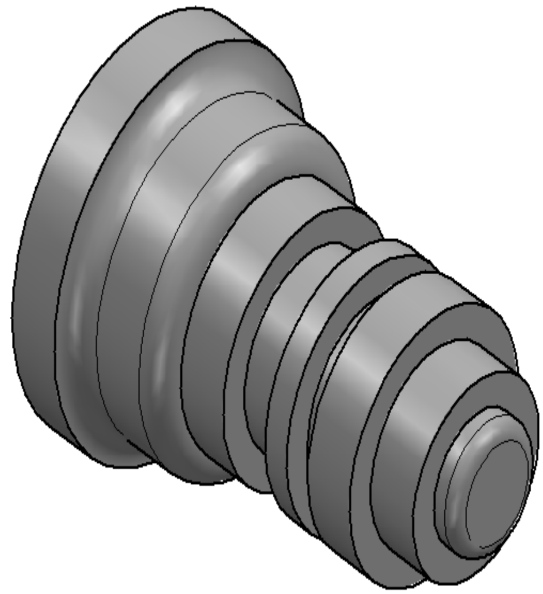
```
O001
N010 G21 G98
N020 G28 U0 W0
N030 M06 T0101
N040 M03 S1200
N050 G00 X30 Z1
N060 G71 U0.5 R1
N070 G71 P080 Q160 U0.1 W0.1 F30
N080 G01 X0 Z0
NO90 X10 Z0
N100 X15 Z-15
N110 X15 Z-25
N120 X20 Z-25
N130 X20 Z-40
N140 X25 Z-40
N150 X25 Z-60
N160 X30 Z-60
N170 G70 P080 Q160
N180 G28 U0 W0
N190 M05
N200 M30
```

Date of conduction		**Faculty**		**Reviewed by**	

PROGRAM 2

Write a program to machine the geometry for the following sketch using SeeNC-Turn.



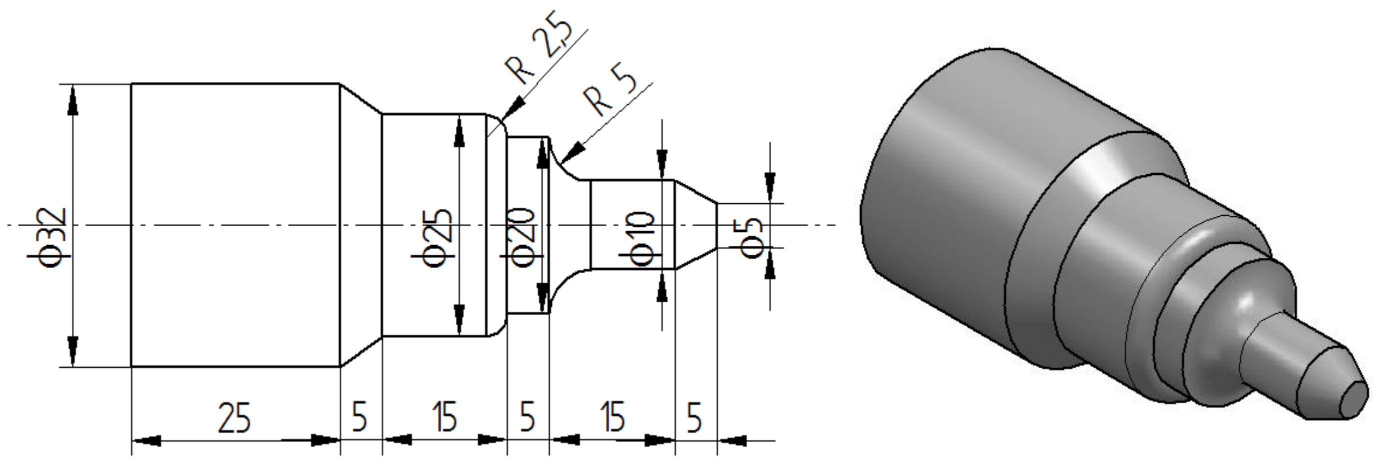


```
O002
N010 G21 G98
N020 G28 U0 W0
N030 M06 T0101
N040 M03 S1200
N050 G00 X60 Z1
N060 G71 U0.5 R1
N070 G71 P080 Q170 U0 W0 F30
N080 G01 X0 Z0
N090 G01 X15 Z0
N100 G03 X20 Z-5 R5 F30
N101 G01 X30 Z-5
N110 G01 X30 Z-15
N120 X40 Z-15
N130 X40 Z-55
N140 G03 X50 Z-60 R5 F30
N150 G01 X50 Z-70
N160 G02 X60 Z-75 R5 F30
N170 G01 X60 Z-85
N180 M06 T0202
N190 G00 X40 Z-28
N200 G75 R1
N210 G75 X30 Z-30 P0500 Q1000 R250 F30
N220 G28 U0 W0
N230 G00 X40 Z-38
N240 G75 R1
N250 G75 X30 Z-45 P0500 Q1000 R0250 F30
N260 G28 U0 W0
N270 M05
N280 M30
```

Date of conduction		**Faculty**		**Reviewed by**	

PROGRAM 3

Write a program to machine the geometry for the following sketch using SeeNC-Turn.

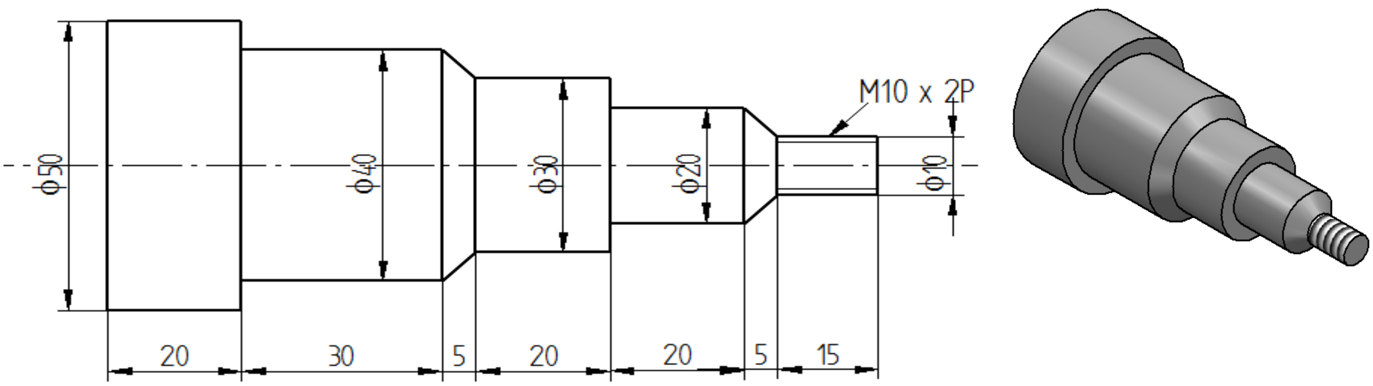



```
O003
N010 G21 G98
N020 G28 U0 W0
N030 M06 T0101
N040 M03 S1200
N050 G00 X32 Z2
N060 G71 U1 R1
N070 G71 P80 Q150 U0.1 W0.1 F50
N080 G01 X0 Z0
N090 G01 X5 Z0
N100 G01 X10 Z-5
N110 X10 Z-15
N120 G02 X20 Z-20 R5
N130 G01 X20 Z-25
N140 G03 X25 Z-30 R5
N150 G00 X25 Z-40
N155 G01 X25 Z-40
N160 X32 Z-45 F30
N165 X32 Z2
N170 G70 P080 Q150
N180 M05
N190 M30
```

Date of conduction		**Faculty**		**Reviewed by**	

Program 4

Write a program to machine the geometry for the following sketch using SeeNC-Turn.

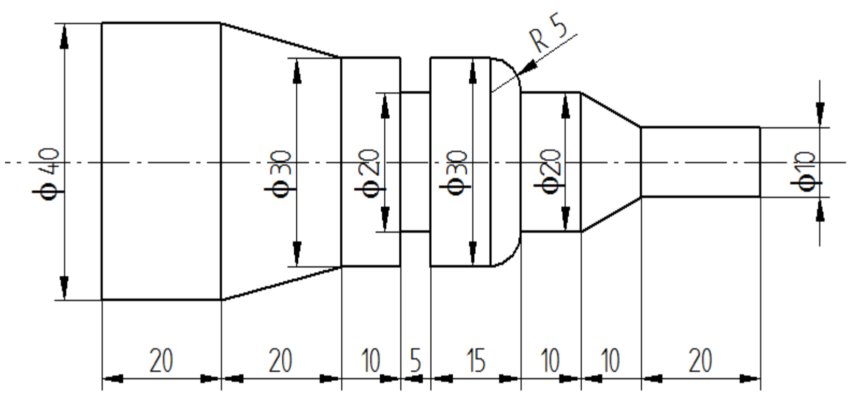


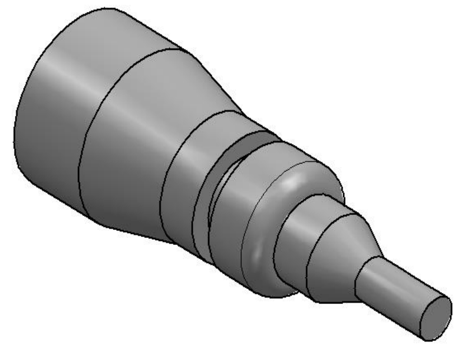

```
O004
N010 G21 G98
N020 G28 U0 W0
N030 M06 T0101
N040 M03 S1200
N050 G00 X51 Z2
N060 G71 U1 R1
N070 G71 P80 Q170 U0.1 W0.1 F50
N080 G01 X0 Z0 F 50
N090 G01 X10 Z0
N100 G01 X10 Z-15
N110 G01 X20 Z-25 F50
N120 G01 X20 Z-45
N130 G01 X30 Z-45
N140 G01 X30 Z-65
N150 G01 X40 Z-75 F30
N160 G01 X40 Z-105
N170 G01 X50 Z-105
N180 G28 U0 W0
N190 M06 T0202
N200 M03 S800
N210 G00 X10 Z0
N220 G76 P031560 Q100 R0.15
N230 G76 X7.54 Z-10 P1226 Q125 F1
N240 G28 U0 W0
N250 M05
N260 M30
```

Date of conduction		Faculty		Reviewed by	

PROGRAM 5

Write a program to machine the geometry for the following sketch using SeeNC-Turn



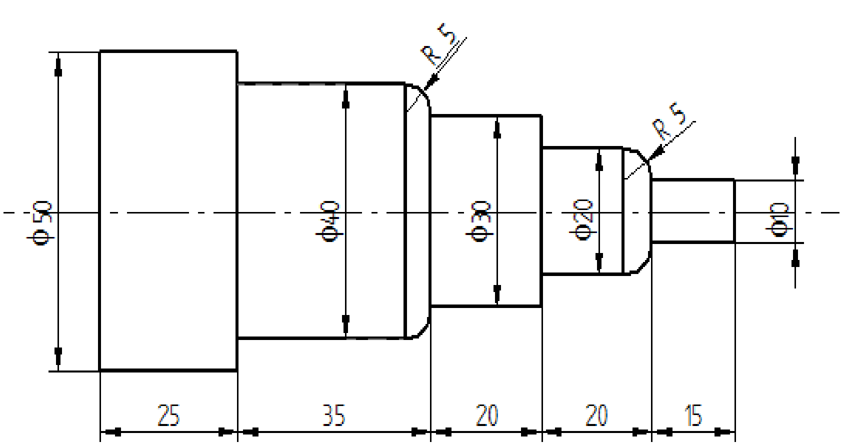


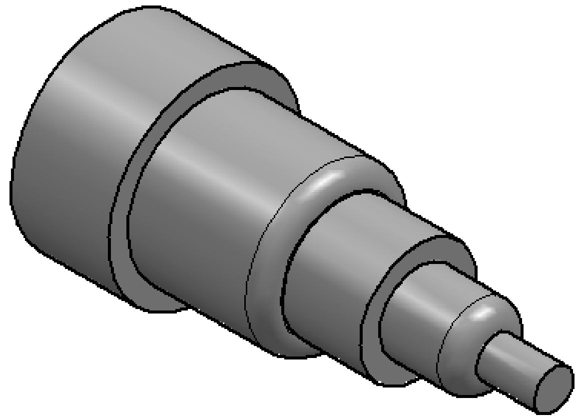
```
O005
N010 G21 G98
N020 G28 U0 W0
N030 M06 T0101
N040 M03 S1200
N050 G00 X40 Z2
N060 G71 U0.5 R1
N070 G71 P80 Q150 U0 W0 F50
N080 G01 X0 Z0
N090 G01 X10 Z0
N100 G01 X10 Z-20
N110 G01 X20 Z-30
N120 G01 X20 Z-40
N130 G03 X30 Z-45 R5
N140 G01 X30 Z-70
N150 G01 X40 Z-90
N160 G28 U0 W0
N170 M06 T0202
N180 M03 S500
N190 G00 X31 Z-58
N200 G75 R1
N210 G75 X20 Z-60 P0500 Q1000 R0250 F100
N220 G28 U0 W0
N230 M05
N240 M30
```

Date of conduction		Faculty		Reviewed by	

PROGRAM 6

Write a program to machine the geometry for the following sketch using SeeNC-Turn.



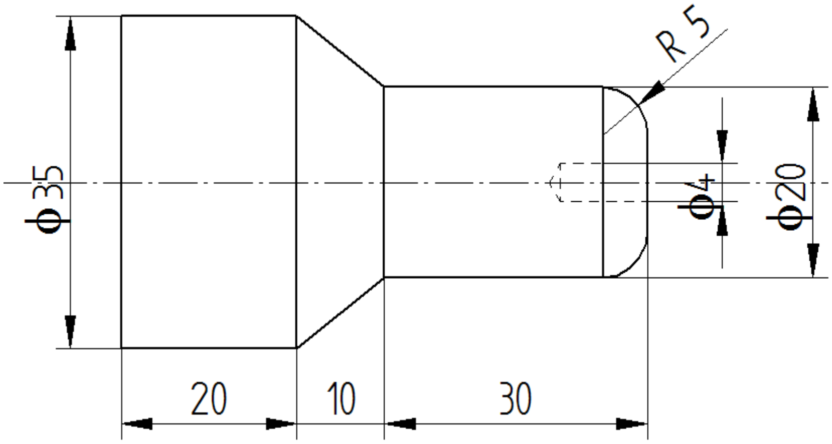


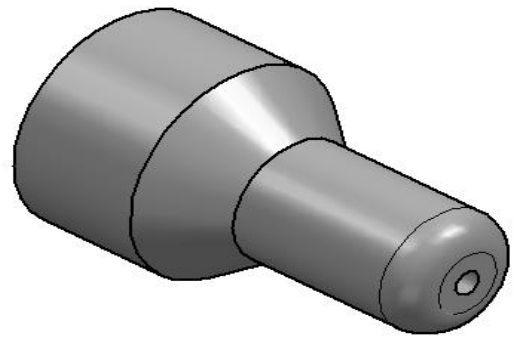
```
O007
N010 G21 G98
N020 G28 U0 W0
N030 M06 T0101
N040 M03 S1200
N050 G00 X50 Z2
N060 G71 U0.5 R1
N070 G71 P080 Q160 U.1 W.1 F50
N080 G01 X0 Z0
N090 X10 Z0
N100 X10 Z-15
N110 G03 X20 Z-20 R05
N111 G01 X20 Z-35
N112 G01 X30 Z-35
N115 G01 X30 Z-55
N120 G01 X30 Z-55
N140 G03 X40 Z-60 R5
N141 G01 X40 Z-90
N160 G01 X50 Z-90
N165 G01 X50 Z-115
N170 G28 U0 W0
N180 M05
N180 M30
```

Date of conduction		**Faculty**		**Reviewed by**	

PROGRAM 7

Write a program to machine the geometry for the following sketch using SeeNC-Turn.





O008
N010 G21 G98
N020 G28 U0 W0
N030 M06 T0101
N040 M03 S1200
N050 G00 X36 Z1
N060 G71 U0.5 R1
N070 G71 P080 Q130 U0.1 W0.1 F30
N080 G01 X0 Z0
NO90 X15 Z0
N100 G03 X20 Z-5 R5
N110 G01 X20 Z-30
N120 X35 Z-40
N130 X35 Z-60
N140 G70 P080 Q160
N150 G28 U0 W0
N160 M06 T0202
N170 G74 R1
N180 G74 X0 Z-10 Q500 R1 F30
N190 G00 X35 Z0
N200 M05
N210 M30

Date of conduction		**Faculty**		**Reviewed by**	

EXERCISE PROGRAM 8

Write a program to machine the geometry for the following sketch using SeeNC-Turn. [Facing, Step Turning, Circular interpolation & Thread cutting cycles]