SYLLABUS

KARPAGAM ACADEMY OF HIGHER EDUCATION

COIMBATORE – 21

FACULTY OF ENGINEERING

DEPARTMENT OF MECHANICAL ENGINEERING

16BEME402 MANUFACTURING TECHNOLOGY II 3 0 0 3 100

OBJECTIVES

1. To understand the concept and basic mechanics of metal cutting, working of standard machine tools such as lathe, shaping and allied machines, milling, drilling and allied machines, grinding and allied machines and broaching.
2. To understand the basic concepts of Computer Numerical Control (CNC) of machine tools and CNC Programming

UNIT I THEORY OF METAL CUTTING AND CUTTING TOOLS 9

Introduction: material removal processes, types of machine tools – theory of metal cutting: chip formation, orthogonal cutting, oblique cutting – Cutting tool materials, tool wear, tool life, surface finish, cutting fluids, heat generation, Merchant circle.

UNIT II CENTRE LATHE AND SEMI AUTOMATIC LATHES 9

Centre lathe– constructional features, various operations, taper turning methods, thread cutting methods, special ttachments, machining time and power estimation. Capstan and turret lathes – automats – single spindle, Swiss type, automatic screw type, multi spindle – Tool layout for Capstan, Turret and Automats.

UNIT III RECIPROCATING MACHINE TOOLS & MILLING MACHINES 9

Shaper – construction, working, work and tool holding device, quick return mechanism, planer – construction, working, mechanism, operations. Slotter – construction, working.

Milling machine – constructions, types, Indexing mechanism, operations, milling cutter, gear hobbing – principle.

UNIT IV OTHER MACHINE TOOLS 9

Drilling – types, radial drilling machine, construction, operations, Boring, types, Jig boring machine – construction, operations, Broaching – types, construction, Grinding – grinding wheel, specifications and selection, cylindrical grinding, surface grinding, centreless grinding – honing, lapping, super finishing, polishing and buffing.

UNIT V CNC MACHINES 9

CNC Machines – Construction – Types of control systems, Manual Part Programming – Computer assisted part programming – Computer aided part programming, Machining centers – principle, Turning centers – principle, CAD/CAM & Integration, Application of CNC Machines

TEXT BOOKS

S. No	Author(s) Name	Title of the book	Publisher	Year of Publication
1	Hajra Choudhury	Elements of Workshop Technology Vol– II	Media Promotors Pvt	2010
2	HMT	Production Technology	Tata McGraw–Hill	2008

REFERENCE

S. No	Author(s) Name	Title of the book	Publisher	Year of Publication
1	P.C. Sharma	A text book of production	S. Chand and Co. Ltd	2014
2	Shrawat N.S. and Narang J.S	CNC Machines	Dhanpat Rai and Co	2002
3.	P.N.Rao	CAD/CAM Principles and Applications’	TATA Mc Craw Hill	2012
4	Milton C.Shaw	Metal Cutting Principles Second Edition	Oxford University Press	2005

WEB REFERENCES

1. www.steelonline.co.in
2. http://mmu.ic.polyu.edu.hk
3. www.waterjetindiana.com
4. www.teskolaser.com
5. www.cncinformation.com
6. www.cncmachineprogramming.net

KARPAGAM ACADEMY OF HIGHER EDUCATION

COIMBATORE – 21

FACULTY OF ENGINEERING

DEPARTMENT OF MECHANICAL ENGINEERING

COURSE PLAN

Subject Name	**: MANUFACTURING TECHNOLOGY II**
Subject Code	**: 16BEME402 (Credits - 3)**
Name of the Faculty	**: P. M. GOPAL**
Designation	**: ASSISTANT PROFESSOR**
Year/Semester/Section	**: II/ III 'A'**
Branch	**: MECHANICAL**

Sl. No.	No. of Periods	Topics to be Covered	Support Materials
		UNIT – I : THEORY OF METAL CUTTING AND CUTTING TOOLS	
1.	1	Introduction: material removal processes	**T(1), R(1)**
2.	1	Types of machine tools	**T(1), R(1)**
3.	1	Theory of metal cutting: chip formation,	**T(1), R(1)**
4.	1	orthogonal cutting, oblique cutting –	**T(1), R(1)**
5.	1	Cutting tool materials,	**T(1), R(1)**
6.	1	Tool wear, tool life,	**T(1), R(1)**
7.	1	Surface finish, cutting fluids,	**T(1), R(1)**
8.	1	Heat generation	**T(1), R(1)**
9.	1	Merchant circle.	**T(1), R(1)**
10.	1	**Discussion on Competitive Examination related Questions / University previous year questions**	**Question Bank**
		Total No. of Hours Planned for Unit - I	**10**

Sl. No.	No. of Periods	Topics to be Covered	Support Materials
		UNIT – II : CENTRE LATHE AND SEMI AUTOMATIC LATHES	
11	1	Centre lathe– constructional features,	**T(1), R(1)**
12	1	Various operations, taper turning methods,	**T(1), R(1)**
13	1	Thread cutting methods, special attachments,	**T(1), R(1)**
14	1	Machining time and power estimation.	**T(1), R(1)**
15	1	Capstan and turret lathes – automats	**T(1), R(1)**
16	1	Single spindle, Swiss type,	**T(1), R(1)**
17	1	Automatic screw type, multi spindle	**T(1), R(1)**
18	1	Tool layout for Capstan	**T(1), R(1)**
19	1	Turret and Automats	**T(1), R(1)**

20	1	**Discussion on Competitive Examination related Questions / University previous year questions**	**Question Bank**
Total No. of Hours Planned for Unit - II			**10**

Sl. No.	No. of Periods	Topics to be Covered	Support Materials
UNIT – III : RECIPROCATING MACHINE TOOLS & MILLING MACHINES			
21	1	Shaper – construction, working,	**T(1), R(1)**
22	1	Work and tool holding device,	**T(1), R(1)**
23	1	Quick return mechanism,	**T(1), R(1)**
24	1	Planer – construction, working, mechanism, operations.	**T(1), R(1)**
25	1	Slotter – construction, working.	**T(1), R(1)**
26	1	Milling machine – constructions,	**T(1), R(1)**
27	1	Types, Indexing mechanism,	**T(1), R(1)**
28	1	Operations, milling cutter,	**T(1), R(1)**
29	1	Gear hobbing – principle.	**T(1), R(1)**
30	1	**Discussion on Competitive Examination related Questions / University previous year questions**	**Question Bank**
Total No. of Hours Planned for Unit - III			**10**

Sl. No.	No. of Periods	Topics to be Covered	Support Materials
UNIT – IV : OTHER MACHINE TOOLS			
31	1	Drilling – types, radial drilling machine,	**T(1), R(1)**
32	1	Construction, operations	**T(1), R(1)**
33	1	Boring, types, Jig boring machine – construction, operations,	**T(1), R(1)**
34	1	Broaching – types, construction,	**T(1), R(1)**
35	1	Grinding – grinding wheel,	**T(1), R(1)**
36	1	Specifications and selection,	**T(1), R(1)**
37	1	Cylindrical grinding, surface grinding,	**T(1), R(1)**
38	1	Centreless grinding – honing, lapping,	**T(1), R(1)**
39	1	Super finishing, polishing and buffing.	**T(1), R(1)**
40	1	**Discussion on Competitive Examination related Questions / University previous year questions**	**Question Bank**
Total No. of Hours Planned for Unit - IV			**10**

Sl. No.	No. of Periods	Topics to be Covered	Support Materials
UNIT – V : CNC MACHINES			

41	1	CNC Machines – Construction	**T(1), R(1)**
42	1	Types of control systems,	**T(1), R(1)**
43	1	Manual Part Programming	**T(1), R(1)**
44	1	Computer assisted part programming	**T(1), R(1)**
45	1	Computer aided part programming,	**T(1), R(1)**
46	1	Machining centers – principle,	**T(1), R(1)**
47	1	Turning centers – principle,	**T(1), R(1)**
48	1	CAD/CAM & Integration,	**T(1), R(1)**
49	1	Application of CNC Machines	**T(1), R(1)**
50	1	**Discussion on Competitive Examination related Questions / University previous year questions**	**Question Bank**
Total No. of Hours Planned for Unit - V			**10**

TOTAL PERIODS : 50

TEXT BOOKS

T [1] – Hajra Choudhury, Elements of Workshop Technology Vol– II, Media Promotors Pvt Ltd., Mumbai, 2010

T [2] – HMT, Production Technology, Tata McGraw–Hill, 2008

REFERENCES

R [1] - P.C. Sharma, A text book of production technology, S. Chand and Co. Ltd, 2014

R [2] - Shrawat N.S. and Narang J.S, CNC Machines, Dhanpat Rai and Co, 2002

R [3] - P.N.Rao, CAD/CAM Principles and Applications', TATA Mc Craw Hill, 2012

R [4] - Milton C.Shaw, Metal Cutting Principles Second Edition, Oxford University Press, 2005

WEBSITES

W [1] - www.steelonline.co.in

W [2] - http://mmu.ic.polyu.edu.hk

W [3] - www.waterjetindiana.com

W [4] - www.teskolaser.com

W [5] - www.cncinformation.com

W [6] - www.cncmachineprogramming.net

UNIT	Total No. of Periods Planned	Lecture Periods	Tutorial Periods
I	10	10	-
II	10	10	-
III	10	10	-
IV	10	10	-
V	10	10	-
TOTAL	50	50	-

I. CONTINUOUS INTERNAL ASSESSMENT : 40 Marks

(Internal Assessment Tests: 30, Attendance: 5, Assignment/Seminar: 5)

II.	**END SEMESTER EXAMINATION**	**: 60 Marks**
	TOTAL	**: 100 Marks**

FACULTY **HOD / MECH** **DEAN / FOE**

UNIT I

THEORY OF METAL CUTTING AND CUTTING TOOLS

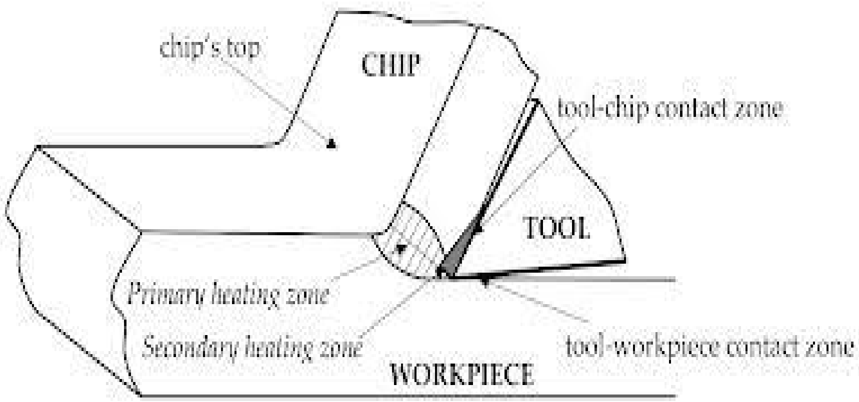
Definitions

Machining: Term applied to all material-removal processes

Metal cutting: The process in which a thin layer of excess metal (chip) is removed by a wedge-shaped single-point or multipoint cutting tool with defined geometry from a work piece, through a process of extensive plastic deformation

1.1 MECHANICS OF CHIP FORMATION

The cutting itself is a process of extensive plastic deformation to form a chip that is removed afterward. The basic mechanism of chip formation is essentially the same for all machining operations. Assuming that the cutting action is continuous, we can develop so-called continuous model of cutting process.



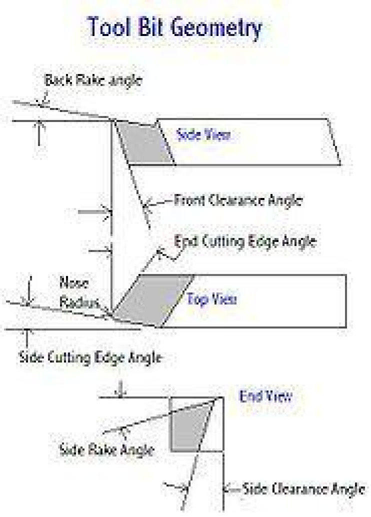
The cutting model shown above is oversimplified. In reality, chip formation occurs not in a plane but in so-called primary and secondary shear zones, the first one between the cut and chip, and the second one along the cutting tool face.

1.2 Single-point cutting tool,

As distinguished from other cutting tools such as a The cutting edge is ground to suit a particular machining operation and may be re sharpened or reshaped as needed. The ground tool bit is held rigidly by a tool holder while it is cutting.

Back Rake is to help control the direction of the chip, which naturally curves into the work due to the difference in length from the outer and inner parts of the cut. It also helps counteract the pressure against the tool from the work by pulling the tool into the work.

Side Rake along with back rake controls the chip flow and partly counteracts the resistance of the work to the movement of the cutter and can be optimized to suit the particular material being cut. Brass for example requires a back and side rake of 0 degrees while aluminum uses a back rake of 35 degrees and a side rake of 15 degrees. Nose Radius makes the finish of the cut smoother as it can overlap the previous cut and eliminate the peaks and valleys that a pointed tool produces. Having a radius also strengthens the tip, a sharp point being quite fragile.



All the other angles are for clearance in order that no part of the tool besides the actual cutting edge can touch the work. The front clearance angle is usually 8 degrees while the side clearance angle is 10-15 degrees and partly depends on the rate of feed expected.

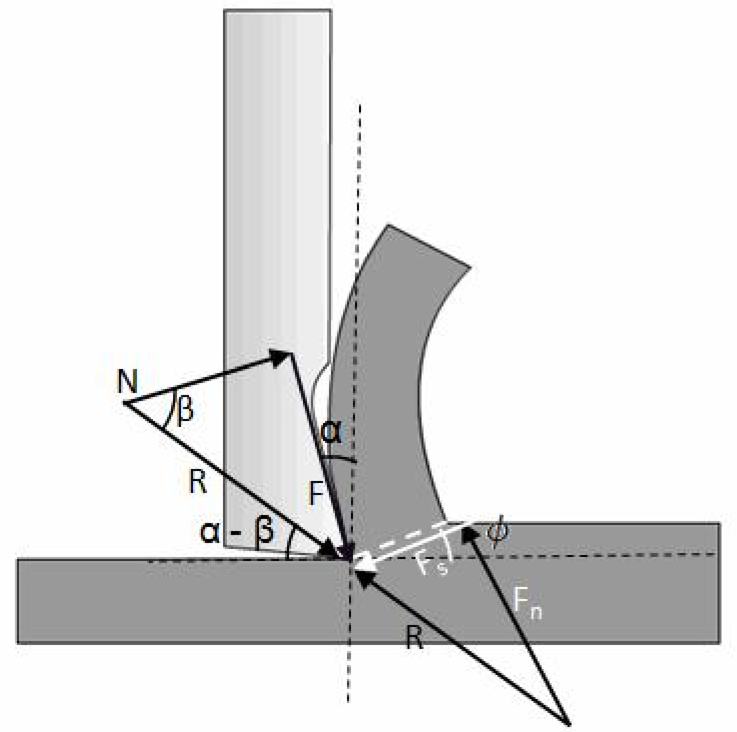
Minimum angles which do the job required are advisable because the tool gets weaker as the edge gets keener due to the lessening support behind the edge and the reduced ability to absorb heat generated by cutting.

The Rake angles on the top of the tool need not be precise in order to cut but to cut efficiently there will be an optimum angle for back and side rake.

1.3 Forces in machining

If you make a free body analysis of the chip, forces acting on the chip would be as follows.

At cutting tool side due to motion of chip against tool there will be a frictional force and a normal force to support that. At material side thickness of the metal increases while it flows from uncut to cut portion. This thickness increase is due to inter planar slip between different metal layers. There should be a shear force (Fs) to support this phenomenon. According to *shear plane theory* this metal layer slip happens at single plane called shear plane. So shear force acts on shear plane. Angle of shear plane can approximately determined using *shear plane theory* analysis. It is as follows $\phi = 45^o + \frac{\alpha}{2} - \frac{\beta}{2}$



Forces acting on the chip on tool side and shear plane side

Shear force on shear plane can be determined using shear strain rate and properties of material. A normal force (Fn) is also present perpendicular to shear plane. The resultant force (R) at cutting tool side and metal side should balance each other in order to make the chip in equilibrium. Direction of resultant force, R is determined as shown in Figure.

1.4 Types of chip

There are three types of chips that are commonly produced in cutting,

Discontinuous chips

Continuous chips

Continuous chips with built up edge

A discontinuous chip comes off as small chunks or particles. When we get this chip it may indicate,

Brittle work material

Small or negative rake angles

Coarse feeds and low speeds

A continuous chip looks like a long ribbon with a smooth shining surface. This chip type may indicate,

Ductile work materials

Large positive rake angles

Fine feeds and high speeds

Continuous chips with a built up edge still look like a long ribbon, but the surface is no longer smooth and shining. Under some circumstances (low cutting speeds of ~0.5 m/s, small or negative rake angles),

Work materials like mild steel, aluminum, cast iron, etc., tend to develop so-called built-up edge, a very hardened layer of work material attached to the tool face, which tends to act as a cutting edge itself replacing the real cutting tool edge. The built-up edge tends to grow until it reaches a critical size (~0.3 mm) and then passes off with the chip, leaving small fragments on the machining surface. Chip will break free and cutting forces are smaller, but the effects is a rough machined surface. The built-up edge disappears at high cutting speeds.

Chip control

Discontinuous chips are generally desired because

They are less dangerous for the operator

Do not cause damage to workpiece surface and machine tool

Can be easily removed from the work zone

Can be easily handled and disposed after machining.

There are three principle methods to produce the favourable discontinuous chip:

Proper selection of cutting conditions

Use of chip breakers

Change in the work material properties

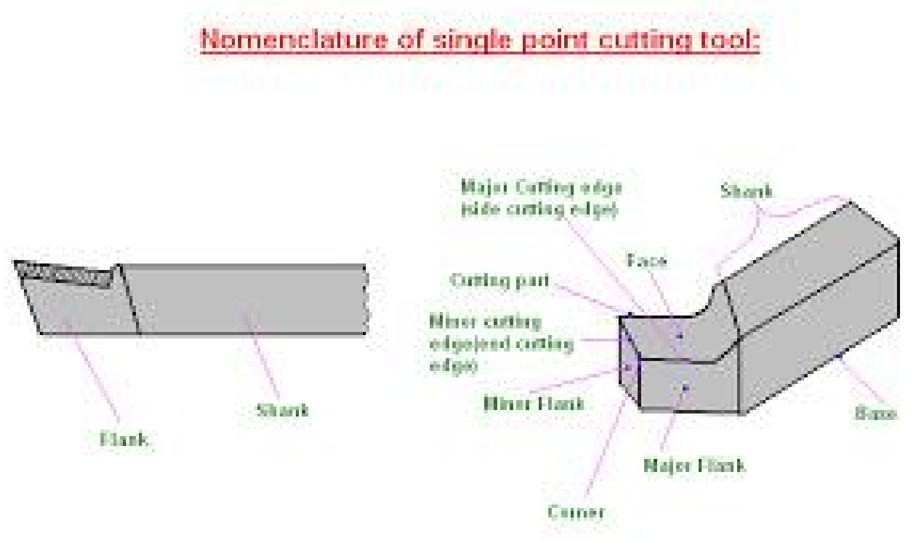
Chip breaker

Chip break and chip curl may be promoted by use of a so-called chip breaker. There are two types of chip breakers

External type, an inclined obstruction clamped to the tool face

Integral type, a groove ground into the tool face or bulges formed onto the tool face

1.5 Cutting tool nomenclature



Back Rake is to help control the direction of the chip, which naturally curves into the work due to the difference in length from the outer and inner parts of the cut. It also helps counteract the pressure against the tool from the work by pulling the tool into the work.

Side Rake along with back rake controls the chip flow and partly counteracts the resistance of the work to the movement of the cutter and can be optimized to suit the particular material being cut. Brass for example requires a back and side rake of 0 degrees while aluminum uses a back rake of 35 degrees and a side rake of 15 degrees.

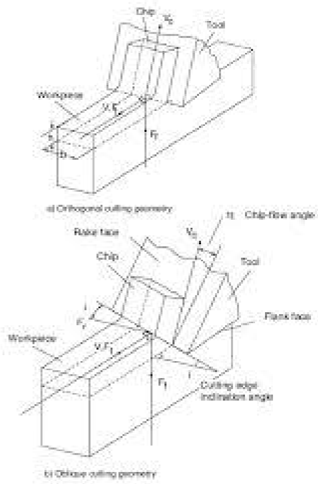
Nose Radius makes the finish of the cut smoother as it can overlap the previous cut and eliminate the peaks and valleys that a pointed tool produces. Having a radius also strengthens the tip, a sharp point being quite fragile.

All the other angles are for clearance in order that no part of the tool besides the actual cutting edge can touch the work. The front clearance angle is usually 8 degrees while the side clearance angle is 10-15 degrees and partly depends on the rate of feed expected.

Minimum angles which do the job required are advisable because the tool gets weaker as the edge gets keener due to the lessening support behind the edge and the reduced ability to absorb heat generated by cutting.

The Rake angles on the top of the tool need not be precise in order to cut but to cut efficiently there will be an optimum angle for back and side rake.

1.6 Orthogonal metal cutting



Orthogonal metal cutting	Oblique metal cutting
Cutting edge of the tool is perpendicular to the direction of tool travel.	The cutting edge is inclined at an angle less than 90^o to the direction of tool travel.
The direction of chip flow is perpendicular to the cutting edge.	The chip flows on the tool face making an angle.
The chip coils in a tight flat spiral	The chip flows side ways in a long curl.
For same feed and depth of cut the force which shears the metal acts on smaller areas. So the life of the tool is less.	The cutting force acts on larger area and so tool life is more.
Produces sharp corners.	Produces a chamfer at the end of the cut
Smaller length of cutting edge is in contact with the work.	For the same depth of cut greater length of cutting edge is in contact with the work.
Generally parting off in lathe, broaching and slotting operations are done in this method.	This method of cutting is used in almost all machining operations.

Depending on whether the stress and deformation in cutting occur in a plane (two-dimensional case) or in the space (three-dimensional case), we consider two principle types of cutting:

Orthogonal cutting the cutting edge is straight and is set in a position that is perpendicular to the direction of primary motion. This allows us to deal with stresses and strains that act in a plane.

Oblique cutting the cutting edge is set at an angle.

According to the number of active cutting edges engaged in cutting, we distinguish again two types of cutting:

Single-point cutting the cutting tool has only one major cutting edge

Examples: turning, shaping, boring

Multipoint cutting the cutting tool has more than one major cutting edge

Examples: drilling, milling, broaching, reaming. Abrasive machining is by definition a process of multipoint cutting.

Cutting conditions

Each machining operation is characterized by cutting conditions, which comprises a set of three elements:

Cutting velocity: The traveling velocity of the tool relative to the work piece. It is measured in m/s or m/min.

Depth of cut: The axial projection of the length of the active cutting tool edge, measured in mm. In orthogonal cutting it is equal to the actual width of cut.

Feed: The relative movement of the tool in order to process the entire surface of the work piece. In orthogonal cutting it is equal to the thickness of cut and is measured in mm.

1.7 Thermal aspects

In cutting, nearly all of energy dissipated in plastic deformation is converted into heat that in turn raises the temperature in the cutting zone. Since the heat generation is closely related to the plastic deformation and friction, we can specify three main sources of heat when cutting,

Plastic deformation by shearing in the primary shear zone

Plastic deformation by shearing and friction on the cutting face

Friction between chip and tool on the tool flank

Heat is mostly dissipated by,

The discarded chip carries away about 60~80% of the total heat

The workpiece acts as a heat sink drawing away 10~20% heat

The cutting tool will also draw away ~10% heat

If coolant is used in cutting, the heat drawn away by the chip can be as big as 90% of the total heat dissipated. Knowledge of the cutting temperature is important because it:

Affects the wear of the cutting tool. Cutting temperature is the primary factor affecting the cutting tool wear can induce thermal damage to the machined surface. High surface temperatures promote the process of oxidation of the machined surface. The oxidation layer has worse mechanical properties than the base material, which may result in shorter service life. Causes dimensional errors in the machined surface. The cutting tool elongates as a result of the increased temperature, and the position of the cutting tool edge shifts toward the machined surface, resulting in a dimensional error of about 0.01~0.02 mm. Since the processes of thermal generation, dissipation, and solid body thermal deformation are all transient, some time is required to achieve a steady-state condition

Cutting temperature determination

Cutting temperature is either measured in the real machining process, or predicted in the machining process design. The mean temperature along the tool face is measured directly by means of different thermocouple techniques, or indirectly by measuring the infrared radiation, or examination of change in the tool material microstructure or micro hardness induced by temperature. Some recent indirect methods are based on the examination of the temper color of a chip, and on the use of thermo sensitive paints.

There are no simple reliable methods of measuring the temperature field. Therefore, predictive approaches must be relied on to obtain the mean cutting temperature and temperature field in the chip, tool and work piece.

For cutting temperature prediction, several approaches are used:

Analytical methods: there are several analytical methods to predict the mean temperature. The interested readers are encouraged to read more specific texts, which present in detail these methods. Due to the complex nature of the metal cutting process, the analytical methods are typically restricted to the case of orthogonal cutting.

Numerical methods: These methods are usually based on the finite element modeling of metal cutting. The numerical methods, even though more complex than the analytical approaches, allow for prediction not only of the mean cutting temperature along the tool face but also the temperature field in orthogonal and oblique cutting.

1.8 Cutting tool materials

Requirements

The cutting tool materials must possess a number of important properties to avoid excessive wear, fracture failure and high temperatures in cutting, the following characteristics are essential for cutting materials to withstand the heavy conditions of the cutting process and to produce high quality and economical parts:

Hardness at elevated temperatures (so-called hot hardness) so that hardness and strength of the tool edge are maintained in high cutting temperatures:

Toughness: ability of the material to absorb energy without failing. Cutting if often accompanied by impact forces especially if cutting is interrupted, and cutting tool may fail very soon if it is not strong enough.

Wear resistance: although there is a strong correlation between hot hardness and wear resistance, later depends on more than just hot hardness. Other important characteristics include surface finish on the tool, chemical inertness of the tool material with respect to the work material, and thermal conductivity of the tool material, which affects the maximum value of the cutting temperature at tool-chip interface.

Cutting tool materials

Carbon Steels

It is the oldest of tool material. The carbon content is 0.6~1.5% with small quantities of silicon, Chromium, manganese, and vanadium to refine grain size. Maximum hardness is about HRC 62. This material has low wear resistance and low hot hardness. The use of these materials now is very limited.

High-speed steel (HSS)

First produced in 1900s. They are highly alloyed with vanadium, cobalt, molybdenum, tungsten and Chromium added to increase hot hardness and wear resistance. Can be hardened to various depths by appropriate heat treating up to cold hardness in the range of HRC 63-65. The cobalt component give the material a hot hardness value much greater than carbon steels. The high toughness and good wear resistance make HSS suitable for all type of cutting tools with complex shapes for relatively low to medium cutting speeds. The most widely used tool material today for taps, drills, reamers, gear tools, end cutters, slitting, broaches, etc.

Cemented Carbides

Introduced in the 1930s. These are the most important tool materials today because of their high hot hardness and wear resistance. The main disadvantage of cemented carbides is their low toughness. These materials are produced by powder metallurgy methods, sintering grains of tungsten carbide (WC) in a cobalt (Co) matrix (it provides toughness). There may be other carbides in the mixture, such as titanium carbide (TiC) and/or tantalum carbide (TaC) in addition to WC.

Ceramics

Ceramic materials are composed primarily of fine-grained, high-purity aluminum oxide (Al2O3), pressed and sintered with no binder. Two types are available:

White, or cold-pressed ceramics, which consists of only Al2O3 cold pressed into inserts and sintered at high temperature.

Black, or hot-pressed ceramics, commonly known as cermets (from ceramics and metal). This material consists of 70% Al2O3 and 30% TiC. Both materials have very high wear resistance but low toughness; therefore they are suitable only for continuous operations such as finishing turning of cast iron and steel at very high speeds. There is no occurrence of built-up edge, and coolants are not required.

Cubic boron nitride (CBN) and synthetic diamonds

Diamond is the hardest substance ever known of all materials. It is used as a coating material in its polycrystalline form, or as a single-crystal diamond tool for special applications, such as mirror finishing of non-ferrous materials. Next to diamond, CBN is the hardest tool material. CBN is used mainly as coating material because it is very brittle. In spite of diamond, CBN is suitable for cutting ferrous materials.

1.9 Tool wear and tool life

The life of a cutting tool can be terminated by a number of means, although they fall broadly into two main categories:

Gradual wearing of certain regions of the face and flank of the cutting tool, and abrupt tool failure. Considering the more desirable case Œ the life of a cutting tool is therefore determined by the amount of wear that has occurred on the tool profile and which reduces the efficiency of cutting to an unacceptable level, or eventually causes tool failure. When the tool wear reaches an initially accepted amount, there are two options,

To resharpen the tool on a tool grinder, or

To replace the tool with a new one.

This second possibility applies in two cases,

When the resource for tool resharpening is exhausted. or

The tool does not allow for resharpening, e.g. in case of the indexable carbide inserts

Wear zones

Gradual wear occurs at three principal locations on a cutting tool. Accordingly, three main types of tool wear can be distinguished,

Crater wear

Flank wear

Corner wear

Crater wear: consists of a concave section on the tool face formed by the action of the chip sliding on the surface. Crater wear affects the mechanics of the process increasing the actual rake angle of the cutting tool and consequently, making cutting easier. At the same time, the crater wear weakens the tool wedge and increases the possibility for tool breakage. In general, crater wear is of a relatively small concern.

Flank wear: occurs on the tool flank as a result of friction between the machined surface of the workpiece and the tool flank. Flank wear appears in the form of so-called wear land and is measured by the width of this wear land, VB, Flank wear affects to the great extend the mechanics of cutting. Cutting forces increase significantly with flank wear. If the amount of flank wear exceeds some critical value (VB > 0.5~0.6 mm), the excessive cutting force may cause tool failure.

Corner wear: occurs on the tool corner. Can be considered as a part of the wear land and respectively flank wear since there is no distinguished boundary between the corner wear and flank wear land. We consider corner wear as a separate wear type because of its importance for the precision of machining. Corner wear actually shortens the cutting tool thus increasing gradually the dimension of machined surface and introducing a significant dimensional error in machining, which can reach values of about 0.03~0.05 mm.

Tool life

Tool wear is a time dependent process. As cutting proceeds, the amount of tool wear increases gradually. But tool wear must not be allowed to go beyond a certain limit in order to avoid tool failure. The most important wear type from the process point of view is the flank wear, therefore the parameter which has to be controlled is the width of flank wear land, VB. This parameter must not exceed an initially set safe limit, which is about 0.4 mm for carbide cutting tools. The safe limit is referred to as allowable wear land (wear criterion),

. The cutting time required for the cutting tool to develop a flank wear land of width is called tool life, T, a fundamental parameter in machining. The general relationship of VB versus cutting time is shown in the figure (so-called wear curve). Although the wear curve shown is for flank wear, a similar relationship occurs for other wear types. The figure shows also how to define the tool life T for a given wear criterion VBk

Parameters, which affect the rate of tool wear, are

Cutting conditions (cutting speed V, feed f, depth of cut d)

Cutting tool geometry (tool orthogonal rake angle)

Properties of work material

1.10 Surface finish

The machining processes generate a wide variety of surface textures. Surface texture consists of the repetitive and/or random deviations from the ideal smooth surface. These deviations are

Roughness: small, finely spaced surface irregularities (micro irregularities)

Waviness: surface irregularities of grater spacing (macro irregularities)

Lay: predominant direction of surface texture

Three main factors make the surface roughness the most important of these parameters:

Fatigue life: the service life of a component under cyclic stress (fatigue life) is much shorter if the surface roughness is high

Bearing properties: a perfectly smooth surface is not a good bearing because it cannot maintain a lubricating film.

Wear: high surface roughness will result in more intensive surface wear in friction.

Surface finish is evaluated quantitatively by the average roughness height, Ra

Roughness control

Factors, influencing surface roughness in machining are

Tool geometry (major cutting edge angle and tool corner radius),

Cutting conditions (cutting velocity and feed), and

Work material properties (hardness).

The influence of the other process parameters is outlined below:

Increasing the tool rake angle generally improves surface finish

Higher work material hardness results in better surface finish

Tool material has minor effect on surface finish.

Cutting fluids affect the surface finish changing cutting temperature and as a result the built-up edge formation.

1.11 Cutting fluids

Cutting fluid (coolant) is any liquid or gas that is applied to the chip and/or cutting tool to improve cutting performance. A very few cutting operations are performed dry, i.e., without the application of cutting fluids. Generally, it is essential that cutting fluids be applied to all machining operations.

Cutting fluids serve three principle functions:

To remove heat in cutting: the effective cooling action of the cutting fluid depends on the method of application, type of the cutting fluid, the fluid flow rate and pressure. The most effective cooling is provided by mist application combined with flooding. Application of fluids to the tool flank, especially under pressure, ensures better cooling that typical application to the chip but is less convenient.

To lubricate the chip-tool interface: cutting fluids penetrate the tool-chip interface improving lubrication between the chip and tool and reducing the friction forces and temperatures.

To wash away chips: this action is applicable to small, discontinuous chips only. Special devices are subsequently needed to separate chips from cutting fluids.

Methods of application

Manual application

Application of a fluid from a can manually by the operator. It is not acceptable even in job-shop situations except for tapping and some other operations where cutting speeds are very low and friction is a problem. In this case, cutting fluids are used as lubricants.

Flooding

In flooding, a steady stream of fluid is directed at the chip or tool-workpiece interface. Most machine tools are equipped with a recirculating system that incorporates filters for cleaning of cutting fluids. Cutting fluids are applied to the chip although better cooling is obtained by applying it to the flank face under pressure

Coolant-fed tooling

Some tools, especially drills for deep drilling, are provided with axial holes through the body of the tool so that the cutting fluid can be pumped directly to the tool cutting edge.

Mist applications

Fluid droplets suspended in air provide effective cooling by evaporation of the fluid. Mist application in general is not as effective as flooding, but can deliver cutting fluid to inaccessible areas that cannot be reached by conventional flooding.

Types of cutting fluid

Cutting Oils

Cutting oils are cutting fluids based on mineral or fatty oil mixtures. Chemical additives like sulphur improve oil lubricant capabilities. Areas of application depend on the properties of the particular oil but commonly, cutting oils are used for heavy cutting operations on tough steels.

Soluble Oils

The most common, cheap and effective form of cutting fluids consisting of oil droplets suspended in water in a typical ratio water to oil 30:1. Emulsifying agents are also added to promote stability of emulsion. For heavy-duty work, extreme pressure additives are used. Oil emulsions are typically used for aluminum and cooper alloys.

Chemical fluids

These cutting fluids consist of chemical diluted in water. They possess good flushing and cooling abilities. Tend to form more stable emulsions but may have harmful effects to the skin.

Environmental issues

Cutting fluids become contaminated with garbage, small chips, bacteria, etc., over time. Alternative ways of dealing with the problem of contamination are:

Replace the cutting fluid at least twice per month,

Machine without cutting fluids (dry cutting), Use a filtration system to continuously clean the cutting fluid.

Disposed cutting fluids must be collected and reclaimed. There are a number of methods of reclaiming cutting fluids removed from working area. Systems used range from simple settlement tanks to complex filtration and purification systems. Chips are emptied from the skips into a pulverizer and progress to centrifugal separators to become a scrap material. Neat oil after separation can be processed and returned, after cleaning and sterilizing to destroy bacteria.

1.12 Machinability

Machinability is a term indicating how the work material responds to the cutting process. In the most general case good machinability means that material is cut with good surface finish, long tool life, low force and power requirements, and low cost.

Machinability of different materials

Steels Leaded steels: lead acts as a solid lubricant in cutting to improve considerably machinability.

Resulphurized steels: sulphur forms inclusions that act as stress raisers in the chip formation zone thus increasing machinability.

Difficult-to-cut steels: a group of steels of low machinability, such as stainless steels, high manganese steels, precipitation-hardening steels.

Other metals

Aluminum: easy-to-cut material except for some cast aluminum alloys with silicon content that may be abrasive.

Cast iron: gray cast iron is generally easy-to-cut material, but some modifications and alloys are abrasive or very hard and may cause various problems in cutting.

Cooper-based alloys: easy to machine metals. Bronzes are more difficult to machine than brass.

Selection of cutting conditions

For each machining operation, a proper set of cutting conditions must be selected during the process planning. Decision must be made about all three elements of cutting conditions,

Depth of cut

Feed

Cutting speed

There are two types of machining operations:

Roughing operations: the primary objective of any roughing operation is to remove as much as possible material from the work piece for as short as possible machining time. In roughing operation, quality of machining is of a minor concern.

Finishing operations: the purpose of a finishing operation is to achieve the final shape, dimensional precision, and surface finish of the machined part. Here, the quality is of major importance. Selection of cutting conditions is made with respect to the type of machining operation. Cutting conditions should be decided in the order depth of cut - feed - cutting speed.

MCQ

Questions	Option A	Option B	Option C	Option D	Ans
Which of the following is the chip removal process	Broching	Extrusion	Die-casting	Rolling	Broching
In machining tool should be ____ than the work piece	Harder	Softer	Tough	None of the above	Harder
Orthogonal cutting is also known as	One-diemensional	Two-dimensional	Three-dimensional	None of the above	Two-dimensional
Oblique cutting is also known as	One-diemensional	Two-dimensional	Three-dimensional	Four-dimensional	Three-dimensional
The force which shears the metal acts on a larger area in the case of	Orthogonal cutting	Oblique cutting	Metal cutting	None of the above	Oblique cutting
In Oblique cutting shear stress	More	Less	Moderate	None of the above	Less
Shear stress is more in	Orthogonal cutting	Oblique cutting	Metal cutting	None of the above	Orthogonal cutting
In orthogonal cutting maximum chip thickness occurs at	Middle	Side	Edge	None of the above	Middle
The plane along which the element shears is called	Metal plane	shear plane	shear zone	Tool zone	shear plane
Continuous chips produced while machine________ material	Brittle	Ductile	Hard	None of the above	Ductile
Continuous chips produced at _________ speed	High	Low	Medium	None of the above	Hgh
Continuous chips formed when cutting edge of the tool was	sharp	dull	Broad	None of the above	Sharp
Continuous chips have ________ friction	More	Moderate	Less	None of the above	Less
A tool with dull cutting edge will form	Continuous	Discontinuous	Built up edge	None of the	Built up edge

_____ chip				above	
______ chip is formed while machining steel	Continuous	Discontinuous	Built up edge	None of the above	Continuous
Discontinuous chips formed while Machine __________ material	Brittle	Ductile	Hard	None of the above	Brittle
The discontinuous chips sometimes called as	Ribbon type	Segmental type	BUE	None of the above	Segmental type
_________ chips are formed when machining cast iron	Continuous	Discontinuous	Built up edge	None of the above	Discontinuous
Continuous chips produced at ________ speed	High	Low	Medium	None of the above	Low
Chips flows through _____ face of the tool	Rake	Flank	side	None of the above	Rake
65% to 75% heat generated in metal cutting at	Tool	shear plane	Workpiece	Chip	shear plane
In Merchant circle the uncut chip thicness assumed as_____	Constant	Equal to chip thickness	Less than chip thickness	None of the above	Constant
In Merchant circle the velocity of the work is assumed as_______	Uniform	Varied	Neither uniform nor varied	None of the above	Uniform
In Merchant circle width of the tool is _______ Width of workpiece	Greater than	Less than	Equal to	None of the above	Greater than
Crater wear occurs maily due to following phenomena	Diffusion	Abrasion	Adhesion	None of the above	Diffusion
In machining, chips breaks due to	Plasticity	Ductility	Workhardening	Shear	Workhardening
Crater wear takes place in a single point cutting tool at	Flank	Tip	Face	Chip	Face
Best coolant and lubricant for castiron is	Water	Dry	Soluble oil	Oil	Dry
The worn out region at the flank is	Wear land	Crater	Wear	None of the	Wear land

called				above	
Flank wear takes place when machining	Plastic	Brittle	Diamond	Ductile	Brittle
Choose the one which affects the tool life	Cutting speed	Tool tip	Work material	Cutting ratio	Cutting speed
Which of the following tool materials has highest cutting speed	Tool steel	Carbide	Carbon steel	HSS	Carbide
The function of the cutting fluid is	Cool the tool and workpiece	Provide lubrication	Wash away the chips	All of these	All of these
The cutting fluid shoud be ________	Transperent	Dark	Any colour	None of the above	Transperent
Viscosity of the cutting Fluid should be	High	Low	Moderate	None of the above	Low
Ceramic tools are made from	Tungsten oxide	silicon carbide	Cobalt	HCS	silicon carbide
Poor surface finish results due to	Coarse feed	Heavy depth of cut	Low cutting speed	Speed	Coarse feed
The hardest manufactured cutting tool material is	Diamond	Cemented carbide	Steel	HSS	Cemented carbide
No cutting fluid is normally used while machining	Mild steel	cast iron	Aluminium	Diamond	cast iron
The cutting fluid shoud be ________	Odour	Odourless	Pleasant smell	None of the above	Odourless
Carbon steels are lose their hardness at temp	200 to 250	250 to 300	100 to 200	500 to 700	200 to 250
Cutting fluids should have ________ conductivity	High	Low	Moderate	None of the above	High
Select the solid lubricant from the following	Water	Stick waxes	Oils	Waste	Stick waxes
Segmental chips are formed when machining	Ductile	Brittle	Heat treated	Hard material	Brittle

With increase in cutting speed, the finish	Remain same	Improves slightly	Gets poor	Decrease	Improves slightly
In tool life equation n is known as	Cutting speed	Tool life	Constant	None of the above	Constant
In tool life equation V is known as	Cutting speed	Tool life	Constant	None of the above	Cutting speed
In tool life equation T is known as	Cutting speed	Tool life	Constant	None of the above	Tool life
Thermal conductivity of the tool should be	High	Low	Moderate	None of the above	High
Retaining hardness at elevated temperature is known as	Hot hardeness	Thermal conductivity	Hardness	None of the above	Hot hardness

2 marks

1. What is tool signature?
2. What is side rake angle? And mention its effects?
3. What is clearance angle? And mention its types?
4. Explain the nose radius.
5. Sketch the orthogonal cutting.
6. What is shear plane?
7. What is cutting force?
8. What is chip and mention its different types?
9. Define machinability of metal.
10. Write Taylor's tool life equation

14 marks

1. Explain orthogonal cutting and oblique cutting with its neat sketches and compare.
2. What is the tool life equation and state the factor affecting the tool life.
3. What is machinability? And explain.
4. Explain the various tool materials.
5. Write short notes on surface finish.
6. What are the different types of cutting fluids used in machining process?
7. Write short notes tool wear.

UNIT II

CENTRE LATHE AND SEMI AUTOMATIC LATHES

2.1 Center Lathes

A lathe is a machine tool that rotates the work piece against a tool whose position it controls. The spindle is the part of the lathe that rotates. Various work holding attachments such as three jaw chucks, collets, and centers can be held in the spindle. The spindle is driven by an electric motor through a system of belt drives and gear trains. Spindle rotational speed is controlled by varying the geometry of the drive train.

The tailstock can be used to support the end of the workpiece with a center, or to hold tools for drilling, reaming, threading, or cutting tapers. It can be adjusted in position along the ways to accommodate different length workpieces. The tailstock barrel can be fed along the axis of rotation with the tailstock hand wheel.

The carriage controls and supports the cutting tool. It consists of:

A saddle that slides along the ways;

An apron that controls the feed mechanisms;

A cross slide that controls transverse motion of the tool (toward or away from the operator);

A tool compound that adjusts to permit angular tool movement; v a tool post that holds the cutting tools.

There are a number of different lathe designs, and some of the most popular are discussed here.

Centre lathe

The basic, simplest and most versatile lathe.

This machine tool is manually operated that is why it requires skilled operators. Suitable for low and medium production and for repair works.

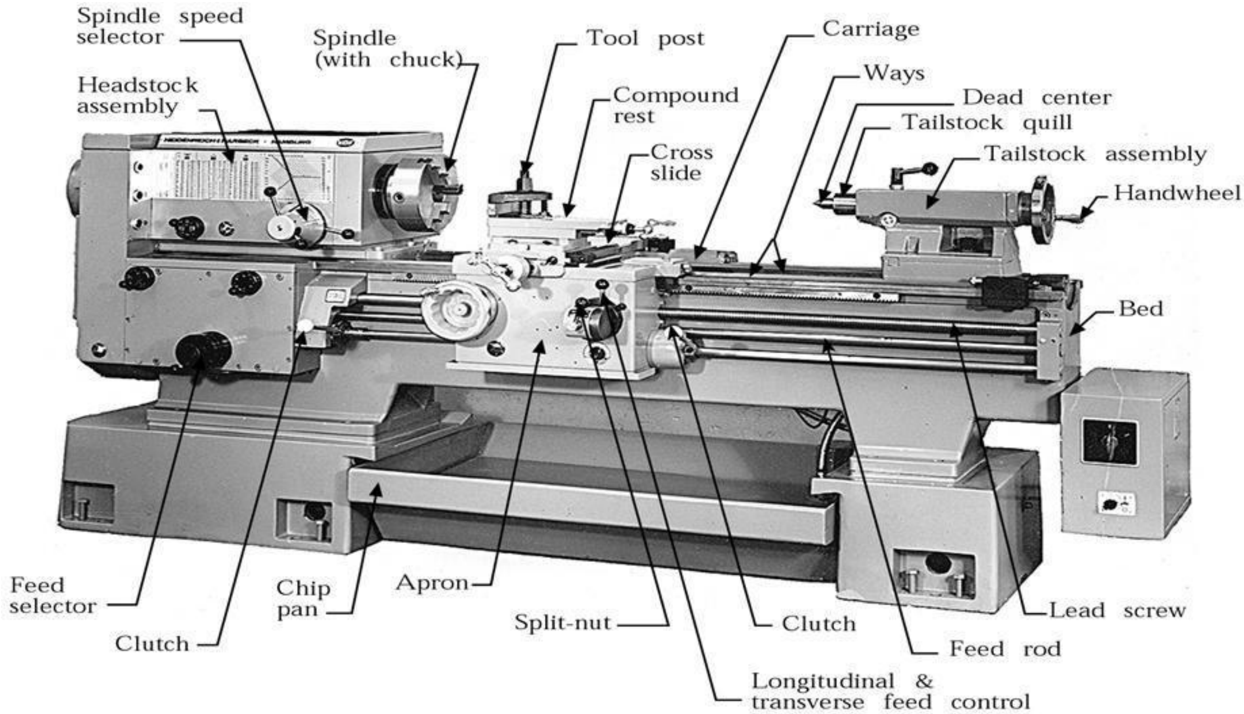
There are two tool feed mechanism in the engine lathes. These cause the cutting tool to move when engaged.

The lead screw will cause the apron and cutting tool to advance quickly. This is used for cutting threads, and for moving the tool quickly.

The feed rod will move the apron and cutting tool slowly forward. This is largely used for most of the turning operations.

Work is held in the lathe with a number of methods.

Between two centers. The work piece is driven by a device called a dog; the method is suitable for parts with high length-to-diameter ratio.

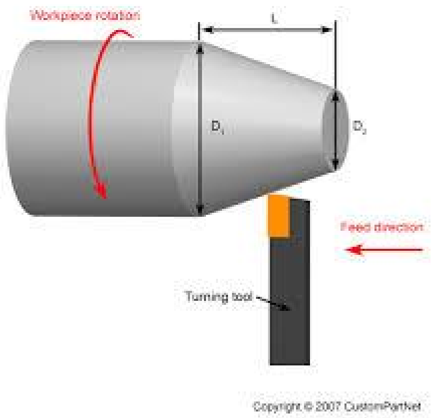


A 3 jaw self-centering chuck is used for most operations on cylindrical work parts. For parts with high length-to-diameter ratio the part is supported by center on the other end.

Collet consists of tubular bushing with longitudinal slits. Collets are used to grasp and hold bar stock. A collet of exact diameter is required to match any bar stock diameter.

A face plate is a device used to grasp parts with irregular shapes:

2.2 Taper turning methods



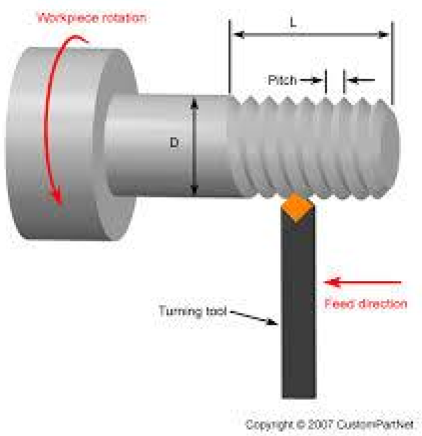
A taper is a conical shape. Tapers can be cut with lathes quite easily. There are some common methods for turning tapers on an center lathe,

Using a form tool: This type of tool is specifically designed for one cut, at a certain taper angle. The tool is plunged at one location, and never moved along the lathe slides. v Compound Slide

Method: The compound slide is set to travel at half of the taper angle. The tool is then fed across the work by hand, cutting the taper as it goes. v Off-Set Tail Stock: In this method the normal rotating part of the lathe still drives the workpiece (mounted between centres), but the centre at the tailstock is offset towards/away from the cutting tool. Then, as the cutting tool passes over, the part is cut in a conical shape. This method is limited to small tapers over long lengths.The tailstock offset h is defined by

$h = L\sin\alpha$, where L is the length of work piece, and α is the half of the taper angle.

2.3 Thread cutting methods



Different possibilities are available to produce a thread on a lathe. Threads are cut using lathes by advancing the cutting tool at a feed exactly equal to the thread pitch. The single-point cutting tool cuts in a helical band, which is actually a thread. The procedure calls for correct settings of the machine, and also that the helix be restarted at the same location each time if multiple passes are required to cut the entire depth of thread. The tool point must be ground so that it has the same profile as the thread to be cut.

Another possibility is to cut threads by means of a thread die (external threads), or a tap (internal threads). These operations are generally performed manually for small thread diameters.

2.4 Special Attachments

Unless a workpiece has a taper machined onto it which perfectly matches the internal taper in the spindle, or has threads which perfectly match the external threads on the spindle (two conditions which rarely exist), an accessory must be used to mount a workpiece to the spindle.

A workpiece may be bolted or screwed to a faceplate, a large, flat disk that mounts to the spindle. In the alternative, faceplate dogs may be used to secure the work to the faceplate.

A workpiece may be mounted on a mandrel, or circular work clamped in a three- or four-jaw chuck. For irregular shaped workpieces it is usual to use a four jaw (independent moving jaws) chuck. These holding devices mount directly to the Lathe headstock spindle.

In precision work, and in some classes of repetition work, cylindrical workpieces are usually held in a collet inserted into the spindle and secured either by a draw-bar, or by a collet closing cap on the spindle. Suitable collets may also be used to mount square or hexagonal workpieces. In precision tool making work such collets are usually of the draw-in variety, where, as the collet is tightened, the workpiece moves slightly back into the headstock, whereas for most repetition work the dead length variety is preferred, as this ensures that the position of the workpiece does not move as the collet is tightened.

A soft workpiece (e.g., wood) may be pinched between centers by using a spur drive at the headstock, which bites into the wood and imparts torque to it.

2.5 Machining time

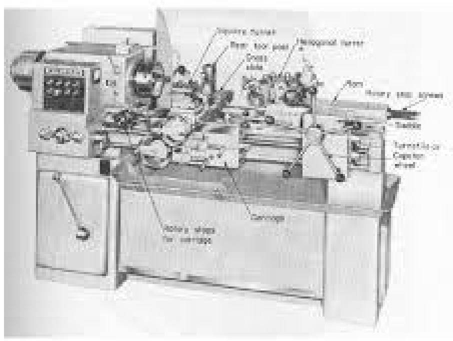
Machining time is the time when a machine is actually processing something. Generally, machining time is the term used when there is a reduction in material or removing some undesirable parts of a material. For example, in a drill press, machining time is when the cutting edge is actually moving forward and making a hole. Machine time is used in other situations, such as when a machine installs screws in a case automatically.

One of the important aspects in manufacturing calculation is how to find and calculate the machining time in a machining operation. Generally, machining is family of processes or operations in which excess material is removed from a starting work piece by a sharp cutting tool so the remaining part has the desired geometry and the required shape. The most common machining operations can be classified into four types: turning, milling, drilling and lathe work.

Calculate Time for Turning

$$\text{Time for Turning} = \frac{\text{Length of the job to be turned}}{\text{Feed/Rev.} \times \text{r.p.m.}} \text{ min.}$$

2.6 Capstan versus turret



Capstan Lathe Turret Lathe

The term "capstan lathe" overlaps in sense with the term "turret lathe" to a large extent. In many times and places, it has been understood to be synonymous with "turret lathe". In other times and places it has been held in technical contradistinction to "turret lathe", with the difference being in whether the turret's slide is fixed to the bed (ram-type turret) or slides on the bed's ways (saddle-type turret). The difference in terminology is mostly a matter of United Kingdom and Common wealth usage versus United States usage. American usage tends to call them all "turret lathes".

The word "capstan" could logically seem to refer to the turret itself, and to have been inspired by the nautical capstan. A lathe turret with tools mounted in it can very much resemble a nautical capstan full of handspikes. This interpretation would lead Americans to treat "capstan" as a synonym of "turret" and "capstan lathe" as a synonym of "turret lathe". However, the multi-spoked handles that the operator uses to advance the slide are also called capstans, and they themselves also resemble the nautical capstan.

No distinction between "turret lathe" and "capstan lathe" persists upon translation from English into other languages. Most translations involve the term "revolver", and serve to translate either of the English terms.

The words "turret" and "tower", the former being a diminutive of the latter, come ultimately from the Latin "turris", which means "tower", and the use of "turret" both to refer to lathe turrets and to refer to gun turrets seems certainly to have been inspired by its earlier connection to the turrets of fortified buildings and to siege towers. The history of the rook in chess is connected to the same history, with the French word for rook, tour, meaning "tower".

It is an interesting coincidence that the word "tour" in French can mean both "lathe" and "tower", with the first sense coming ultimately from Latin "tornus", "lathe", and the second sense coming ultimately from Latin "turris", "tower". "Tour revolver", "tour tourelle", and "tour tourelle revolver" are various ways to say "turret lathe" in French.

2.7 Semi-automatic

Sometimes machines similar to those above, but with power feeds and automatic turret-indexing at the end of the return stroke, are called "semi-automatic turret lathes". This nomenclature distinction is blurry and not consistently observed. The term "turret lathe" encompasses them all. During the 1860s, when semi-automatic turret lathes were developed, they were sometimes called "automatic". What we today would call "automatics", that is, fully automatic machines, had not been developed yet. During that era both manual and semi-automatic turret lathes were sometimes called "screw machines", although we today reserve that term for fully automatic machines.

2.8 Automatic

During the 1870s through 1890s, the mechanically automated "automatic" turret lathe was developed and disseminated. These machines can execute many part-cutting cycles without human intervention. Thus the duties of the operator, which were already greatly reduced by the manual turret lathe, were even further reduced, and productivity increased. These machines use cams to automate the sliding and indexing of the turret and the opening and closing of the chuck. Thus, they execute the part-cutting cycle somewhat analogously to the way in which an elaborate cuckoo clock performs an automated theater show. Small- to medium-sized automatic turret lathes are usually called "screw machines" or "automatic screw machines", while larger ones are usually called "automatic chucking lathes", "automatic chuckers", or "chuckers".

MCQ

Questions	Option A	Option B	Option C	Option D	Ans
Process of removing unwanted material from the workpiece is known as	Casting	Welding	Machining	all the above	Machining
lathe used for machining	hollow components	prismatic components	cylindrical components	all the above	cylindrical components
One of the important parameters of lathe specification is	Swing over tool bed	Swing over tool post	Horse power	Motor	Swing over tool bed
The arrangement of bed column and frame is known as	Structure	Base	Slides	all the above	Structure
Lathe bed is made up of	Steel	Cast iron	Cabide steels	all the above	Cast iron
Bed should have	Compressive strength	Shock absorbing capacity	High hardness	all the above	all the above
Bed is made up of ______ piece	Single	Double	Multiple	None of the above	Single
_________ is used for varying speed of the lathe	Spindle	Motor	Step cone pulley	None of the above	Step cone pulley
Head stock is placed at _______ end of the bed	Left	Right	Middle	None of the above	Left
______ is used to hold the workpiece	Tool post	Carriage	Chuck	tail stock	Chuck
Tail stock is placed at _______ end of the bed	Left	Right	Middle	None of the above	Right
Dead centre is used to hold______	Length jobs	Short workpiece	Both a & b	None of the above	Length jobs
______ is used to hold and feed the drills and reamers in lathes	Chuck	Tool post	Tail stock	Carriage	Tail stock
Angles (in degrees) are marked at	Compound rest	Saddle	Tool post	Apron	Compound rest
Tool is clamped on	Tool post	Compound test	Chuck	Carriage	Tool post

Tool post, Saddle, Compound rest, Apron are placed at	Carriage	Head stock	Tail stock	Saddle	Carriage
Lathe can perform	Turning	Drilling	Thread cutting	all the above	all the above
The process of reducing length of the workpiece is called as	Turning	Knurling	Facing	Drilling	Facing
The process of reducing diameter of the workpiece is called as	Turning	Knurling	Facing	Drilling	Turning
The process of producing different diameters on the workpiece is	Step Turning	Knurling	Facing	Drilling	Step Turning
The uniform increasing of diameter on the workpiece is	Facing	Turning	Step Turning	Taper turning	Taper turning
By setting over the tail stock centre is for _________ operation	Facing	Turning	Step Turning	Taper turning	Taper turning
By swivelling the comp[ound rest is suitable for	Long tapers	Short tapers	Medium tapers	None of the above	Short tapers
The process of producing cylindrical hole on the workpiece is	Step Turning	Knurling	Facing	Drilling	Drilling
The process of enlarging the hole is	Step Turning	Boring	Facing	Drilling	Boring
The process of finishing the hole is	Step Turning	Boring	Reaming	Drilling	Reaming
The process of reducing diameter of the workpiece over narrow surface is	Turning	Knurling	Facing	Grooving	Grooving
Operation of embossing the diamond shapped pattern on workpiece	Turning	Knurling	Facing	Grooving	Knurling
__________ is used to hold the work in conjunction with work piece	Face plate	Angle plate	Driving plate	all the above	Angle plate
Mandrels are also called as _______	Face plate	Angle plate	Driving plate	Arbors	Arbors
___________ is used to hold bored parts in lateh	Face plate	Angle plate	Driving plate	Arbors	Arbors
Mandrels are available in	Collar	Stepped	Expanding	all the above	all the above
Volume of material removed per unit time is called	MRR	Depth of cut	Feed	None of the above	MRR

Speed X Feed X Depth of cut X Material constant =______	MRR	Power	Force	Velocity	Power
Limitation of lathe	Single tool in a time	Setting time	Idle time	all the above	all the above
Turret lathe machine is ________ capstan lathe	Larger than	Smaller than	Equals to	None of the above	Larger than
In turret lathe, turret head is directly mounted on	Ram	Saddle	Carriage	None of the above	Ram
In turret lathe, Saddle can be	Movale	Fixed	Rotates	None of the above	Movable
Turret lathe can handle __________ jobs	Heavier	Lighter	Moderate	None of the above	Heavier
Vibration induced in	Capstan lathe	Turret Lathe	Both a & b	None of the above	Capstan lathe
__________ lathe is very rigid	Capstan lathe	Turret Lathe	Centre lathe	None of the above	Turret lathe
Turret lathe can handle bars having dia upto	100 mm	200 mm	300 mm	400 mm	200 mm
In _________ lathe tool travel is limited	Capstan lathe	Turret Lathe	Both a & b	None of the above	Capstan lathe
Cross feeding can be possible in	Capstan lathe	Turret Lathe	Both a & b	None of the above	Turret Lathe
In capstan lathe, turret head is directly mounted on	Ram	Saddle	Carriage	None of the above	Saddle
In capstan lathe, Saddle can be	Movale	Fixed	Rotates	None of the above	Fixed
Turret capstan can handle __________ jobs	Heavier	Lighter	Moderate	None of the above	Lighter
In Turret lathe turret head is in ________ shape	Hexogonal	Circular	Both a & b	None of the above	Hexogonal
In capstan lathe turret head is in ________ shape	Hexogonal	Circular	Both a & b	None of the	Both a & b

				above	
Which one have high productivity and accuracy	Capstan lathe	Turret Lathe	Centre lathe	Automates	Automates

2 marks

1. What is swing diameter?
2. Write the specification of a typical lathe.
3. Write down the names of any four lathe accessories.
4. What is the application of air operated chuck?
5. Define the term "Concity".
6. Write down the formula for calculating taper turning angle by compound rest method.
7. Define the term „Thread catching‟.
8. Define automatic machine.
9. State the principal of multi spindle automats.
10. Classify multi spindle automats.

14 marks

1. Sketch a center lathe and mention various parts. .
2. List various type of feed mechanisms and explain briefly about tumbler gear reversing mechanism with a sketch.
3. Explain taper turning operation i n a lathe by a taper turning attachment .Discuss its advantages.
4. Explain the following methods of taper turning in a lathe.
 (i) By swiveling the compound rest. (ii) By a taper turning attachment.
5. Explain the Working principle of capstan and turret lathes.
6. Explain the tooling layout for the production of a Hexagonal bolt in a capstan lathe.
7. Discuss the tooling layout for the production of a Hexagonal nut in Turret lathe.
8. Classify transfer machines. Sketch and explain the working of Swiss type automatic screw machine. What are the advantages of automatic machines?
9. Describe a typical single spindle automatic chucking machine.
10. Describe a typical single spindle automatic bar machine
11. Differentiate between parallel action and progressive action multi -spindle automatics.

UNIT III

RECIPROCATING MACHINE TOOLS & MILLING MACHINES

3.1 Shapers

Shaping is performed on a machine tool called a shaper. The major components of a shaper are the ram, which has the tool post with cutting tool mounted on its face, and a worktable, which holds the part and accomplishes the feed motion.

A shaper is a type of machine tool that uses linear relative motion between the workpiece and a single-point cutting tool tomachine a linear toolpath. Its cut is analogous to that of a lathe, except that it is (archetypally) linear instead of helical. (Adding axes of motion can yield helical toolpaths, as also done in helical planing.) A shaper is analogous to a planer, but smaller, and with the cutter riding a ram that moves above a stationary workpiece, rather than the entire workpiece moving beneath the cutter. The ram is moved back and forth typically by a crank inside the column; hydraulically actuated shapers also exist.

3.2 Types of Shapers

Shapers are mainly classified as standard, draw-cut, horizontal, universal, vertical, geared, crank, hydraulic, contour and traveling head.[1] The horizontal arrangement is the most common. Vertical shapers are generally fitted with a rotary table to enable curved surfaces to be machined (same idea as in helical planing). The vertical shaper is essentially the same thing as a slotter (slotting machine), although technically a distinction can be made if one defines a true vertical shaper as a machine whose slide can be moved from the vertical. A slotter is fixed in the vertical plane.

Small shapers have been successfully made to operate by hand power. As size increases, the mass of the machine and its power requirements increase, and it becomes necessary to use a motor or other supply of mechanical power. This motor drives a mechanical arrangement (using a pinion gear, bull gear, and crank, or a chain over sprockets) or a hydraulic motor that supplies the necessary movement via hydraulic cylinders.

The workpiece mounts on a rigid, box-shaped table in front of the machine. The height of the table can be adjusted to suit this workpiece, and the table can traverse sideways underneath the reciprocating tool, which is mounted on the ram. Table motion may be controlled manually, but is usually advanced by an automatic feed mechanism acting on the feedscrew.

The ram slides back and forth above the work. At the front end of the ram is a vertical tool slide that may be adjusted to either side of the vertical plane along the stroke axis. This tool-slide holds the *clapper box* and toolpost, from which the tool can be positioned to cut a straight, flat surface on the top of the workpiece. The tool-slide permits feeding the tool downwards to deepen a cut. This adjustability, coupled with the use of specialized cutters and toolholders, enable the operator to cut internal and external gear tooth profiles, splines, dovetails, and keyways.

The most common use is to machine straight, flat surfaces, but with ingenuity and some accessories a wide range of work can be done. Other examples of its use are:

- Keyways in the boss of a pulley or gear can be machined without resorting to a dedicated broaching setup.
- Dovetail slides
- Internal splines and gear teeth.
- Keyway, spline, and gear tooth cutting in blind holes
- Cam drums with toolpaths of the type that in CNC milling terms would require 4- or 5-axis contouring or turn-mill cylindrical interpolation
- It is even possible to obviate wire EDM work in some cases. Starting from a drilled or cored hole, a shaper with a boring-bar type tool can cut internal features that don't lend themselves to milling or boring (such as irregularly shaped holes with tight corners).

3.3 Drilling and Reaming

Drilling and reaming operations

Drilling operation

Drilling is used to drill a round blind or through hole in a solid material. If the hole is larger than ~30 mm, its a good idea to drill a smaller pilot hole before core drilling the final one. For holes larger than ~50 mm, three-step drilling is recommended; v Core drilling is used to increase the diameter of an existing hole; v Step drilling is used to drill a stepped (multi-diameter) hole in a solid material;

Counterboring provides a stepped hole again but with flat and perpendicular relative to hole axis face. The hole is used to seat internal hexagonal bolt heads;

Countersinking is similar to counterboring, except that the step is conical for flat head screws:

Reaming provides a better tolerance and surface finish to an initially drilled hole. Reaming slightly increases the hole diameter. The tool is called reamer;

Center drilling is used to drill a starting hole to precisely define the location for subsequent drilling. The tool is called center drill. A center drill has a thick shaft and very short flutes. It is therefore very stiff and will not walk as the hole is getting started;

Gun drilling is a specific operation to drill holes with very large length-to-diameter ratio up to L/D ~300. There are several modifications of this operation but in all cases cutting fluid is delivered directly to the cutting zone internally through the drill to cool and lubricate the cutting edges, and to remove the chips (see Section 5.6 Cutting Fluids);

Drills and Reamers

Reamer

Twist drill

The twist drill does most of the cutting with the tip of the bit. It has two flutes to carry the chips up from the cutting edges to the top of the hole where they are cast off. The standard drill geometry

The typical helix angle of a general purpose twist drill is 18~30 degree, while the point angle (which equals two times the major cutting edge angle, see page 101) for the same drill is 118deg.

Some standard drill types are,

straight shank: this type has a cylindrical shank and is held in a chuck;

taper shank: his type is held directly in the drilling machine spindle.

Reamers

The reamer has similar geometry. The difference in geometry between a reamer and a twist drill are:

The reamer contains four to eight straight or helical flutes, respectively cutting edges.

The tip is very short and does not contain any cutting edges.

3.4 Boring

Boring is a process of producing circular internal profiles on a hole made by drilling or another process. It uses single point cutting tool called a boring bar. In boring, the boring bar can be rotated, or the workpart can be rotated. Machine tools which rotate the boring bar against a stationary workpiece are called boring machines (also boring mills). Boring can be accomplished on a turning machine with a stationary boring bar positioned in the tool post and rotating workpiece held in the lathe chuck as illustrated in the figure. In this section, we will consider only boring on boring machines.

Vertical Boring

Boring machines

Boring machines can be horizontal or vertical according to the orientation of the axis of rotation of the machine spindle. In horizontal boring operation, boring bar is mounted in a tool slide, which position is adjusted relative to the spindle face plate to machine different diameters. The boring bar must be supported on the other end when boring long and small-diameter holes. A vertical boring mill is used for large, heavy work parts with diameters up to 12 m. The typical boring mill can position and feed several cutting tools simultaneously. The work part is mounted on a rotating worktable.

Cutting tool for boring

The typical boring bar is shown in the figure. When boring with a rotating tool, size is controlled by changing the radial position of the tool slide, which holds the boring bar, with respect to the spindle axis of rotation. For finishing machining, the boring bar is additionally mounted in an adjustable boring head for more precise control of the bar radial position.

3.5 Tapping

A tap cuts a thread on the inside surface of a hole, creating a female surface which functions like a nut. The three taps in the image illustrate the basic types commonly used by most machinists:

Taps

Bottoming tap or plug taps

The tap illustrated in the top of the image has a continuous cutting edge with almost no taper — between 1 and 1.5 threads of taper is typical. This feature enables a bottoming tap to cut threads to the bottom of a blind hole. A bottoming tap is usually used to cut threads in a hole that has already been partially threaded using one of the more tapered types of tap; the tapered end ("tap chamfer") of a bottoming tap is too short to successfully start into an unthreaded hole. In the US, they are commonly known as bottoming taps, but in Australia and Britain they are also known as plug taps.

Intermediate tap, second tap, or plug tap

The tap illustrated in the middle of the image has tapered cutting edges, which assist in aligning and starting the tap into an untapped hole. The number of tapered threads typically ranges from 3 to 5.Plug taps are the most commonly used type of tap.[citation needed] In the US, they are commonly known as plug taps, whereas in Australia and Britain they are commonly known as second taps.

3.6 Milling

Milling is a process of producing flat and complex shapes with the use of multi-tooth cutting tool, which is called a milling cutter and the cutting edges are called teeth. The axis of rotation of the cutting tool is perpendicular to the direction of feed, either parallel or perpendicular to the machined surface. The machine tool that traditionally performs this operation is a milling machine. Milling is an interrupted cutting operation: the teeth of the milling cutter enter and exit the work during each revolution. This interrupted cutting action subjects the teeth to a cycle of impact force and thermal shock on every rotation. The tool material and cutter geometry must be designed to withstand these conditions. Cutting fluids are essential for most milling operations. Three types of feed in milling can be identified:

Feed per tooth: the basic parameter in milling equivalent to the feed in turning.

Feed per tooth is selected with regard to the surface finish and dimensional accuracy required. Feeds per tooth are in the range of 0.05~0.5 mm/tooth, lower feeds are for finishing cuts; feed per revolution: it determines the amount of material cut per one full revolution of the milling cutter. Feed per revolution is calculated as fr = fz being the number of the cutter's teeth;

Feed per minute fm: Feed per minute is calculated taking into account the rotational speed

N and number of the cutter's teeth z, fm = fzN = frN

Feed per minute is used to adjust the feed change gears.

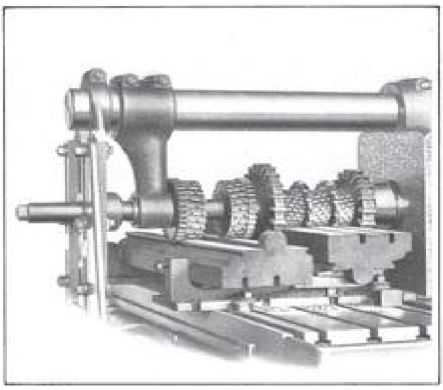
Three types of feed in milling can be identified:

Feed per tooth fz: the basic parameter in milling equivalent to the feed in turning.

Feed per tooth is selected with regard to the surface finish and dimensional accuracy required (see Section 5.10 Selection of Cutting Conditions). Feeds per tooth are in the range of 0.05~0.5 mm/tooth, lower feeds are for finishing cuts; feed per revolution fr: it determines the amount of material cut per one full revolution of the milling cutter. Feed per revolution is calculated as

fr = fz ,z being the number of the cutter's teeth;

Feed per minute fm: Feed per minute is calculated taking into account the rotational speed N and number of the cutter's teeth z, fm = fzN = fr,NFeed per minute is used to adjust the feed change gears. In down milling, the cutting force is directed into the work table, which allows thinner workparts tobe machined. Better surface finish is obtained but the stress load on the teeth is abrupt, which may damage the cutter.In up milling, the cutting force tend to lift the workpiece. The work conditions for the cutter are more favourable. Because the cutter does not start to cut when it makes contact (cutting at zero cut is impossible), the surface has a natural waviness.



Milling Operations

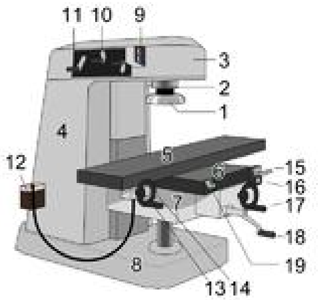
Owing to the variety of shapes possible and its high production rates, milling is one of the most versatile and widely used machining operations. The geometric form created by milling fall into three major groups: Plane surfaces: the surface is linear in all three dimensions. The simplest and most convenient type of surface;

Two-dimensional surfaces: the shape of the surface changes in the direction of two of the axes and is linear along the third axis. Examples include cams;

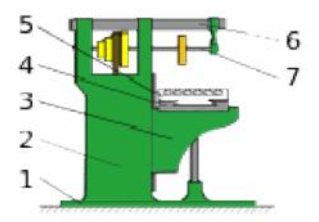
Three-dimensional surfaces: the shape of the surface changes in all three directions.

Examples include die cavities, gas turbine blades, propellers, casting patterns, etc.

Milling machines



Vertical milling machine



Horizontal milling machine

The conventional milling machines provide a primary rotating motion for the cutter held in the spindle, and a linear feed motion for the workpiece, which is fastened onto the worktable. Milling machines for machining of complex shapes usually provide both a rotating primary motion and a curvilinear feed motion for the cutter in the spindle with a stationary workpiece. Various machine designs are available for various milling operations. In this section we discuss only the most popular ones, classified into the following types:

Column-and-knee milling machines; v Bed type milling machines;

Machining centers.

Column-and-knee milling machines

The column-and-knee milling machines are the basic machine tool for milling. The name comes from the fact that this machine has two principal components, a column that supports the spindle, and a knee that supports the work table. There are two different types of column-and-knee milling machines according to position of the spindle axis:

horizontal, and vertical.

Milling cutters

Brazed cutters: Very limited numbers of cutters (mainly face mills) are made with brazed carbide inserts. This design is largely replaced by mechanically attached cutters.

Mechanically attached cutters: The vast majority of cutters are in this category. Carbide inserts are either clamped or pin locked to the body of the milling cutter.

Classification of milling cutters may also be associated with the various milling operations

3.7 Gear

Gears can be manufactured by most of manufacturing processes discussed so far (casting, forging, extrusion, powder metallurgy, blanking). But as a rule, machining is applied to achieve the final dimensions, shape and surface finish in the gear. The initial operations that produce a semi finishing part ready for gear machining as referred to as blanking operations; the starting product in gear machining is called a gear blank.

Two principal methods of gear manufacturing include

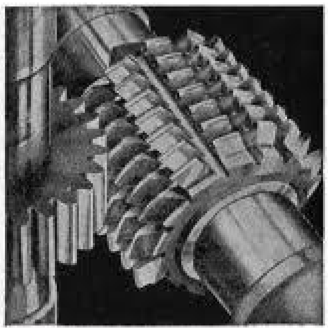
Gear forming, and Gear generation.

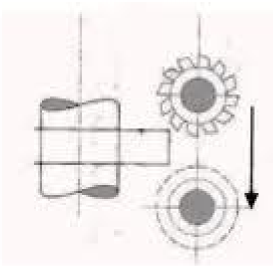
Each method includes a number of machining processes, the major of them included in this section.

Gear forming

In gear form cutting, the cutting edge of the cutting tool has a shape identical with the shape of the space between the gear teeth.

Two machining operations, milling and broaching can be employed to form cut gear teeth





3.8 Gear milling

In form milling, the cutter called a form cutter travels axially along the length of the gear tooth at the appropriate depth to produce the gear tooth. After each tooth is cut, the cutter is withdrawn, the gear blank is rotated (indexed), and the cutter proceeds to cut another tooth. The process continues until all teeth are cut.

Each cutter is designed to cut a range of tooth numbers. The precision of the form-cut tooth profile depends on the accuracy of the cutter and the machine and its stiffness. In form milling, indexing of the gear blank is required to cut all the teeth. Indexing is the process of evenly dividing the circumference of a gear blank into equally spaced divisions. The index head of the indexing fixture is used for this purpose.

The index fixture consists of an index head (also dividing head, gear cutting attachment) and footstock, which is similar to the tailstock of a lathe. The index head and footstock attach to the worktable of the milling machine. An index plate containing graduations is used to control the rotation of the index head spindle. Gear blanks are held between centers by the index head spindle and footstock. Workpieces may also be held in a chuck mounted to the index head spindle or may be fitted directly into the taper spindle recess of some indexing fixtures.

3.9 Gear hobbing

Gear hobbing is a machining process in which gear teeth are progressively generated by a series of cuts with a helical cutting tool (hob). All motions in hobbing are rotary, and the hob and gear blank rotate continuously as in two gears meshing until all teeth are cut when bobbing a spur gear, the angle between the hob and gear blank axes is 90° minus the lead angle at the hob threads. For helical gears, the hob is set so that the helix angle of the hob is parallel with the tooth direction of the gear being cut. Additional movement along the tooth length is necessary in order to cut the whole tooth length: The action of the hobbing machine (also gear hobber) is shown in the figures. The cutting of a gear by means of a hob is a continuous operation. The hob and the gear blank are connected by a proper gearing so that they rotate in mesh. To start cutting a gear, the rotating hob is fed inward until the proper setting for tooth depth is achieved, then cutting continues until the entire gear is finished.

The gear hob is a formed tooth milling cutter with helical teeth arranged like the thread on a screw. These teeth are fluted to produce the required cutting edges.

3.10 Shaping with a pinion-shaped cutter

This modification of the gear shaping process is defined as a process for generating gear teeth by a rotating and reciprocating pinion-shaped cutter:

The cutter axis is parallel to the gear axis. The cutter rotates slowly in timed relationship with the gear blank at the same pitch-cycle velocity, with an axial primary reciprocating motion; to produce the gear teeth. A train of gears provides the required relative motion between the cutter shaft and the gear-blank shaft. Cutting may take place either at the down stroke or upstroke of the machine. Because the clearance required for cutter travel is small, gear shaping is suitable for gears that are located close to obstructing surfaces such as flanges. The tool is called gear cutter and resembles in shape the mating gear from the conjugate gear pair, the other gear being the blank.

Gear shaping is one of the most versatile of all gear cutting operations used to produce internal gears, external gears, and integral gear-pinion arrangements. Advantages of gear shaping with pinion-shaped cutter are the high dimensional accuracy achieved and the not too expensive tool. The process is applied for finishing operation in all types of production rates.

3.11 Finishing operations

As produced by any of the process described, the surface finish and dimensional accuracy may not be accurate enough for certain applications. Several finishing operations are available, including the conventional process of shaving, and a number of abrasive operations, including grinding, honing, and lapping.

MCQ

Questions	Option A	Option B	Option C	Option D	Ans
Planer is _____ Machine	Heavier	Lighter	Moderate	None of the above	Heavier
Planer requires ________ floor area	More	Less	Moderate	None of the above	More
____________ cutting tool is used in shaper and planer	Single point	two point	multi point	None of the above	Single point
In planer ______ reciprocates horizontally	Work	Tool	Table	All of these	Work
In planer ________ is stationary	Work	Tool	Both a & b	All of these	Tool
Flat thin work in held on planer by	C-clamp	Toe dogs and stops	Vise	Die	Toe dogs and stops
Size of planer is specified	Size of table and height of cross rail	Size of table	Size of length	None of the above	Size of table and height of cross rail
____________ is used for machinig large workpieces	Planer	Shaper	Both a & b	None of the above	Planer
Which is costlier?	Planer	Shaper	Both a & b	None of the above	Planer
Planer employed _____ tool	Several	Single	Both a & b	None of the above	Several
Shaper is _____ Machine	Heavier	Lighter	Moderate	None of the above	Lighter
Shaper requires ________ floor area	More	Less	Moderate	None of the above	Less
In shaper ______ reciprocates horizontally	Work	Tool	Table	All of these	Tool
Quick return mechanism used in following machine	shaper	lathe	Milling machine	Grinding machine	shaper
Large jobs on shaper are held with the	Vise	Clamps,bolts,squares	Chuck	None of the	Clamps,bolts,squares

help of				above	
In shaper _________ is stationary	Work	Tool	Both a & b	All of these	Work
Size of shaper is specified	Length of stroke	Size of table	Motor	None of the above	Length of stroke
Which of the following is not the part of a shaper	Ram	Table	Cross slide	Tool zone	Cross slide
Shaper employed _____ tool	One	Two	Three	Four	One
Heavy cuts and feed possibe in	Planer	Shaper	Both a & b	None of the above	Planer
Clamping work is easy in	Planer	Shaper	Both a & b	None of the above	Shaper
Feeding of planer is done through	Friction mechanism	Hydraulic drive	Electrical drive	All of these	All of these
________ is used in tool room	Universal shaper	Standard shaper	Both a & b	None of the above	Universal shaper
Table of the shaper is made up of __________	Cast iron	steel	ceramics	Aluminium	Cast iron
In slotter tool moves	Horizontally	Vertically	Fixed	None of the above	Vertically
In slotter stroke of the ram is _________	More	Less	Moderate	None of the above	Less
Large amount of metal is removed in heavy material is ________	Puncher slotter	Tool room slotter	General slotters	All of these	Puncher slotter
_________ is used for more accuracy works	Puncher slotter	Tool room slotter	General slotters	All of these	Tool room slotter
Slotter can be used for cutting	Internal grooves	Internal gears	Recesses	All of these	All of these
Slotter is also called as __________ axis shaper	Horizontal	Vertical	Both a & b	None of the above	Vertical
Milling machine have_________ cutting tool	Single point	Multi point	Both a & b	None of the above	Multi point

The cutting tool in a milling machine is mounted on	Tool holder	Arbor	Spindle	Table	Arbor
Standard milling arbor size is	31.75mm	27mm	52mm	87mm	31.75mm
________ used to support saddle table and clamping devices	Base	Knee	Arbor	All of these	Knee
Irregular shapes are machined by	Plain milling	Face Milling	Angular milling	Form Milling	Form Milling
The opeartion of milling two sides of a workpiece simultaneously is called	Gang milling	Straddle milling	End milling	Many milling	Straddle milling
More than two cutters are used in	Gang milling	Straddle milling	End milling	Many milling	Gang milling
________ is used to make grooves narrow slots and keyways	Gang milling	Straddle milling	End milling	Many milling	End milling
Thickness of layer of material removed from the workpiece in a pass is called	Speed	Feed	Depth of cut	None of the above	Depth of cut
Single or double angle cutter is used in	Plain milling	Face Milling	Angular milling	Form Milling	Angular milling
In _______ milling cutter rotation and feed are in opposite direction	Up milling	Down milling	Both a & b	None of the above	Up milling
In _______ milling cutter rotation and feed are in same direction	Up milling	Down milling	Both a & b	None of the above	Down milling
Chips are thrown out in ________ process	Up milling	Down milling	Both a & b	None of the above	Up milling
Chips are fall downt in ________ process	Up milling	Down milling	Both a & b	None of the above	Down milling
_________ have good surface finish	Up milling	Down milling	Both a & b	None of the above	Down milling
Tool wear is more in	Up milling	Down milling	Both a & b	None of the above	Up milling
________ is used for thick sheets	Up milling	Down milling	Both a & b	None of the above	Up milling

_________ is used for thin sheets	Up milling	Down milling	Both a & b	None of the above	Down milling
High MRR is possible in	Up milling	Down milling	Both a & b	None of the above	Down milling
Backlash occurs in	Up milling	Down milling	Both a & b	None of the above	Down milling
Rotating worm cutter is used in	Gear shaper	Gear planning	Gear hobbing	None of the above	Gear Hobbibg
In gear hobbing the production rate is	High	Low	Moderate	None of the above	High
Hobbing is applicable for	Ferrous	Non ferrous	Non metals	All of these	All of these

2 marks

1. Write down any four operations performed by a shaper.
2. Define feed and depth of cut.
3. What is the function of clapper block in a planner?
4. What are the differences between up milling and down milling?
5. Define “Face milling “.
6. Write down the rule for gear ratio in differential indexing.
7. How do you specify radial drilling machine.
8. Write down any four operations performed by a drilling machine.
9. What is meant by “Sensitive hand feed”?
10. Calculate the tap drill size to cut an internal thread for bolt of outside diameter 10mm, pitch 1.5mm and depth of the thread 0.61 pitch.

14 marks

1. With a simple sketch, explain the working of the crank and slotted link quick return motion mechanism used in shaper.
2. Write down any four differences between shaper and planer. .
3. Explain the Working principle of planer with a neat sketch.
4. How do you specify a planer?
5. Describe the working mechanism of a universal dividing head, with neat diagram.
6. With a neat sketch, indicate the various parts of an arbor assembly.
7. With a simple sketch, explain the principal parts and angles of a plain milling cutter .Explain them.
8. Explain the twist drill nomenclature and define various elements of twist drill.
9. With a simple sketch, explain the working of a vertical boring machine.
10. Explain the counter boring and counter ringing operation.
11. Explain the Working principle of a Jig boring machine with a neat sketch.

UNIT IV

OTHER MACHINE TOOLS

4.1 Abrasive Processes

Abrasive machining processes can be divided into two categories based on how the grains are applied to the workpiece.

In bonded abrasive processes, the particles are held together within a matrix, and their combined shape determines the geometry of the finished workpiece. For example, in grinding the particles are bonded together in a wheel. As the grinding wheel is fed into the part, its shape is transferred onto the workpiece.

In loose abrasive processes, there is no structure connecting the grains. They may be applied without lubrication as dry powder, or they may be mixed with a lubricant to form a slurry. Since the grains can move independently, they must be forced into the workpiece with another object like a polishing cloth or a lapping plate.

Common abrasive processes are listed below.

Fixed (bonded) abrasive processes

- Grinding
- Honing, superfinishing
- Tape finishing, abrasive belt machining
- Buffing, brushing
- Abrasive sawing, Diamond wire cutting, Wire saw
- Sanding

Loose abrasive processes

- Polishing
- Lapping
- Abrasive flow machining (AFM)
- Hydro-erosive grinding
- Water-jet cutting
- Abrasive blasting
- Mass finishing,

4.2 Grinding Wheels

A grinding wheel is an expendable wheel that is composed of an abrasive compound used for various grinding (abrasive cutting) and abrasive machining operations. They are used in grinding machines.

The wheels are generally made from a matrix of coarse particles pressed and bonded together to form a solid, circular shape. Various profiles and cross sections are available depending on the

intended usage for the wheel. They may also be made from a solid steel or aluminium disc with particles bonded to the surface.

The manufacture of these wheels is a precise and tightly controlled process, due not only to the inherent safety risks of a spinning disc, but also the composition and uniformity required to prevent that disc from exploding due to the high stresses produced on rotation.

There are five characteristics of a cutting wheel: material, grain size, wheel grade, grain spacing, and bond type. They will be indicated by codes on the wheel's label.

Abrasive Grain, the actual abrasive, is selected according to the hardness of the material being cut.

- Aluminum Oxide (A)
- Silicon Carbide (S)
- Ceramic (C)
- Diamond (D, MD, SD)
- Cubic Boron Nitride (B)

Grinding wheels with diamond or Cubic Boron Nitride (CBN) grains are called superabrasives. Grinding wheels with Aluminum Oxide (corundum), Silicon Carbide or Ceramic grains are called conventional abrasives.

Grain size, from 8 (coarsest) 1200 (finest), determines the physical size of the abrasive grains in the wheel. A larger grain will cut freely, allowing fast cutting but poor surface finish. Ultra-fine grain sizes are for precision finish work.

Wheel grade, from A (soft) to Z (hard), determines how tightly the bond holds the abrasive. Grade affects almost all considerations of grinding, such as wheel speed, coolant flow, maximum and minimum feed rates, and grinding depth.

Grain spacing, or structure, from 1 (densest) to 16 (least dense). Density is the ratio of bond and abrasive to air space. A less-dense wheel will cut freely, and has a large effect on surface finish. It is also able to take a deeper or wider cut with less coolant, as the chip clearance on the wheel is greater.

Wheel bond, how the wheel holds the abrasives, affects finish, coolant, and minimum/maximum wheel speed.

- Vitrified (V)
- Resinoid (B)
- Silicate (S)

- Shellac (E)
- Rubber (R)
- Metal (M)
- Oxychloride (O)

4.3 Types of Grinding Processes

Straight wheel

Straight wheel

To the right is an image of a straight wheel. These are by far the most common style of wheel and can be found on bench or pedestal grinders. They are used on the periphery only and therefore produce a slightly concave surface (*hollow ground*) on the part. This can be used to advantage on many tools such as chisels.

Straight Wheels are generally used for cylindrical, centreless, and surface grinding operations. Wheels of this form vary greatly in size, the diameter and width of face naturally depending upon the class of work for which is used and the size and power of the grinding machine.

Cylinder or wheel ring

Cylinder wheels provide a long, wide surface with no center mounting support (hollow). They can be very large, up to 12" in width. They are used only in vertical or horizontal spindle grinders. Cylinder or wheel ring is used for producing flat surfaces, the grinding being done with the end face of the wheel.

Tapered wheel

A straight wheel that tapers outward towards the center of the wheel. This arrangement is stronger than straight wheels and can accept higher lateral loads. Tapered face straight wheel is primarily used for grinding thread, gear teeth etc.

Straight cup

Straight cup wheels are an alternative to cup wheels in tool and cutter grinders, where having an additional radial grinding surface is beneficial.

Dish cup

A very shallow cup-style grinding wheel. The thinness allows grinding in slots and crevices. It is used primarily in cutter grinding and jig grinding.

Saucer wheel

A special grinding profile that is used to grind milling cutters and twist drills. It is most common in non-machining areas, as saw filers use saucer wheels in the maintenance of saw blades.

Diamond wheels

Diamond wheel

Diamond wheels are grinding wheels with industrial diamonds bonded to the periphery.

They are used for grinding extremely hard materials such as carbide cutting tips, gemstones or concrete. The saw pictured to the right is a slitting saw and is designed for slicing hard materials, typically gemstones.

Mounted points

Mounted points are small grinding wheels bonded onto a mandrel. Diamond mounted points are tiny diamond rasps for use in a jig grinder doing profiling work in hard material. Resin and vitrified bonded mounted points with conventional grains are used for deburring applications, especially in the foundry industry.

Cut off wheels

Cut off wheels, also known as *parting wheels*, are self-sharpening wheels that are thin in width and often have radial fibres reinforcing them. They are often used in the construction industry for cutting reinforcement bars (rebar), protruding bolts or anything that needs quick removal or trimming. Most handymen would recognise an angle grinder and the discs they use.

4.4 Cylindrical grinding

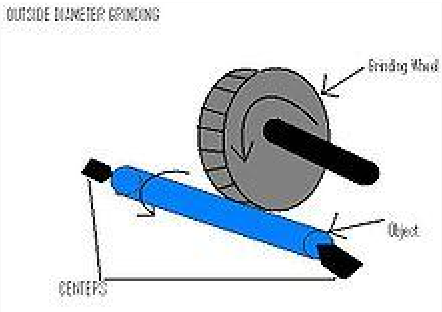
The cylindrical grinder is a type of grinding machine used to shape the outside of an object. The cylindrical grinder can work on a variety of shapes; however the object must have a central axis of rotation. This includes but is not limited to such shapes as a cylinder, an ellipse, a cam, or a crankshaft.

Cylindrical grinding is defined as having four essential actions:

1. The work (object) must be constantly rotating
2. The grinding wheel must be constantly rotating
3. The grinding wheel is fed towards and away from the work
4. Either the work or the grinding wheel is traversed with respect to the other.

While the majority of cylindrical grinders employ all four movements, there are grinders that only employ three of the four actions.

There are five different types of cylindrical grinding: outside diameter (OD) grinding, inside diameter (ID) grinding, plunge grinding, creep feed grinding, and centerless grinding.



A basic overview of Outside Diameter Cylindrical Grinding. The Curved Arrows refer to direction of rotation

4.5 Outside Diameter Grinding

OD grinding is grinding occurring on external surface of an object between the centers. The centers are end units with a point that allow the object to be rotated. The grinding wheel is also being rotated in the same direction when it comes in contact with the object. This effectively means the two surfaces will be moving opposite directions when contact is made which allows for a smoother operation and less chance of a jam up.

Plunge grinding

A form of OD grinding, however the major difference is that the grinding wheel makes continuous contact with a single point of the object instead of traversing the object.

Creep feed grinding

Creep Feed is a form of grinding where a full depth of cut is removed in a single pass of the wheel. Successful operation of this technique can reduce manufacturing time by 50%, but often the grinding machine being used must be designed specifically for this purpose. This form occurs in both cylindrical and

Surface Grinding

Surface grinding is used to produce a smooth finish on flat surfaces. It is a widely used abrasive machining process in which a spinning wheel covered in rough particles (grinding wheel) cuts chips of metallic or nonmetallic substance from a workpiece, making a face of it flat or smooth.

Surface grinding is the most common of the grinding operations. It is a finishing process that uses a rotating abrasive wheel to smooth the flat surface of metallic or nonmetallic materials to give them a more refined look or to attain a desired surface for a functional purpose.

The surface grinder is composed of an abrasive wheel, a workholding device known as a chuck, and a reciprocating or rotary table. The chuck holds the material in place while it is being worked on. It can do this one of two ways: ferromagnetic pieces are held in place by a magnetic chuck, while non-ferromagnetic and nonmetallic pieces are held in place by vacuum or mechanical means. A machine vise (made from ferromagnetic steel or cast iron) placed on the magnetic chuck can be used to hold non-ferromagnetic workpieces if only a magnetic chuck is available.

Factors to consider in surface grinding are the material of the grinding wheel and the material of the piece being worked on.

Typical workpiece materials include cast iron and mild steel. These two materials don't tend to clog the grinding wheel while being processed. Other materials are aluminum, stainless steel, brass and some plastics. When grinding at high temperatures, the material tends to become weakened and is more inclined to corrode. This can also result in a loss of magnetism in materials where this is applicable.

The grinding wheel is not limited to a cylindrical shape and can have a myriad of options that are useful in transferring different geometries to the object being worked on. Straight wheels can be dressed by the operator to produce custom geometries. When surface grinding an object, one must keep in mind that the shape of the wheel will be transferred to the material of the object like a mirror image.

Spark out is a term used when precision values are sought and literally means "until the sparks are out (no more)". It involves passing the workpiece under the wheel, without resetting the depth of cut, more than once and generally multiple times. This ensures that any inconsistencies in the machine or workpiece are eliminated.

A surface grinder is a machine tool used to provide precision ground surfaces, either to a critical size or for the surface finish.

The typical precision of a surface grinder depends on the type and usage, however +/- 0.002 mm (+/- 0.0001") should be achievable on most surface grinders.

The machine consists of a table that traverses both longitudinally and across the face of the wheel. The longitudinal feed is usually powered by hydraulics, as may the cross feed, however any mixture of hand, electrical or hydraulic may be used depending on the ultimate usage of the machine (i.e.: production, workshop, cost). The grinding wheel rotates in the spindle head and is also adjustable for height, by any of the methods described previously. Modern surface grinders are semi-automated, depth of cut and spark-out may be preset as to the number of passes and, once set up, the machining process requires very little operator intervention.

Depending on the workpiece material, the work is generally held by the use of a magnetic chuck. This may be either an electromagnetic chuck, or a manually operated, permanent magnet type chuck; both types are shown in the first image.

The machine has provision for the application of coolant as well as the extraction of metal dust (metal and grinding particles).

Types of surface grinders

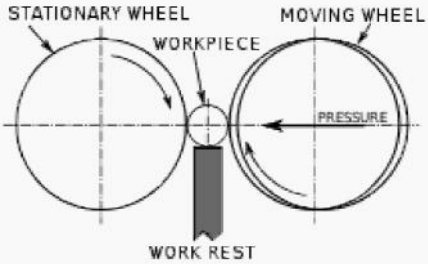
Horizontal-spindle (peripheral) surface grinders**.** The periphery (flat edge) of the wheel is in contact with the workpiece, producing the flat surface. Peripheral grinding is used in high-precision work on simple flat surfaces; tapers or angled surfaces; slots; flat surfaces next to shoulders; recessed surfaces; and profiles.

Vertical-spindle (wheel-face) grinders**.** The face of a wheel (cup, cylinder, disc, or segmental wheel) is used on the flat surface. Wheel-face grinding is often used for fast material removal, but some machines can accomplish high-precision work. The workpiece is held on a reciprocating table, which can be varied according to the task, or a rotary-table machine, with continuous or indexed rotation. Indexing allows loading or unloading one station while grinding operations are being performed on another.

Disc grinders and double-disc grinders. Disc grinding is similar to surface grinding, but with a larger contact area between disc and workpiece. Disc grinders are available in both vertical and horizontal spindle types. Double disc grinders work both sides of a workpiece simultaneously. Disc grinders are capable of achieving especially fine tolerances.

4.6 Centerless grinding

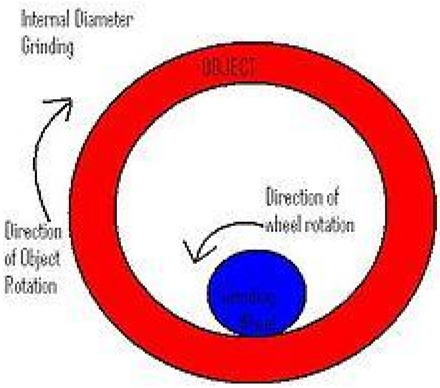
Centerless cylindrical grinder



A schematic of the centerless grinding process.

Centerless grinding is a form of grinding where there is no collet or pair of centers holding the object in place. Instead, there is a regulating wheel positioned on the opposite side of the object to the grinding wheel. A work rest keeps the object at the appropriate height but has no bearing on its rotary speed. The workblade is angled slightly towards the regulating wheel, with the workpiece centerline above the centerlines of the regulating and grinding wheel; this means that high spots do not tend to generate corresponding opposite low spots, and hence the roundness of parts can be improved. Centerless grinding is much easier to combine with automatic loading procedures than centered grinding; throughfeed grinding, where the regulating wheel is held at a slight angle to the part so that there is a force feeding the part through the grinder, is particularly efficient.

4.7 Internal Grinding



A basic overview of Internal Diameter Cylindrical Grinding. The Curved Arrows refer to direction of rotation.

ID grinding is grinding occurring on the inside of an object. The grinding wheel is always smaller than the width of the object. The object is held in place by a collet, which also rotates the object in place. Just as with OD grinding, the grinding wheel and the object rotated in opposite directions giving reversed direction contact of the two surfaces where the grinding occurs.

4.8 Concepts of surface Integrity

Surface integrity is the surface condition of a workpiece after being modified by a manufacturing process. The surface integrity of a workpiece or item changes the material's properties. The consequences of changes to surface integrity are a mechanical engineering design problem, but the preservation of those properties are a manufacturing consideration.

Surface integrity can have a great impact on a parts function; for example, Inconel 718 can have a fatigue limit as high as 540 MPa (78,000 psi) after a gentlegrinding or as low as 150 MPa (22,000 psi) after electrical discharge machining (EDM).

There are two aspects to surface integrity: topography characteristics and surface layer characteristics. The topography is made up of surface roughness, waviness, errors of form, and flaws. The surface layer characteristics that can change through processing are: plastic deformation, residual stresses, cracks, hardness, overaging, phase changes, recrystallization, intergranular attack, and hydrogen embrittlement. When a traditional manufacturing process is used, such as machining, the surface layer sustains local plastic deformation.

The processes that affect surface integrity can be conveniently broken up into three classes: traditional processes, non-traditional processes, and finishing treatments. Traditional processes are defined as processes where the tool contacts the workpiece surface; for example: grinding, turning, and machining. These processes will only damage the surface integrity if the improper parameters are used, such as dull tools, too high feed speeds, improper coolant or lubrication, or incorrect grinding wheel hardness. Nontraditional processes are defined as processes where the tool does not contact the workpiece; examples of this type of process include EDM, electrochemical machining, and chemical milling. These processes will produce different surface integrity depending on how the processes are controlled; for instance, they can leave a stress-free surface, a remelted surface, or excessive surface roughness. Finishing treatments are defined as processes that negate surface finishes imparted by traditional and non-traditional processes or improve the surface integrity. For example, compressive residual stress can be enhanced via peening or roller burnishing or the recast layer left by EDMing can be removed via chemical milling.

Finishing treatments can affect the workpiece surface in a wide variety of manners. Some clean and/or remove defects, such as scratches, pores, burrs, flash, or blemishes. Other processes improve or modify the surface appearance by improving smoothness, texture, or color. They can also improve corrosion resistance,wear resistance, and/or reduce friction. Coatings are another type of finishing treatment that may be used to plate an expensive or scarce material onto a less expensive base material.

Variables

Manufacturing processes have five main variables: the workpiece, the tool, the machine tool, the environment, and process variables. All of these variables can affect the surface integrity of the workpiece by producing:

- High temperatures involved in various machining processes
- Plastic deformation in the workpiece (residual stresses)
- Surface geometry (roughness, cracks, distortion)
- Chemical reactions, especially between the tool and the workpiece

4.9 Broaching Machines

- Broaching machines are relatively simple as they only have to move the broach in a linear motion at a predetermined speed and provide a means for handling the broach automatically. Most machines are hydraulic, but a few specialty machines are mechanically driven. The machines are distinguished by whether their motion is horizontal or vertical. The choice of machine is primarily dictated by the stroke required. Vertical broaching machines rarely have a stroke longer than 60 in (1.5 m).

- Vertical broaching machines can be designed for push broaching, pull-down broaching, pull-up broaching, or surface broaching. Push broaching machines are similar to an arbor press with a guided ram; typical capacities are 5 to 50 tons. The two ram pull-down machine is the most common type of broaching machine. This style machine has the rams under the table. Pull-up machines have the ram above the table; they usually have more than one ram. Most surface broaching is done on a vertical machine.
- Horizontal broaching machines are designed for pull broaching, surface broaching, continuous broaching, and rotary broaching. Pull style machines are basically vertical machines laid on the side with a longer stroke. Surface style machines hold the broach stationary while the workpieces are clamped into fixtures that are mounted on a conveyor system. Continuous style machines are similar to the surface style machines except adapted for internal broaching.
- Horizontal machines used to be much more common than vertical machines, however today they represent just 10% of all broaching machines purchased. Vertical machines are more popular because they take up less space.

4.10 Push Type Broaching Machine

Vertical internal push-down: Vertical push-down machines are often nothing more than general-purpose hydraulic presses with special fixtures. They are available with capacities of 2 to 25 tons, strokes up to 36" and speeds as high as 40 FPM. In some cases, universal machines have been designed which combine as many as three different broaching operations, such as push, pull, and surface, simply through the addition of special fixtures.

4.11 Pull Type Broaching Machine

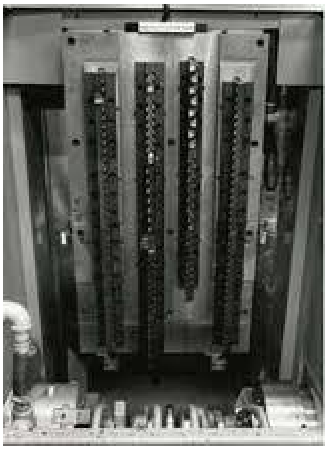
Vertical internal pull-up: The pull-up type, in which the workpiece is placed below the worktable, was the first to be introduced. Its principal use is in broaching round and irregular shaped holes. Pull-up machines are now furnished with pulling capacities of 6 to 50 tons, strokes up to 72", and broaching speeds of 30 FPM. Larger machines are available; some have electro-mechanical drives for greater broaching speed and higher productivity.

Vertical internal pull-down: The more sophisticated pull-down machines, in which the work is placed on top of the table, were developed later than the pull-up type. These pull-down machines are capable of holding internal shapes to closer tolerances by means of locating fixtures on top of the worktable. Machines come with pulling capacities of 2 to 75 tons, 30" to 110" strokes, and speeds of up to 80 FPM.

4.12 Surface broaches

The broaches used to remove material from an external surface are commonly known as surface broaches. Such broaches are passed over the workpiece surface to be cut, or the workpiece passes over the tool on horizontal, vertical or chain machines to produce flat or contoured surfaces.

While some surface broaches are of solid construction, most are of built-up design, with sections, inserts or indexable tool bits that are assembled end-to-end in a broach holder or sub holder. The holder fits on the machine slide and provides rigid alignment and support.



4.13 Continuous Chain Broaching

Continuous chain, or simply chain broaching refers to the type of machine that is used to broach a piece part.

Chain broaching is oriented towards high volume production, and is an extremely fast and efficient operation. However, because the fixtures used to hold the piece parts are mounted on chains that are driven by sprockets, it is difficult to hold extremely close tolerances. This process is suitable for high-volume, external cutting.

Continuous Chain Broaching Industries

- Biomedical
- Electronics
- Defense

A chain broaching machine resembles a very long tunnel, through which passes a series of holding fixtures, or cars. Piece parts are loaded, usually automatically, into the cars, which themselves are mounted on, and carried through the tunnel by a very large continuous chain. The broach tooling is mounted on the inside walls of the tunnel, and this tooling cuts the piece part as it passes through the tunnel. Contact us today to learn more.

MCQ

Questions	Option A	Option B	Option C	Option D	Ans
Trepannings operation is done by	Drilling machine	Lathe	Milling machine	Grinding	Drilling machine
Reamer tool is used to	Finishing the holes	Produce hoels	Produce threads	Cleaning	Finishing the holes
Process of enlarging the hole is called	Boring	Reaming	Drilling	Non of these	Boring
0.0025 mm accuarcy obtained from	Horizontal boring machine	Jig boring machine	Vertical boring machine	Milling machine	Jig boring machine
A twist drill is a	Side cutting tool	Front cutting tool	End cutting tool	None of these	End cutting tool
The cutting speed for drilling ___________ with high speed steel drills is 24 to 45 m/min.	Mild steel	Copper	Aluminium	Brass	Mild steel
_________ drilling machine is used for space restricted area	Portable	Sensitive	Upright	Gang	Portable
Sensitive drilling machine is used for ___________	Heavier work	Lighter work	Both a & b	None of these	Lighter work
Heavy large workpieces drilled by	Sensitive drilling machine	Upright drilling machine	Portable drilling machine	Radial drilling machine	Radial drilling machine
______ drilling machine is used in tool rooms and large scale die manufacturing	Portable	Sensitive	Upright	Radial	Radial
In __________ drilling several drilling spindles are mounted on a table	Portable	Sensitive	Upright	Gang	Gang
In gang drilling machine spindle speed and depth of cut	same	Varied	Both a & b	None of these	Varied

of each spindle will be					
Reaming, counter boring, tapping also possible in ______ drilling	Portable	Sensitive	Upright	Gang	Gang
Numerical control is available for_________ drilling	Turret	Sensitive	Upright	Gang	Turret
Two fixtures are available in _____ drilling	Turret	Sensitive	Upright	Gang	Turret
Cycle time is less in ______ drilling	Turret	Sensitive	Upright	Gang	Turret
High speed low feed drilling machine is called	Turret	Deep hole	Upright	Gang	Deep hole
Work holding devices of drilling machine	Vice	V block	Jigs	All of these	All of these
Enlargement of drilled hole partially for the specific length is called	Boring	Counter boring	Counter sinking	All of these	Counter boring
Forming a conical shape at the end of drilled hole is callled	Boring	Counter boring	Counter sinking	All of these	Counter sinking
Process of squaring the hole is called	Boring	Counter boring	Counter sinking	Spot facing	Spot facing
External threads formed on existing hole is called	Boring	Tapping	Counter sinking	Spot facing	Tapping
___________ is precise boring machine	Jig boring	Vertical boring	Horizontal boring	None of these	Jig boring
A fine grained grinding wheel is used to grind	Hard and brittle materials	Soft and ductile materials	Hard and ductile materials	Soft and brittle materials	Hard and brittle materials

The tool life in case of a grinding wheel is the time	Between two successive regrinds of the wheel	Taken for the wheel to be balanced	Taken between two successive wheel dressings	Taken for a wear of 1 mm on its diameter	Taken between two successive wheel dressings
A grinding wheel is said to be of __________ if it holds the abrasive grains more securely.	Soft grade	Medium grade	Hard grade	None of these	Hard grade
The grinding of long, slender shafts or bars is usually done by	In-feed grinding	Through feed grinding	End-feed grinding	Any one of these	Through feed grinding
The advantage of a broaching operation is that	Rate of production is very high	High accuracy and high class of surface finish is possible	Roughing and finishing cuts are completed in one pass of the tool	All of the above	All of the above
The silicon carbide abrasive is chiefly used for grinding	Cemented carbide	Ceramic	Cast iron	All of these	All of these
The grinding operation is a	Shaping operation	Forming operation	Surface finishing operation	Dressing operation	Surface finishing operation
Grinding wheels should be tested for balance	Only at the time of manufacture	Before starting the grinding operation	At the end of grinding operation	Occasionally	Occasionally
In grinding irregular, curved, tapered, convex and concave surfaces, the grinder used is	Cylindrical grinder	Internal grinder	Surface grinder	Tool and cutter grinder	Surface grinder
In grinding operation, for faster removal of material	Fine grain size is used	medium grain size is used	coarse grain size is used	None of the above	coarse grain size is used
In grinding operation, for grinding harder material	softer grade is used	high grade is used	medium grade is used	None of the above	softer grade is used
Crack is developed in grinding	slower speed	high speed	hard work	None of the	high speed

wheels because of				above	
In grinding operation, for grinding softer material	softer grade is used	high grade is used	medium grade is used	None of the above	high grade is used
The process of improving cutting action of grinding wheels is called	dressing operation	truing operation	cutting operation	buff operation	dressing operation
Grinding wheel is immersed with coolant in order to	remove chips	remove heat	clean the job	None of the above	remove heat
Choose the correct formula for machining time in grinding	s(n)(k)/l min	l(k)/s(n) min	s(k)/l(n) min	both 1&2	l(k)/s(n) min
Surface grinding is done to produce	Tapered surface	Flat surface	Internal cylindrical holes	All of these	Flat surface
The soft grade grinding wheels are denoted by the letters	A TO H	I TO P	Q TO Z	A TO P	A TO H
Special gears, bushing sleeves, compressor wheels are formed by	Broaching	Milling	Drilling	Grinding	Broaching
Clay bond which is brown and reddish colour	Vitrified	Silicate	Resinoid	All of these	Vitrified
_______________ bond is in light grey colour	Vitrified	Silicate	Resinoid	All of these	Silicate
___________ bond is used for finish work	Vitrified	Silicate	Resinoid	Shellac	Shellac
_____________ bond is most flexible	Vitrified	Silicate	Rubber	Shellac	Rubber
In grinding wheel specification frist letter denotes	Abrasive	Grain size	Grade	Structure	Abrasive
Honing is a ___________ process	Roughing	Finishing	Both a & b	None of these	Finishing
For high degree surfacefinish	Honing	Lapping	Both a & b	None of these	Lapping

__________ is preferred					
_____________ has good surface finish	Cylindrical grinding	Centreless grinding	Both a & b	None of these	Cylindrical grinding

2 marks

1. What is the process of self sharpening of the grinding wheel?
2. What are the four moments in a cylindrical centre type grinding.
3. What is meant by centerless grinding?
4. Define the terms abrasive grains.
5. What is meant by grit or grains size.
6. Define the term grade used i n grinding wheel.
7. What is open and dense structure?
8. What is meant by dressing and truing?
9. What is meant by honing?
10. What is super finishing?

14 marks

1. What are the various methods of centerless grinding and each briefly?
2. Explain the external cylindrical grinding process and surface grinding process.
3. Explain the vitrified and resinoid bonding process.
4. E xplain the operations of horizontal broaching machine with neat sketch.
5. Explain the gear cutting by a formed tool.
6. Differentiate between gear forming and generating.
7. Explain the principle of operation of gearing hobbing operation what are the advantages of gear hobbing.
8. Give advantages and limitations of gear hobbing.

UNIT V

CNC MACHINES

5.1 Numerical Control (NC) Machine Tools

Numerical Control (NC) refers to the method of controlling the manufacturing operation by means of directly inserted coded numerical instructions into the machine tool. It is important to realize that NC is not a machining method, rather, it is a concept of machine control. Although the most popular applications of NC are in machining, NC can be applied to many other operations, including welding, sheet metalworking, riveting, etc.

The major advantages of NC over conventional methods of machine control are as follows:

Higher precision

Machining of complex three-dimensional shapes

Better quality

Higher productivity

Multi-operational machining

Low operator qualification

5.2 Types of NC systems

Machine controls are divided into three groups,

Traditional numerical control (NC);

Computer numerical control (CNC);

Distributed numerical control (DNC).

The original numerical control machines were referred to as NC machine tool. They have "hardwired" control, whereby control is accomplished through the use of punched paper (or plastic) tapes or cards. Tapes tend to wear, and become dirty, thus causing misreadings. Many other problems arise from the use of NC tapes, for example the need to manual reload the NC tapes for each new part and the lack of program editing abilities, which increases the lead time. The end of NC tapes was the result of two competing developments, CNC and DNC.

CNC refers to a system that has a local computer to store all required numerical data. While CNC was used to enhance tapes for a while, they eventually allowed the use of other storage media, magnetic tapes and hard disks. The advantages of CNC systems include but are not limited to the possibility to store and execute a number of large programs (especially if a three or more dimensional machining of complex shapes is considered), to allow editing of programs, to execute cycles of machining commands, etc.

The development of CNC over many years, along with the development of local area networking, has evolved in the modern concept of DNC. Distributed numerical control is similar to CNC, except a remote computer is used to control a number of machines. An off-site mainframe host computer holds programs for all parts to be produced in the DNC facility. Programs are downloaded from the mainframe computer, and then the local controller feeds instructions to the hardwired NC machine.

The recent developments use a central computer which communicates with local CNC computers (also called Direct Numerical Control)

Controlled axes

NC system can be classified on the number of directions of motion they are capable to control simultaneously on a machine tool. Each free body has six degree of freedom, three positive or negative translations along x, y, and z-axis, and three rotations clockwise or counter clockwise about these axes. Commercial NC systems are capable of controlling simultaneously two, two and half, three, four and five degrees of freedom, or axes. The NC systems which control three linear translations (3-axis systems), or three linear translations and one rotation of the worktable (4-axis systems) are the most common.

Although the directions of axes for a particular machine tool are generally agreed as shown in the figure, the coordinate system origin is individual for each part to be machined and has to be decided in the very beginning of the process of CNC part programming.

Point-to-point vs. continuous systems

The two major types of NC systems are (see the figure):

Point-to-point (PTP) system, and

Contouring system.

PTP is a NC system, which controls only the position of the components. In this system, the path of the component motion relative to the workpiece is not controlled. The travelling between different positions is performed at the traverse speed allowable for the machine tool and following the shortest way.

Contouring NC systems are capable of controlling not only the positions but also the component motion, i.e., the travelling velocity and the programmed path between the desired positions.

Computer numerical control (CNC)

Numerical control (NC) is the automation of machine tools that are operated by precisely programmed commands encoded on a storage medium, as opposed to controlled manually via hand wheels or levers, or mechanically automated via cams alone. Most NC today is computer numerical control (CNC), in which computers play an integral part of the control.

In modern CNC systems, end-to-end component design is highly automated using computer-aided design (CAD) and computer-aided manufacturing (CAM) programs. The programs produce a computer file that is interpreted to extract the commands needed to operate a particular machine via a post processor, and then loaded into the CNC machines for production. Since any particular component might require the use of a number of different tools – drills, saws, etc., modern machines often combine multiple tools into a single "cell". In other installations, a number of different machines are used with an external controller and human or robotic operators that move the component from machine to machine. In either case, the series of steps needed to produce any part is highly automated and produces a part that closely matches the original CAD design.

The first NC machines were built in the 1940s and 1950s, based on existing tools that were modified with motors that moved the controls to follow points fed into the system on punched tape. These early servomechanisms were rapidly augmented with analog and digital computers, creating the modern CNC machine tools that have revolutionized the machining processes.

Modern CNC mills differ little in concept from the original model built at MIT in 1952. Mills typically consist of a table that moves in the X and Y axes, and a tool spindle that moves in the Z (depth). The position of the tool is driven by motors through a series of step-down gears in order to provide highly accurate movements, or in modern designs, direct-drive stepper motor or servo motors. Open-loop control works as long as the forces are kept small enough and speeds are not too great. On commercial metalworking machines closed loop controls are standard and required in order to provide the accuracy, speed, and repeatability demanded.

As the controller hardware evolved, the mills themselves also evolved. One change has been to enclose the entire mechanism in a large box as a safety measure, often with additional safety interlocks to ensure the operator is far enough from the working piece for safe operation. Most new CNC systems built today are completely electronically controlled.

CNC-like systems are now used for any process that can be described as a series of movements and operations. These include laser cutting, welding, friction stir welding, ultrasonic welding, flame and plasma cutting, bending, spinning, hole-punching, pinning, gluing, fabric cutting, sewing, tape and fiber placement, routing, picking and placing (PnP), and sawing.

Mills

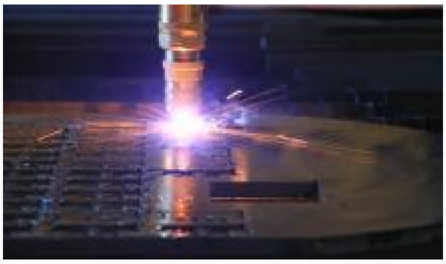
CNC mills use computer controls to cut different materials. They are able to translate programs consisting of specific number and letters to move the spindle to various locations and depths. Many use G-code, which is a standardized programming language that many CNC machines understand, while others use proprietary languages created by their manufacturers. These proprietary languages while often simpler than G-code are not transferable to other machines.

Lathes

Lathes are machines that cut spinning pieces of metal. CNC lathes are able to make fast, precision cuts using indexable tools and drills with complicated programs for parts that normally cannot be cut on manual lathes. These machines often include 12 tool holders and coolant pumps to cut down on tool wear. CNC lathes have similar control specifications to

CNC mills and can often read G-code as well as the manufacturer's proprietary programming language.

Plasma cutters



CNC plasma cutting

Plasma cutting involves cutting a material using a plasma torch. It is commonly used to cut steel and other metals, but can be used on a variety of materials. In this process, gas (such as compressed air) is blown at high speed out of a nozzle; at the same time an electrical arc is formed through that gas from the nozzle to the surface being cut, turning some of that gas toplasma. The plasma is sufficiently hot to melt the material being cut and moves sufficiently fast to blow molten metal away from the cut.

Electric discharge machining

Electric discharge machining (EDM), sometimes colloquially also referred to as spark machining, spark eroding, burning, die sinking, or wire erosion, is a manufacturing process in which a desired shape is obtained using electrical discharges (sparks). Material is removed from the workpiece by a series of rapidly recurring current discharges between two electrodes, separated by a dielectric fluid and subject to an electric voltage. One of the electrodes is called the tool-electrode, or simply the "tool" or "electrode," while the other is called the workpiece-electrode, or "workpiece."

When the distance between the two electrodes is reduced, the intensity of the electric field in the space between the electrodes becomes greater than the strength of the dielectric (at least in some point(s)), which breaks, allowing current to flow between the two electrodes. This phenomenon is the same as the breakdown of a capacitor. As a result, material is removed from both the electrodes. Once the current flow stops (or it is stopped – depending on the type of generator), new liquid dielectric is usually conveyed into the inter-electrode volume enabling the solid particles (debris) to be carried away and the insulating proprieties of the dielectric to be restored. Adding new liquid dielectric in the inter-electrode volume is commonly referred to as flushing. Also, after a current flow, a difference of potential between the two electrodes is restored to what it was before the breakdown, so that a new liquid dielectric breakdown can occur.

Wire EDM

Also known as wire cutting EDM, wire burning EDM, or traveling wire EDM, this process uses spark erosion to machine or remove material with a traveling wire electrode from any electrically conductive material. The wire electrode usually consists of brass or zinc-coated brass material.

Sinker EDM

Sinker EDM, also called cavity type EDM or volume EDM, consists of an electrode and workpiece submerged in an insulating liquid—often oil but sometimes other dielectric fluids. The electrode and workpiece are connected to a suitable power supply, which generates an electrical potential between the two parts. As the electrode approaches the workpiece, dielectric breakdown occurs in the fluid forming a plasma channel) and a small spark jumps.

Water jet cutters

A water jet cutter, also known as a waterjet, is a tool capable of slicing into metal or other materials (such as granite) by using a jet of water at high velocity and pressure, or a mixture of water and an abrasive substance, such as sand. It is often used during fabrication or manufacture of parts for machinery and other devices. Waterjet is the preferred method when the materials being cut are sensitive to the high temperatures generated by other methods. It has found applications in a diverse number of industries from mining to aerospace where it is used for operations such as cutting, shaping, carving, and reaming.

Other CNC tools: Many other tools have CNC variants, including:

- Drills
- EDMs
- Embroidery machines
- Lathes
- Milling machines
- Wood routers
- Sheet metal works (Turret punch)
- Wire bending machines
- Hot-wire foam cutters
- Plasma cutters
- Water jet cutters
- Laser cutting
- Oxy-fuel
- Surface grinders
- Cylindrical grinders
- 3D Printing
- Induction hardening machines
- submerged welding
- knife cutting
- glass cutting

5.3 Programming Fundamentals CNC

Fanuc G-Code List (Lathe)

G code	Description
G00	Rapid traverse
G01	Linear interpolation
G02	Circular interpolation CW
G03	Circular interpolation CCW
G04	Dwell
G09	Exact stop

G10	Programmable data input
G20	Input in inch
G21	Input in mm
G22	Stored stroke check function on
G23	Stored stroke check function off
G27	Reference position return check
G28	Return to reference position
G32	Thread cutting
G40	Tool nose radius compensation cancel
G41	Tool nose radius compensation left
G42	Tool nose radius compensation right
G70	Finish machining cycle
G71	Turning cycle
G72	Facing cycle
G73	Pattern repeating cycle
G74	Peck drilling cycle
G75	Grooving cycle
G76	Threading cycle
G92	Coordinate system setting or max. spindle speed setting
G94	Feed Per Minute
G95	Feed Per Revolution
G96	Constant surface speed control
G97	Constant surface speed control cancel

Fanuc G-Code List (Mill)

G code	Description
G00	Rapid traverse
G01	Linear interpolation
G02	Circular interpolation CW
G03	Circular interpolation CCW
G04	Dwell
G17	X Y plane selection
G18	Z X plane selection
G19	Y Z plane selection
G28	Return to reference position

G30	2nd, 3rd and 4th reference position return
G40	Cutter compensation cancel
G41	Cutter compensation left
G42	Cutter compensation right
G43	Tool length compensation + direction
G44	Tool length compensation – direction
G49	Tool length compensation cancel
G53	Machine coordinate system selection
G54	Workpiece coordinate system 1 selection
G55	Workpiece coordinate system 2 selection
G56	Workpiece coordinate system 3 selection
G57	Workpiece coordinate system 4 selection
G58	Workpiece coordinate system 5 selection
G59	Workpiece coordinate system 6 selection
G68	Coordinate rotation
G69	Coordinate rotation cancel
G73	Peck drilling cycle
G74	Left-spiral cutting circle
G76	Fine boring cycle
G80	Canned cycle cancel
G81	Drilling cycle, spot boring cycle
G82	Drilling cycle or counter boring cycle
G83	Peck drilling cycle
G84	Tapping cycle
G85	Boring cycle
G86	Boring cycle
G87	Back boring cycle
G88	Boring cycle
G89	Boring cycle
G90	Absolute command
G91	Increment command
G92	Setting for work coordinate system or clamp at maximum spindle speed
G98	Return to initial point in canned cycle
G99	Return to R point in canned cycle

Fanuc M-Code List (Lathe)

M code	Description
M00	Program stop
M01	Optional program stop
M02	End of program
M03	Spindle start forward CW
M04	Spindle start reverse CCW
M05	Spindle stop
M08	Coolant on
M09	Coolant off
M29	Rigid tap mode
M30	End of program reset
M40	Spindle gear at middle
M41	Low Gear Select
M42	High Gear Select
M68	Hydraulic chuck close
M69	Hydraulic chuck open
M78	Tailstock advancing
M79	Tailstock reversing
M94	Mirrorimage cancel
M95	Mirrorimage of X axis
M98	Subprogram call
M99	End of subprogram

Fanuc M-Code List (Mill)

M code	Description
M00	Program stop
M01	Optional program stop
M02	End of program
M03	Spindle start forward CW
M04	Spindle start reverse CCW
M05	Spindle stop
M06	Tool change
M07	Coolant ON – Mist coolant/Coolant thru spindle
M08	Coolant ON – Flood coolant

M09	Coolant OFF
M19	Spindle orientation
M28	Return to origin
M29	Rigid tap
M30	End of program (Reset)
M41	Low gear select
M42	High gear select
M94	Cancel mirrorimage
M95	Mirrorimage of X axis
M96	Mirrorimage of Y axis
M98	Subprogram call
M99	End of subprogram

5.4 Manual Part Programming

Lathe

G02 G03 G Code Circular Interpolation

G02 G Code Clock wise Circular Interpolation.

G03 G Code Counter Clock wise Circular Interpolation.

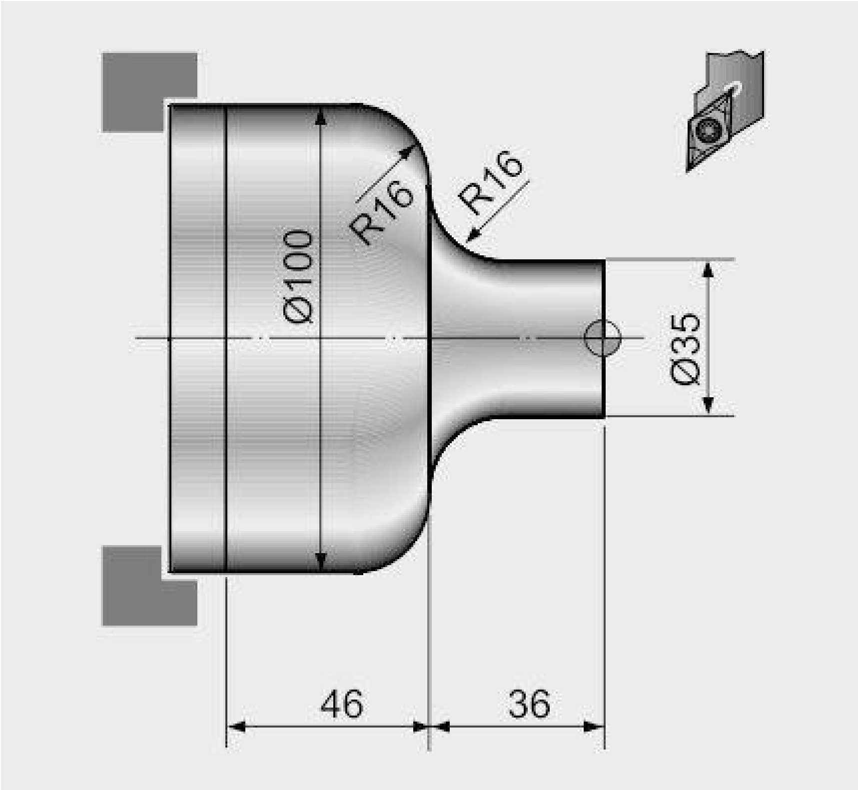
There are multiple articles/cnc program examples about G code circular interpolation, here is the list of few articles so that cnc machinists can easily navigate through different cnc programming articles.

G02 G03 G Code Example CNC Programs (G code Arc Examples)

- CNC Circular Interpolation Tutorial G02 G03
- Fanuc CNC Lathe Programming Example
- CNC Programming Example G Code G02 Circular Interpolation Clockwise
- Fanuc G20 Measuring in Inches with CNC Program Example
- CNC Arc Programming Exercise
- CNC Programming for Beginners a CNC Programming Example
- CNC Lathe Programming Example

Here is a new CNC programming examples which shows the use of G02 G03 G code circular interpolation.

G02 G03 G Code Example Program



G02 G03 G Code Circular Interpolation Example Program

```
N20 G50 S2000 T0300
G96 S200 M03
G42 G00 X35.0 Z5.0 T0303 M08
G01 Z-20.0 F0.2
G02 X67.0 Z-36.0 R16.0
G01 X68.0 :
G03 X100.0 Z-52.0 R16.0
G01 Z-82.0
G40 G00 X200.0 Z200.0 M09 T0300
M30
```

G Code G02 G03 I & K Example Program

G02 G03 G Code Circular Interpolation can be programmed in two ways,

```
G02 X... Z... R...
G02 X... Z... I... K...
```

The below is the same cnc program but this version uses I & K with G02 G03 G code.

```
N20 G50 S2000 T0300
G96 S200 M03
G42 G00 X35.0 Z5.0 T0303 M08
G01 Z-20.0 F0.2
G02 X67.0 Z-36.0 I16.0 K0
G01 X68.0 :
G03 X100.0 Z-52.0 I0 K-16.0
G01 Z-82.0
G40 G00 X200.0 Z200.0 M09 T0300
```

```
M30
```

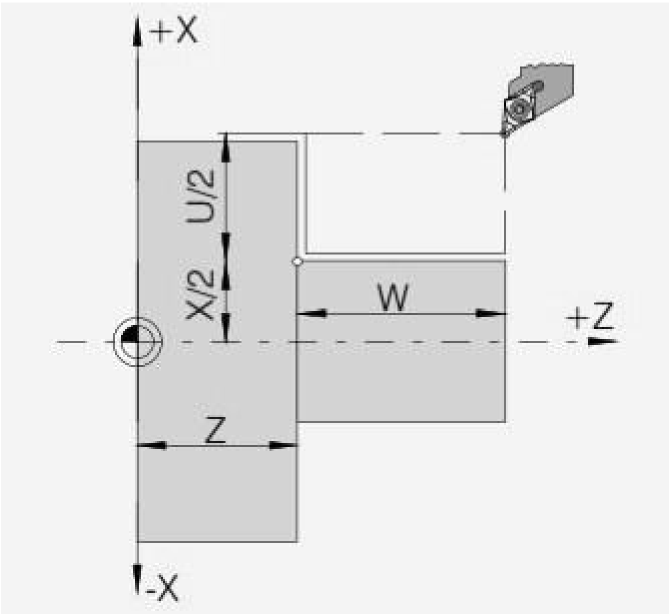
G20 Turning Cycle Format for Straight Turning

```
G20 X… Z… F…
```

or

```
G20 U… W… F…
```

X – Diameter to be cut (absolute).
Z – End point in z-axis (absolute).
F – Feed-rate.
U – Diameter to be cut (incremental).
W – End point in z-axis (incremental).



G20 Turning Cycle – CNC Lathe Fanuc 21 TB

G20 Turning Cycle Format for Taper Turning

```
G20 X… Z… R… F…
```

or

```
G20 U… W… R… F…
```

X – Diameter to be cut (absolute).
Z – End point in z-axis (absolute).
R – Incremental taper dimension in X with direction (+/-)
F – Feed-rate.
U – Diameter to be cut (incremental).
W – End point in z-axis (incremental).

As cnc machinists can use X or U value for the contour value, same way Z or W can be used or you can even mix both absolute (X, Z) and incremental (U, W) values.

G20 Turning Cycle Example CNC Program Code

```
G96 S200 M03
G00 X56.0 Z2.0
G20 X51.0 W-20.0 F0.25
X46.0
X41.0
X36.0
X31.0
X30.0
G00 X100 Z100
M30
```

CNC Program Code Explanation

As you can see in the above cnc program code,
Tool is at X56 Z2 point,
First cut is made at X51 and tool travels W-20 in Z-axis.
Second cut is made at X46
Third cut is made at X41
…
Last cut is made at X30

G20 Turning Cycle Function

As if you study the above cnc program code you will notice that,
1 – with G20 both absolute (X51.0) and incremental (W-20.0) values are used to make cuts.
2 – If above code also shows a very powerful functionality of G20 turning cycle which is that a cnc machinist can control depth-of-cut of every pass of G20 turning cycle which is impossible to achieve with other Turning Canned Cycle like G71 Rough Turning Cycle. So you will notice first five-cuts are of 5mm deep but the last one is just 1mm deep.

Cancellation of G20 Turning Cycle

G20 turning cycle is a modal G-code.
"Modal" G-code meaning that they stay in effect until they are cancelled or replaced by a contradictory G code.
It means G20 turning cycle remains active until another motion command is given like G00, G01 etc. As in above cnc program example G20 G code is cancelled with G00 G code.

Milling

Programming

```
G72.1 P... L... X... Y... R...
```

Parameters

Parameter	Description
P	Subprogram number
L	Number of times the operation is repeated
X	Center of rotation on the X axis

Y	Center of rotation on Y axis
R	Angular displacement (a positive value indicates a counter clockwise angular displacement. Specify an incremental value.)

G-Code Data

Modal/Non-Modal	G-Code Group
Non-Modal	00

Programming Notes

Notes

1. In the G72.1 block, addresses other than P, L, X, Y and R are ignored.
2. P, X, Y and R must always be specified.
3. If L is not specified, the figure is copied once.
4. The coordinate of the center of rotation is handled as an absolute value even if it is specified in the incremental mode.
5. Specify an increment in the angular displacement at address R. The angular displacement (degree) for the Nth figure is calculated as follows: Rx(N-1).

First block of the subprogram

Always specify a move command in the first block of a subprogram that performs a rotational copy. If the first block contains only the program number such as O00001234; and does not have a move command, movement may stop at the start point of the figure made by the n-th (n = 1,2, 3, …) copying.

Example of an incorrect program

```
O00001234 ;
G00 G90 X100.0 Y200.0 ;
;
;
M99 ;
```

Example of a correct program

```
O00001000 G00 G90 X100.0 Y200.0 ;
;
;
M99 ;
```

Limitation

Specifying two or more commands to copy a figure

G72.1 cannot be specified more than once in a subprogram for making a rotational copy (If this is attempted, alarmPS0900 will occur).

In a subprogram that specifies rotational copy, however, linear copy (G72.2) can be specified. Similarly, in a subprogram that specifies linear copy, rotational copy can be specified.

Commands that must not be specified

Within a program that performs a rotational copy, the following must not be specified:

——————————Command for changing the selected plane (G17 to G19)

——————————Command for specifying polar coordinates (G16)

——————————Reference position return command(G28)

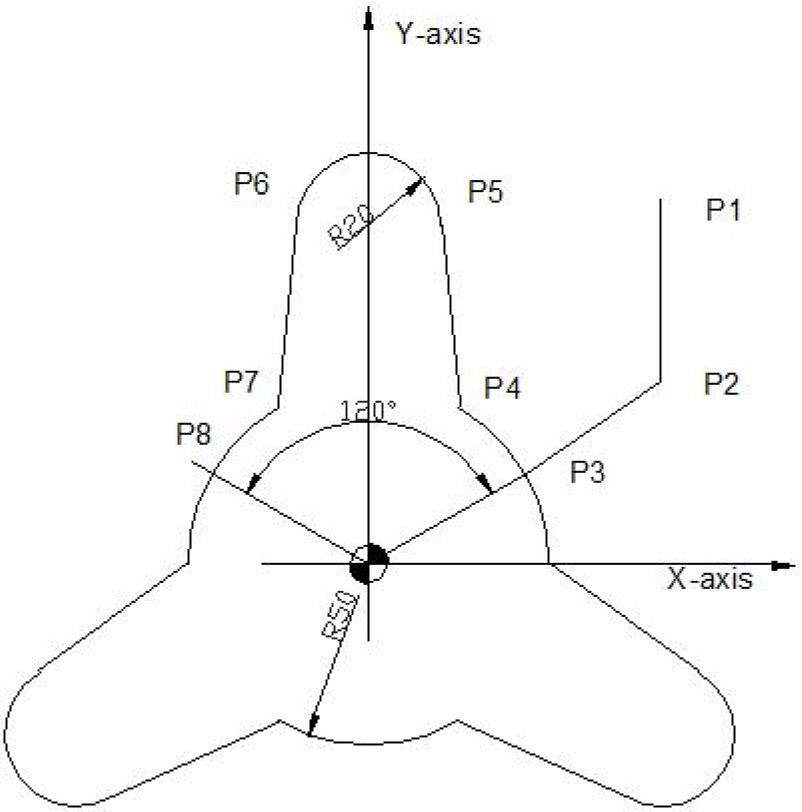
——————————Axis switching

——————————————Coordinate system rotation (G68)
——————————————scaling (G51)
——————————————programmable mirror image (G51.1)
The command for rotational copying can be specified after a command for coordinate system rotation, scaling, or programmable mirror image is executed.

Single block

Single-block stops are not performed in a block with G721.1 or G72.2.

G72.1 Programming Example



Main program

```
O1000 ;
N10 G90 G00 X80. Y100. ;            (P1)
N20 Y50. ;                     (P2)
N30 G01 G17 G42 X43.301 Y25. D01 F100 ;(P3)
N40 G72.1 P1100 L3 X0 Y0 R120. ;
N50 G90 G40 G01 X80. Y50. ;          (P2)
N60 G00 X80. Y100. ;             (P1)
N70 M30 ;
```

Sub program

```
O1100 G91 G03 X-18.301 Y18.301 R50. ; (P4)
N100 G01 X-5. Y50. ;                  (P5)
N200 G03 X-40. I-20. ;                (P6)
N300 G01 X-5. Y-50. ;                 (P7)
N400 G03 X-18.301 Y-18.301 R50. ;     (P8)
N500 M99 ;
```

5.5 Micromachining

Superfinishing, a metalworking process for producing very fine surface finishes

Various micro electro mechanical systems

Bulk micromachining

Surface micromachining

High-aspect-ratio microstructure technologies

Bulk micromachining is a process used to produce micro machinery or micro electro mechanical systems (MEMS).

Unlike surface micromachining, which uses a succession of thin film deposition and selective etching, bulk micromachining defines structures by selectively etching inside a substrate. Whereas surface micromachining creates structures *on top* of a substrate, bulk micromachining produces structures *inside* a substrate.

Usually, silicon wafers are used as substrates for bulk micromachining, as they can be anisotropically wet etched, forming highly regular structures. Wet etching typically uses alkaline liquid solvents, such as potassium hydroxide (KOH) or tetramethylammonium hydroxide (TMAH) to dissolve silicon which has been left exposed by the photolithography masking step. These alkali solvents dissolve the silicon in a highly anisotropic way, with some crystallographic orientations dissolving up to 1000 times faster than others. Such an approach is often used with very specific crystallographic orientations in the raw silicon to produce V-shaped grooves. The surface of these grooves can be atomically smooth if the etch is carried out correctly, and the dimensions and angles can be precisely defined.

Bulk micromachining starts with a silicon wafer or other substrates which is selectively etched, using photolithography to transfer a pattern from a mask to the surface. Like surface micromachining, bulk micromachining can be performed with wet or dry etches, although the most common etch in silicon is the anisotropic wet etch. This etch takes advantage of the fact that silicon has a crystal structure, which means its atoms are all arranged periodically in lines and planes. Certain planes have weaker bonds and are more susceptible to etching. The etch results in pits that have angled walls, with the angle being a function of the crystal orientation of the substrate. This type of etching is inexpensive and is generally used in early, low-budget research.

Unlike Bulk micromachining, where a silicon substrate (wafer) is selectively etched to produce structures, surface micromachining builds microstructures by deposition and etching of different structural layers on top of the substrate. Generally polysilicon is commonly used as one of the layers and silicon dioxide is used as a sacrificial layer which is removed or etched out to create the necessary void in the thickness direction. Added layers are generally very thin with their size varying from 2-5 Micro metres. The main advantage of this machining process is the possibility of realizing monolithic microsystems in which the electronic and the mechanical components(functions) are built in on the same substrate. The surface micromachined components are smaller compared to their counterparts, the bulk micromachined ones.

As the structures are built on top of the substrate and not inside it, the substrate's properties are not as important as in bulk micromachining, and the expensive silicon wafers can be replaced by cheaper substrates, such as glass or plastic. The size of the substrates can also be much larger than a silicon wafer, and surface micromachining is used to produce TFTs on large area glass substrates for flat panel displays. This technology can also be used for the manufacture of thin film solar cells, which can be deposited on glass, but also on PET substrates or other non-rigid materials.

HARMST is an acronym for **H**igh **A**spect **R**atio **M**icrostructure **T**echnology that describes fabrication technologies, used to create high-aspect-ratio microstructures with heights between tens of micrometers up to a centimeter and aspect ratios greater than 10:1. Examples include the LIGA fabrication process, advanced silicon etch, and deep reactive ion etching.

5.6 Water Machining

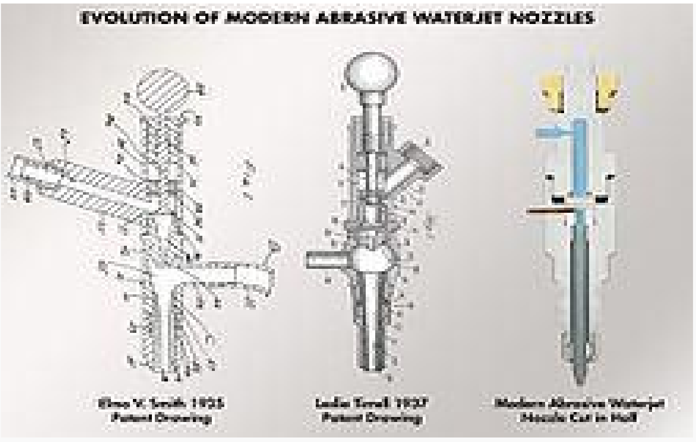
A water jet cutter, also known as a waterjet or waterjet, is an industrial tool capable of cutting a wide variety of materials using a very high-pressure jet of water, or a mixture of water and an abrasive substance. The term abrasive jet refers specifically to the use of a mixture of water and abrasive to cut hard materials such as metal or granite, while the terms pure waterjet and water-only cutting refer to waterjet cutting without the use of added abrasives, often used for softer materials such as wood or rubber. Waterjet cutting is often used during fabrication of machine parts. It is the preferred method when the materials being cut are sensitive to the high temperatures generated by other methods. Waterjet cutting is used in various industries, including mining andaerospace, for cutting, shaping, and reaming.

Water jet CNC cutting Machine

While using high-pressure water for erosion dates back as far as the mid-1800s with hydraulic mining, it was not until the 1930s that narrow jets of water started to appear as an industrial cutting device. In 1933, the Paper Patents Company in Wisconsin developed a paper metering, cutting, and reeling machine that used a diagonally moving waterjet nozzle to cut a horizontally moving sheet of continuous paper. These early applications were at a low pressure and restricted to soft materials like paper.

Waterjet technology evolved in the post-war era as researchers around the world searched for new methods of efficient cutting systems. In 1956, Carl Johnson of Durox International in Luxembourg developed a method for cutting plastic shapes using a thin stream high-pressure waterjet, but those materials, like paper, were soft materials.[3] In 1958, Billie Schwacha of North American Aviation developed a system using ultra-high-pressure liquid to cut hard materials.[4] This system used a 100,000 psi (690 MPa) pump to deliver ahypersonic liquid jet that could cut high strength alloys such as PH15-7-MO stainless steel. Used as a honeycomb laminate on the Mach 3 North American XB-70 Valkyrie, this cutting method resulted in delaminating at high speed, requiring changes to the manufacturing process. While not effective for the XB-70 project, the concept was valid and further research continued to evolve waterjet cutting. In 1962, Philip Rice of Union Carbideexplored using a pulsing waterjet at up to 50,000 psi (345 MPa) to cut metals, stone, and other materials. Research by S.J. Leach and G.L. Walker in the mid-1960s expanded on traditional coal waterjet cutting to determine ideal nozzle shape for high-pressure waterjet cutting of stone, and Norman Franz in the late 1960s focused on waterjet cutting of soft materials by dissolving long chain polymers in the water to improve the cohesiveness of the jet stream. In the early 1970s, the desire to improve the durability of the waterjet nozzle led Ray Chadwick, Michael Kurko, and Joseph Corriveau of the Bendix Corporation to come up with the idea of using corundum crystal to form a waterjet orifice, while Norman Franz expanded on this and created a waterjet nozzle with an orifice as small as 0.002 inches (0.05 mm) that operated at pressures up to 70,000 psi (483 MPa). John Olsen, along with George Hurlburt and Louis Kapcsandy at Flow Research (later Flow Industries), further improved the commercial potential of the waterjet by showing that treating the water beforehand could increase the operational life of the nozzle.

Abrasive waterjet



The Evolution of the Abrasive Waterjet Nozzle

While cutting with water is possible for soft materials, the addition of an abrasive turned the waterjet into a modern machining tool for all materials. This began in 1935 when the idea of adding an abrasive to the water stream was developed by Elmo Smith for the liquid abrasive blasting. Smith's design was further refined by Leslie Tirrell of the Hydroblast Corporation in 1937, resulting in a nozzle design that created a mix of high-pressure water and abrasive for the purpose of wet blasting. Producing a commercially viable abrasive waterjet nozzle for precision cutting came next by Dr. Mohamed Hashish who invented and led an engineering research team at Flow Industries to develop the modern abrasive waterjet cutting technology. Dr. Hashish, who also coined the new term "Abrasive Waterjet" AWJ, and his team continued to develop and improve the AWJ technology and its hardware for many applications which is now in over 50 industries worldwide. A most critical development was creating a durable mixing tube that could withstand the power of the high-pressure AWJ, and it was Boride Products (now Kennametal) development of their ROCTEC line of ceramic tungsten carbide composite tubes that significantly increased the operational life of the AWJ nozzle. Current work on AWJ nozzles is on micro abrasive waterjet so cutting with jets smaller than 0.015 inch in diameter can be commercialized.

Applications

Because the nature of the cutting stream can be easily modified the water jet can be used in nearly every industry; there are many different materials that the water jet can cut. Some of them have unique characteristics that require special attention when cutting.

Materials commonly cut with a water jet include rubber, foam, plastics, leather, composites, stone, tile, metals, food, paper and much more. Materials that cannot be cut with a water jet are tempered glass, diamonds and certain ceramics. Water is capable of cutting materials over eighteen inches (45 cm) thick.

MCQ

Questions	Option A	Option B	Option C	Option D	Ans
In Numerical Control machine tool, the position feedback package is connected between	control unit and programmer	control unit and machine tool	programmer and machine tool	programmer and process planning	control unit and machine tool
Which one of the following CNC machine is highly suited for machining on cubical component in a single set up?	VMC	HMC	Slant bed lathe	Turret punch press	VMC
Axis parallel to spindle of a CNC machine is	Z-axis	Y-axis	X-axis	A -axis	X-axis
In Numerical Control machine tool, the position feedback package is connected between	control unit and programmer	control unit and machine tool	programmer and machine tool	programmer and process planning	control unit and machine tool
Which one of the following CNC machine is highly suited for machining on cubical component in a single set up?	VMC	HMC	Slant bed lathe	Turret punch press	VMC
The feedback device used for spindle orientation in CNC lathes	Proximity switch	Incremental encoder	Absolute encoder	Resolver	Resolver
The type of belt used in CNC machine axis drive systems	V-belt	Flat belt	Chains	Timing belt	Timing belt
CNC stands for	Computer Numerical control	Combined number control	Numerical computer	None of the above	Computer Numerical control
In NC, program developed by using	Numbers	Letters	Symbols	ALL THE ABOVE	ALL THE ABOVE
In _________ system, several NC's are controlled by a large computer	NC	CNC	DNC	ALL THE ABOVE	DNC
NC stands for	Numerical	Number control	Numerical	None of the	Numerical control

	control		computer	above	
DNC stands for	Direct Numerical Control	Direct Number control	Both a & b	None of the above	Direct Numerical Control
Punched tapes are used in	NC	CNC	DNC	ALL THE ABOVE	NC
A part program is written, using .	G AND M CODES	E AND M CODES	A AND S CODES	E AND I CODES	G AND M CODES
G refers to the G code Function.	MAIN PROGRAM	SUB PROGRAM	Preparatory	None of the above	Preparatory
M refers to the M code Function.	MACHINING	Miscellaneous	MAINTENANCE	ALL THE ABOVE	Miscellaneous
ATSe refers to	automatic tool change	ALL TOOL CHANGE	ADDITIONAL TOOL CHANGE	ANY TOOL CHANNGE	automatic tool change
M15 is used for	Coolant OFF	Spindle OFF	Coolant +Spindle OFF	ALL THE ABOVE	Coolant +Spindle OFF
Which code is used for absolute dimension programming	G85	G89	G90	G91	G90
Which code is used to stop the spindle	M02	M03	M04	M05	M05
The miscellaneous function M08 refers to ________ in CNC lathes	Tool change	Coolant ON	Coolant OFF	Spindle CW	Coolant ON
M codes are used to perform	Primary function	Miscellaneous function	Both a & b	None of the above	Miscellaneous function
The G-function G28 refers to	Reference Point Return	Screw cutting cycle	Screw cutting cycle	Tool nose radius compensation	Reference Point Return
G-03 used for	Linear interpolation	Circular interpolation clockwise	Circular interpolation anticlockwise	None of the above	Circular interpolation anticlockwise
G-01 used for	Linear	Circular	Both a & b	None of the	Linear interpolation

	interpolation	interpolation		above	
___________ denotes end of part program	M05	M03	M20	M30	M30
______________ denotes feed rate mm/rev	G91	G92	G94	G95	G95
______________ denotes feed rate mm/min	G91	G92	G94	G95	G94

2 marks

1. State the advantages of NC machines.
2. Draw the simple configuration of CNC machine
3. Mention the main different between C NC and DNC
4. What is the function of servo valve?
5. Define absolute and incremental programming.
6. What are the important steps to be followed while preparing part programming?
7. What is meant by MACRO?
8. Define subroutine
9. What do mean by canned cycle
10. What is meant by APT programme?

14 marks

1. List the various drive systems explain the principle of any two rive system.
2. Explain open loop and closed loop system
3. What is machining centers explain in detail
4. Explain various types of CMM
5. Explain the part programming procedure with a good example
6. List and explain G and M code for turning milling operations
7. Explain NC axis conventions.