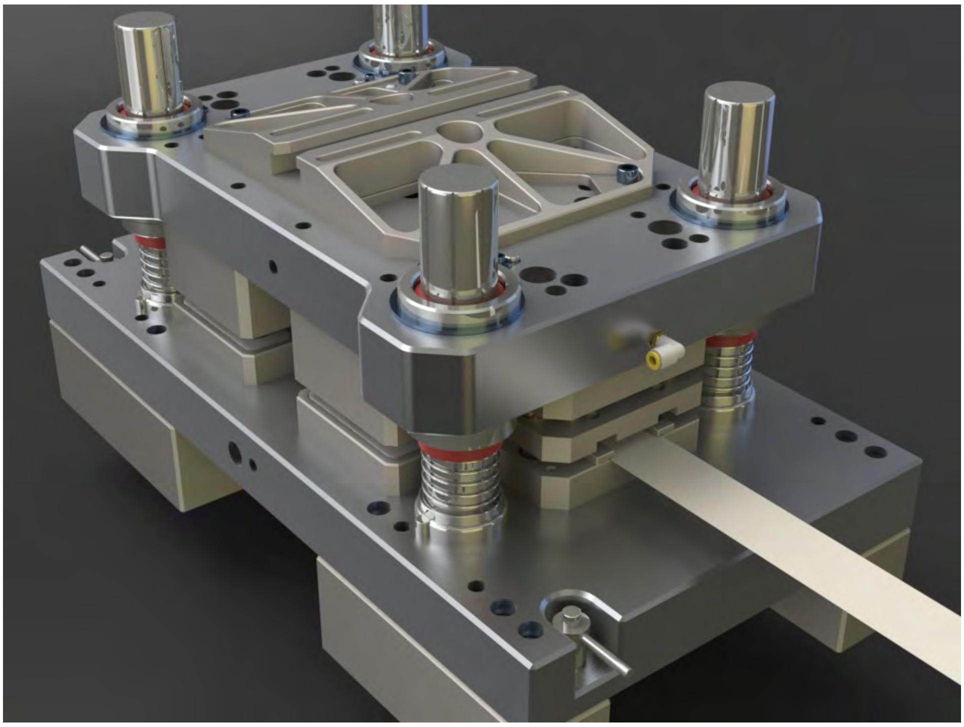


Tool and Die Design

Metal Cutting and Special Tools

YOU MAY GET STUDY MATERIAL FROM
AMIESTUDYCIRCLE.COM

INFO@AMIESTUDYCIRCLE.COM

WHATSAPP/CALL: 9412903929

Metal Cutting & Special Tools

BASIC REQUIREMENTS FOR CUTTING TOOL MATERIALS

The tool machining performance is mainly influenced by three aspects related to tool materials, tool structure and tool geometry, and the tool material is the key factor which determines the cutting performance and plays a great influence on machining efficiency, machining cost, machining quality and tool life.

Different machining applications require different cutting tool materials, and the ideal material should have the basic properties below:

High hardness and abrasive resistance

In order to achieve effective cutting performance, cutting tool material must be harder than the workpiece material. Generally, the more harder the tool material, the better the wear resistance will be, while the wear resistance of tool material is also determined by its chemical composition and the stability of its metallographic structure.

High strength and toughness

In mechanics of materials, the strength of a material refers to its ability to withstand an applied load without failure or plastic deformation. Toughness is the ability of a material to absorb energy and plastically deform without fracturing. In general, hardness increases as toughness decreases, so it is really necessary to choose the proper tool materials as per the machining applications to meet the requirements of production.

High heat resistance

Heat resistance (also named red hardness) is an index to measure the performance of tool materials. It comprehensively reflects the ability of the tool material to maintain hardness, wear resistance, strength, oxidation resistance, adhesion resistance and diffusion resistance at high temperatures.

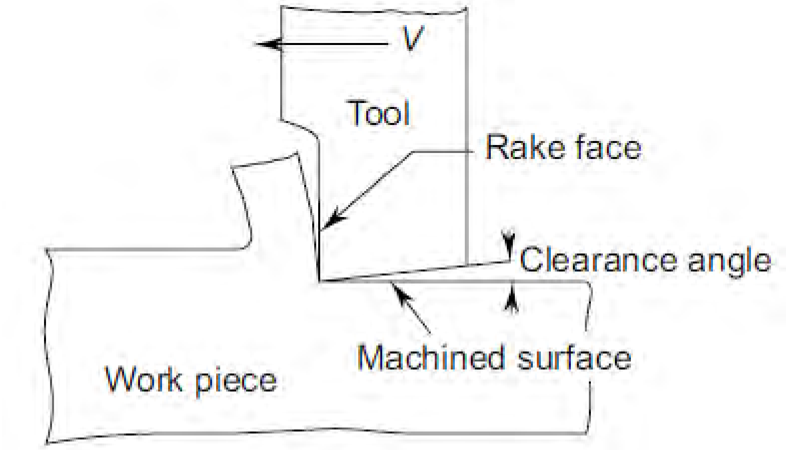
CUTTING TOOL GEOMETRY

Cutting tools are designed with sharp edges to shear the workpiece and minimize rubbing contact. Variations in the shape of the cutting tool influence tool life, the surface finish of the workpiece, and the amount of force required to shear a chip from the parent metal. Various angles on a tool comprise what is often termed the tool geometry:

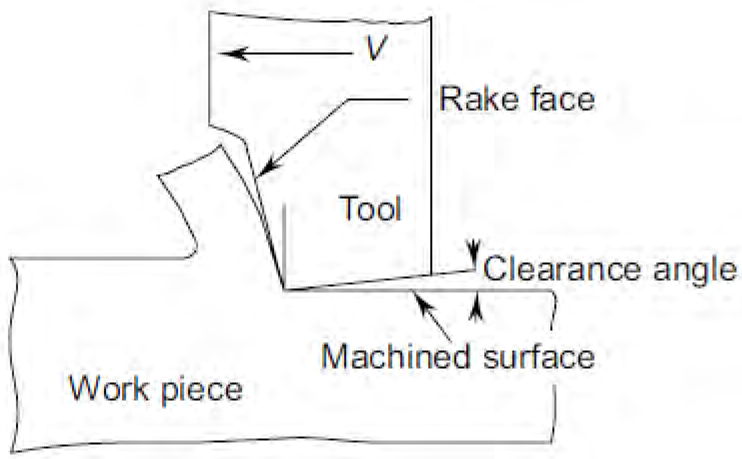
- the surface that the chip flows across is called the **face or rake face**;
- the surface that forms the other boundary of the wedge is called the **flank**,
- the **rake angle** is the angle between the tool face and a line perpendicular to the cut workpiece surface, and
- the **relief or clearance angle** is the angle between the tool flank and the cut workpiece surface.

Rake angle

- It is the angle between the face of the tool called the rake face and the normal to the machining direction.
- This angle specifies the ease with which a metal is cut. Higher the rake angle better is the cutting and less is the cutting force.
- Increasing the rake angle reduces the metal backup available at the tool rake face. This reduces the strength of the tool tip as well as the heat dissipation through the tool. Thus, there is a maximum limit to the rake angle and is generally of the order of 15° for high speed steel tools cutting mild steel.
- It is possible to have rake angle as zero or negative as shown in following figure. These are generally used in the case of highly brittle tool materials such as carbides or diamond for giving extra strength to the tool tip.



Zero rake angle



Negative rake angle

Clearance angle (relief angle)

This is the angle between the machined surface and the underside of the tool called the flank face. The clearance angle is provided such that the tool will not rub or spoil the machined surface, but at the same time will increase the cutting forces. A very large clearance angle reduces the strength of the tool tip, hence normally an angle of the order of 5 to 6° is generally used.

Technical terms used in equations

The conditions which have a predominant influence on the metal cutting are: work material, cutting tool material, cutting tool geometry, cutting speed, feed rate, depth of cut and cutting fluid used.

- The **cutting speed**, V, is the speed with which the cutting tool moves through the work material. This is generally expressed in metres per second (ms^{-1}).
- **Feed rate**, f, may be defined as the small relative movement per cycle (per revolution or per stroke) of the cutting tool in a direction usually normal to the cutting speed direction.
- **Depth of cut**, d, is the normal distance between the unmachined surface and the machined surface.

CHIP FORMATION

The majority of metal-cutting operations involve the creation of chips, or waste, from the workpiece. Chip formation involves three basic requirements.

There must be:

- a cutting tool harder and more wear-resistant than the workpiece material;
- interference between the tool and workpiece as designated by the feed and depth of cut; and
- a relative motion or cutting velocity between the tool and workpiece with sufficient force to overcome the resistance of the workpiece material.

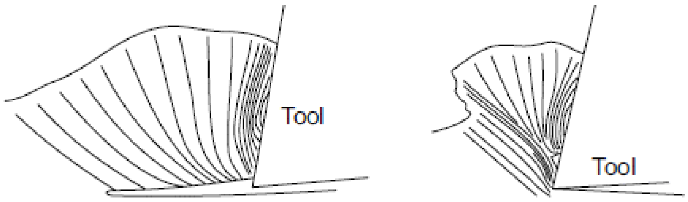
As long as these three conditions exist, the portion of material being machined that interferes with free passage of the tool will be displaced to create a chip.

The chip formation in metal cutting could be broadly categorised into three types:

- Discontinuous chip
- Continuous chip, and
- Continuous chip with BUE

Discontinuous Chip

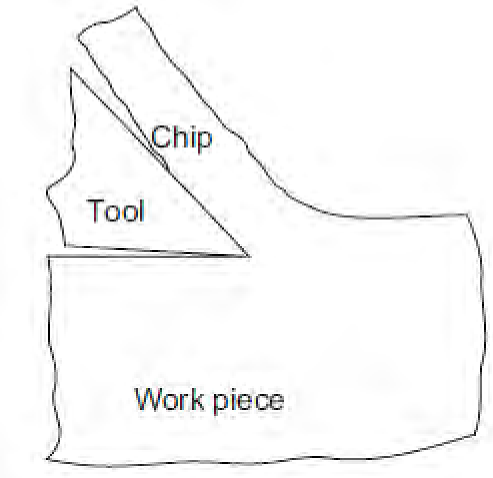
When a brittle material like cast iron or bronze is cut, it is broken along the shear plane. The same may happen if the material is ductile and the friction between the chip and tool is very high. The chips come off in small pieces or segments, and are pushed away by the tool. A chip formed in this way is called a type I, discontinuous, or segmental chip. In this type the deformed material instead of flowing continuously gets ruptured periodically.



(a) tear type chip formation (b) shear type chip formation

Continuous Chip

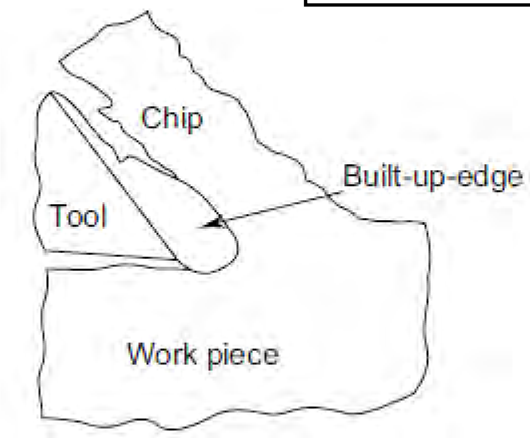
A ductile material, cut optimally, is not broken up but comes off like a ribbon, as shown in figure. This is known as a type II or continuous chip. The continuous chip is like a ribbon flows along the rake face.



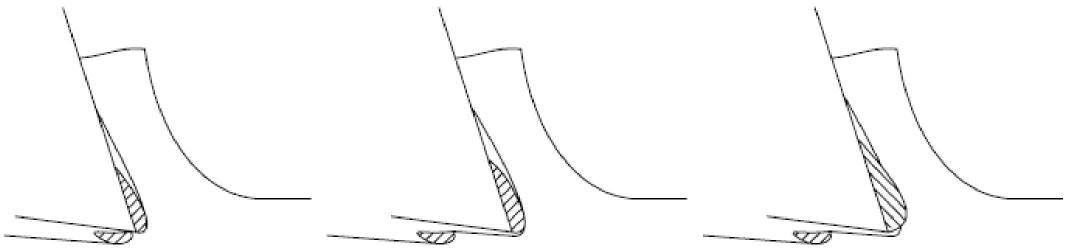
This is the most desirable form of chip since the surface finish obtained is good and cutting is smooth. It also helps in achieving higher tool life and lower power consumption. However, because of the large coils of chips, the chip disposal is a problem. To help in this direction various forms of chip breakers have been developed which are in the form of a step or groove in the tool rake face. The chip breakers allow the chips to be broken into small pieces so that they can be easily disposed of.

Continuous Chip with BUE

When the friction between tool and chip is high while machining ductile materials, some particles of chip adhere to the tool rake face near the tool tip. When such sizeable material piles up on the rake face, it acts as a cutting edge in place of the actual cutting edge as shown in figure.



This is termed as **built up edge (BUE)**. By virtue of work hardening, BUE is harder than the parent work material. As the size of BUE grows, it becomes unstable and parts of it get removed while cutting. The removed portions of BUE partly adhere to the chip underside and partly to the machined surface as shown in following figure.



This causes the finished surface to be rough. However, since the cutting is carried by the BUE and not the actual tool tip, the life of the cutting tool increases while cutting with BUE. In this way BUE is not harmful during rough machining.

The conditions that normally induce the formation of BUE are low cutting speed, high feed and low rake angle.

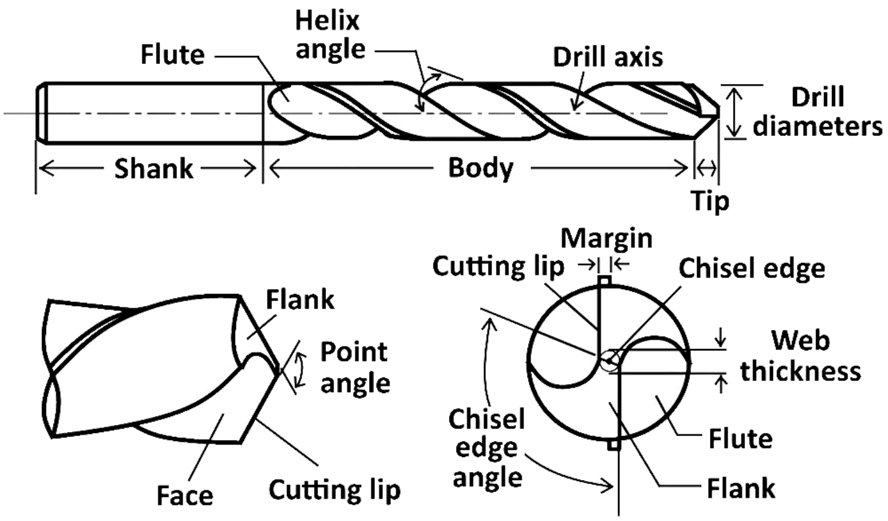
One of the prerequisites for the formation of BUE is the work hardenability of the work piece material. Higher the work hardenability, rougher is the machined surface produced.

TWIST DRILL

The cutting tool used for making holes in solid material is called the twist drill. It basically consists of two parts; the body consisting of the cutting edges and the shank which is used for holding purpose.

This has two cutting edges and two opposite spiral flutes cut into its surface as shown in figure. These flutes serve to provide clearance to the chips produced at the cutting edges. They also allow the cutting fluid to reach the cutting edges.

The drill blanks are made by forging and then twisted to provide the torsional rigidity. Then the flutes are machined and hardened before the final grinding of the geometry. Twist drill geometry is shown in the figure.



These are made with either straight or taper shank. Straight shank drills are held in the machine spindle in a drill chuck. The taper shank drills are directly held in the spindle with the help of the self-holding taper.

The tang at the end of the taper shank fits into a slot in the spindle. The tang helps to drive the drill, prevents it from slipping and provides a means of removing it from spindle.

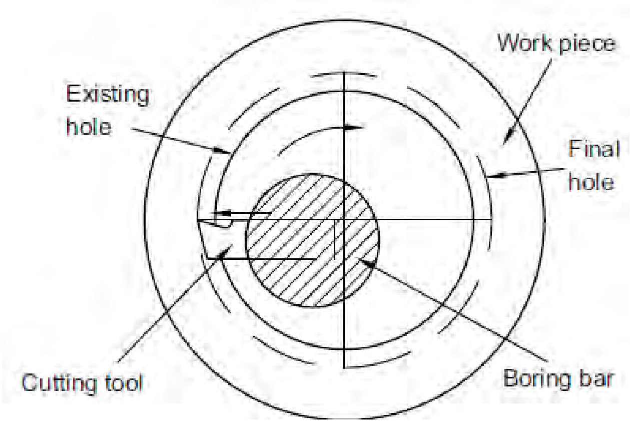
Technical terms

- The surface on the drill, which extends behind the cutting lip to the following flute, is termed as **flank**.
- **Face** is the portion of the flute surface adjacent to the cutting lip on which the chip moves as it is cut from the work piece.
- The **cutting lip** is the edge formed by the intersection of the cutting edge or face and the flank face.
- **Land or margin** is the cylindrically ground body surface on the leading edge of the drill sometimes also termed as cylindrical land. The cutting edge is reduced in diameter after the margin to provide a body clearance.
- **Axial rake angle** is the angle between the face and the line parallel to the drill axis. At the periphery of the drill, it is equivalent to the helix angle.
- **Helix angle** is the angle between the leading edge of the land and the axis of the drill. Sometimes it is also called spiral angle.
- The **lip clearance angle** is the angle formed by the portion of the flank adjacent to the land and a plane at right angles to the drill axis measured at the periphery of the drill.
- **Lead of the helix** is the distance measured parallel to the drill axis, between corresponding point on the leading edge of the land in one complete revolution.
- The **chisel edge** is formed by the intersection of the two flanks. In order to provide strength to the drill the cutting edge is thickened gradually from the bottom. It is termed as **web**. Also a back taper is provided on the body towards the shank to provide longitudinal clearance.

The shape of the drill point is the most important. The lip angle should be correct for the given application. In general 118° is found to be suitable for mild steel and other general materials. Larger values are used for hard and brittle materials, while smaller values are used for soft materials.

BORING

Boring is an operation of enlarging a hole. The single point cutting tool used for the boring operation is shown in given figure.



Generally the single point tool bit is mounted in the boring bar of suitable diameter commensurate with the diameter to be bored. The overhang of the tool is to be maintained as small as possible to reduce the chatter, which is very common in boring.

Major problem of boring with a boring bar with single point turning tool is lack of rigidity of the boring bar. The size of boring bar is dictated by the size of the hole to be bored, while its length depends upon the geometry of the bore.

Boring problems

- **Too short and hard chips** due to (i) Too high feed (ii) Too low cutting speed (iii) Unsuitable geometry
- **Too long chips** due to (i) Too low feed (ii) Too high cutting speed (iii) Unsuitable geometry
- **Tool vibrations** due to (i) Too high tool length/coupling size ratio (ii) Unstable conditions (iii) Too low feed (iv) Too high feed (v) Too high speed (vi) Too large cutting depth (vii) Too high cutting force
- **Machine power** due to Limited machine power
- **Lack of rigidity**

Boring bars

Boring deep holes can involve extreme length-to-diameter ratios, or overhang, when it comes to tooling assemblies. Since it can be difficult to maintain accuracy and stability in these scenarios, we need boring bars to extend tooling assemblies and while maintaining the rigidity to make perfect circles with on-spec finishes.

Solid boring bars

Typically made of carbide for finishing or heavy metal for roughing, solid boring bars have dense structures that make for a more stable cut as axial force is applied.

Damping bars

When cutting speeds are compromised, or surface finishes show chatter in a long-reach boring operation, damping bars are an option. They have integrated damping systems. Our version, the Smart Damper, works as both a counter damper and friction damper so that chatter is essentially absorbed.

Characteristics of boring

- The tool structure is simple, and the radial size can be adjusted, and holes with different diameters can be machined with one tool.
- It can correct the axis skew and position error of the original hole.
- Because the boring machine has many forms of motion, the workpiece is placed on the worktable, which can conveniently and accurately adjust the relative position of the processed hole and the tool, thus ensuring the accuracy of the mutual position of the processed hole and other surfaces.
- The quality of boring mainly depends on the accuracy of the machine tool and the technical level of the workers, so the technical requirements of the operator are relatively high.
- Compared with reaming, single-edged boring cutters have poor rigidity and use a smaller cutting amount, so the productivity is lower and it is not easy to ensure stable machining accuracy.
- It is not suitable for processing slender holes.

Boring vs turning

Concept

Boring is an inner diameter cutting process that uses tools to expand holes or other circular contours. Turning is lathe processing. Lathe processing mainly uses turning tools to turn rotating workpieces.

Sizes

Boring is the use of a rotating single-edged boring tool to expand the prefabricated hole on the workpiece to a certain size. Turning is a cutting process in which the turning tool moves in a straight line or a curve in a plane.

Uses

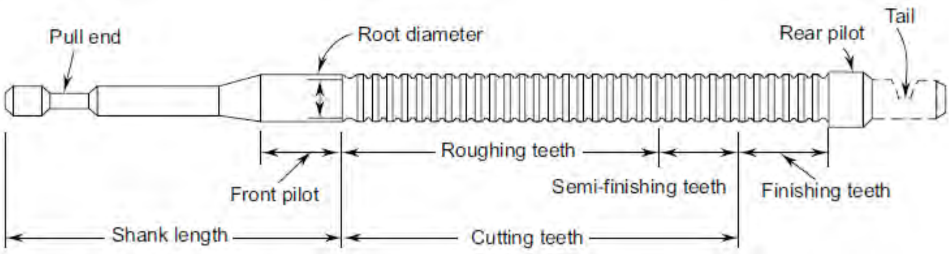
When boring, the workpiece is installed on the machine tool table or the machine fixture, the boring tool is clamped on the boring bar (also can be integrated with the boring bar), and the spindle is driven to rotate. When the boring die is used, the boring bar and the spindle are floatingly connected, and the machining accuracy depends on the accuracy of the boring die;

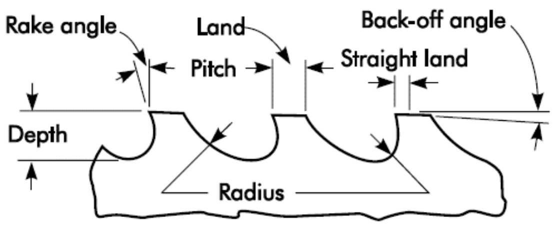
when the boring die is not used, the boring bar is rigidly connected to the spindle, and the machining accuracy depends on the accuracy of the machine tool.

Turning uses different turning tools or other tools on the lathe to process various rotating surfaces, such as inner and outer cylindrical surfaces, inner and outer conical surfaces, threads, grooves, end surfaces and forming surfaces, etc. The processing accuracy can reach IT8 to IT7, and the surface roughness The value of Ra is 1.6～0.8. Turning is commonly used to process parts with a single axis, such as straight shafts, general discs, and sleeve parts.

BROACHING

- The most common multiple-point, linear travel tool is the broach.
- Broaches are used for producing either external or internal surfaces. The surfaces produced may be flat, circular, or of an intricate profile, as viewed in a section normal to tool travel.
- A broach is essentially a series of single-point tools following each other in the axial direction along a tool body or holder.
- Successive teeth vary in size or shape in such a manner that each following tooth will cut a chip of the proper thickness.
- The basic elements of broach construction are illustrated in following figure.





- The spacing and shape of broach teeth are determined by the length of the workpiece and the chip thickness per tooth, as well as by the type of chips formed.
- Broach teeth are provided with rake and relief angles in the same manner as other cutting tools. Standard broaching nomenclature designates the rake angle as the face angle and the relief clearance as the back-off angle.

The pitch as shown in given figure determines the length of cut and chip thickness that a particular broach can handle.

$$P = 0.35\sqrt{length\, of\ cut}$$

The total stock, D_s to be removed is distributed among the teeth uniformly as depth of cut per tooth, D_T, and then the length of broach, L_B can be given as

Length of broach

$$L_B = \left(\frac{D_s}{D_T} + Z_f \right) P$$

Where Z_f = the number of teeth required for finishing the operation and is assumed to be 4 or 5 teeth.

The broaching time, T_B in min

$$\frac{L}{1000V_f} + \frac{L}{1000V_r}$$

where L = Length of stroke, mm;

V_f = Cutting speed in the forward stroke, m/min;

V_r = Return speed, m/min.

Advantages of broaching

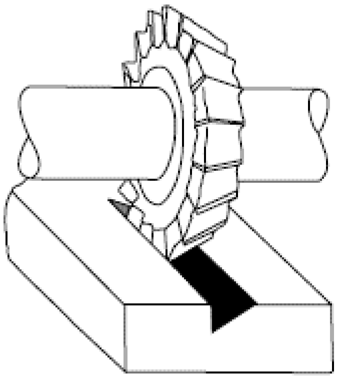
- It is the fastest way of finishing an operation with a single stroke.
- Since all the machining parameters are built into the broach, very little skill is required from the operator.
- Broaching machine is simple since only a single reciprocating motion is required for cutting.
- Final cost of the machining operation is one of the lowest for mass production.
- Any type of surface, internal or external, can be generated with broaching.
- Many surfaces which are very difficult or impossible by other means can be done by broaching. For example, square hole and internal splines can be easily produced by broaching.
- Good surface finish and fine dimensional tolerances can be achieved by broaching, often better than boring or reaming.

Limitations of broaching

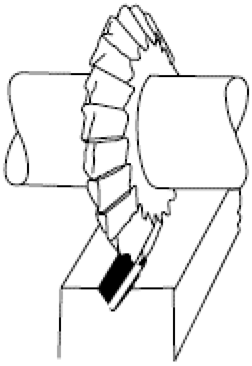
- Custom made broaches are very expensive and can therefore be justified only for very large volume production.
- A broach has to be designed for a specific application and can be used only for that application. Hence the lead time for manufacture is more for custom designed broaches.
- Broaching being a very heavy metal removal operation requires that the work piece is rigid and capable of withstanding the large forces.
- Broaching can only be carried out on the work piece whose geometry is such that there is no interference for the broach movement for the cutting.

SPECIAL FORM CUTTERS

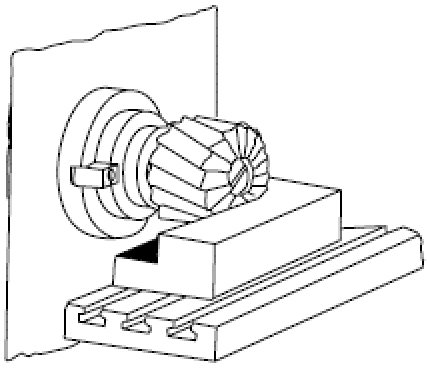
In addition to the general type of milling cutters described above, there are a large number of special form milling cutters available, which are used for machining specific profiles.



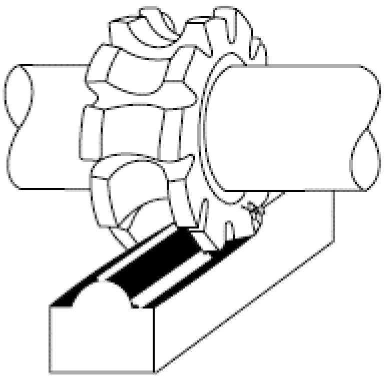
Angle milling cutter



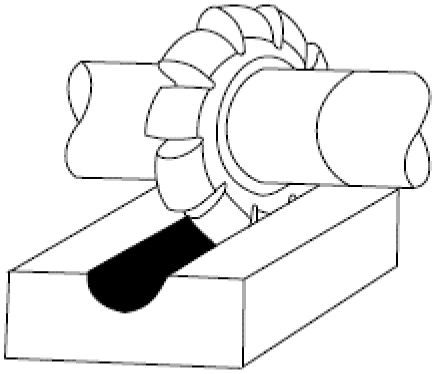
Angle milling cutter



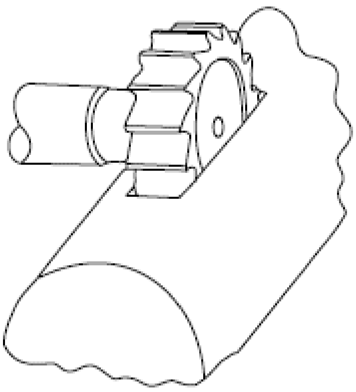
Shell end mill



Form relieved circular cutter



Form relieved circular cutter



Woodruff key cutter

Angular milling cutters are made in single or double angle cutters for milling any angle such as 30,45 or 60°.

Form relieved cutters are made of various shapes such as circular, comer rounding, convex or concave shapes.

T-slot milling cutters are used for milling T-slots such as those in the milling machine table. The central slot is to be milled first using an end mill before using the T-slot milling cutter.

Woodruff key seat milling cutters as the name suggests are used for milling Woodruff key seats.

HIGH PRECISION PARTS WITH CYLINDRICAL ID/OD GRINDING

Machines used for cylindrical grinding – a process used to finish off the outside or inside diameter of a cylindrical part – are a mainstay of many metalworking shops. With cylindrical grinding, both the workpiece and the grinding wheel are constantly rotating, as the grinding wheel is fed towards and away from the piece. These flexible machines can work on objects with various shapes, diameters, and sizes, as long as they have a central axis of rotation.

There are various types of single-purpose, single operation cylindrical grinders, including OD (outside diameter) or ID (inside diameter).

OD grinders work on the external (outside) surface of an object as it is rotated between the centers.

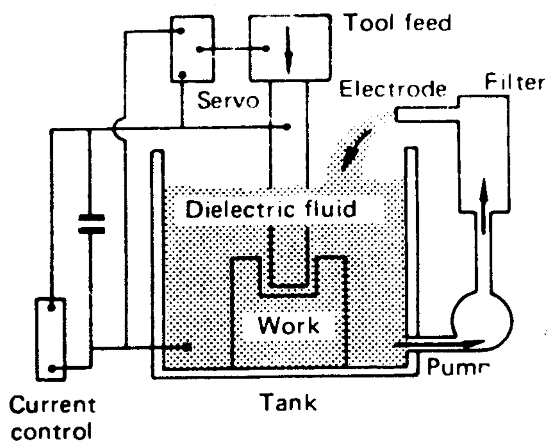
With ID grinding, small amounts of material are removed from a part to finish off its inner surface. Just as with OD grinding, the ID grinding wheel and the object rotate in opposite directions, giving reversed directional contact of the two surfaces where the grinding occurs.

Because of their ability to produce parts with high-quality surface finishes and excellent accuracy, ID/OD grinding machines are an invaluable option for extremely precise grinding applications (such as bearing and hydraulic components) and meeting the very tight tolerances required by industries such as aerospace, automotive, and defence.

ELECTRIC-DISCHARGE MACHINING (EDM)

Principles of Operations

Perhaps you have observed an are caused by an accidental short circuit and noticed the pitting that occurred on the surface of the shorted material. This is the principle that operates in EDM machining. In EDM, the spark is pulsed at frequencies ranging from a few hundred to several hundred thousand kilohertz. The workpiece and electrode are separated by a dielectric, as shown schematically in given figure The dielectric is usually a low-viscosity hydrocarbon or mineral oil. As the tool and the workpiece are brought together, the gap between them is maintained by a servomechanism comparing actual gap voltage to a preset reference voltage.



The spark discharge is contained in a microscopically small area, but the force and temperatures resulting are impressive. Temperatures around 10,000 ^{0}C and pressures many thousand times greater than atmospheric are created. Thus a minute part of the workpiece is vaporized. The metal is expelled as the column of ionized dielectric vapor collapse. The tiny particles are cooled into spheres and are swept away from the machining gap by the flow of dielectric fluid.

Electrode Materials. Any electrically conductive material can serve as the electrode. Ever since the early days of EDM, graphite has been widely used because it has a relatively low wear rate and a high degree of electrical efficiency and is relatively easy to machine and inexpensive. Various electrode materials used: Graphite, Copper, Copper graphite, Brass, Zinc alloys, Steel, Copper tungsten, Silver tungsten, Tungsten, etc. Some

Applications

The most frequent use of EDM is for tool and die work and moulds for plastics.

Advantages

- The hardness of the workpiece is not a factor. As long as the material can conduct current, it can be machined.
- Any shape that can be produced in a tool can be reproduced in the workpiece. Complicated tooling may be made up in segments and fastened together with an adhesive.
- The absence of almost all mechanical force makes it possible to machine the most fragile components without distortion. A 0.05 cm diameter hole, for example, can be produced in a small, delicate part using a very fine wire as a tool.

Disadvantages

- Tool wear requires stepped tooling or redressing of tools for deep holes, except for CNC machines.
- EDM leaves a recast layer at the surface of the cut. This may be an advantage or disadvantage. Where undesirable, it may be removed by a light finishing cut or by polishing. Depending on the current density used, the recast layer may be from 0.005 to 0.13 mm in thickness.
- EDM is slow when compared to conventional methods. Whenever possible, the cavities are roughed out prior to heat treatment and then finished by EDM after heat treatment.

NUMERICAL CONTROL MACHINE

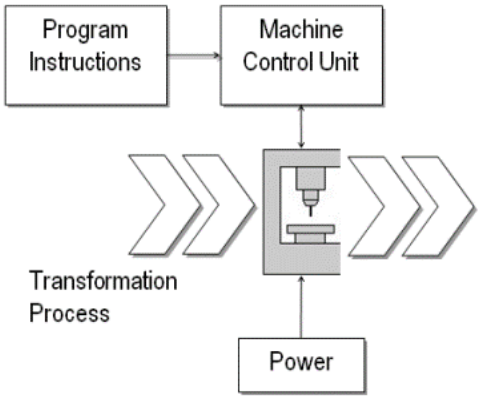
Numerical control, popularly known as the NC is very commonly used in the machine tools. Numerical control is defined as the form of programmable automation, in which the process is controlled by the number, letters, and symbols. In case of the machine tools this programmable automation is used for the operation of the machines.

In other words, the numerical control machine is defined as the machined that is controlled by the set of instructions called as the program. In numerical control method the numbers form the basic program instructions for different types of jobs; hence the name numerical control is given to this type of programming. When the type of job changes, the program instructions of the job also change. It is easier to write the new instructions for each job, hence NC provides lots of flexibility in its use.

The NC technology can be applied to wide variety of operations like drafting, assembly, inspection, sheet metal working, etc. But it is more prominently used for various metal machining processes like turning, drilling, milling, shaping etc. Due to NC all the machining operations can be performed at the fast rate resulting in bulk manufacturing becoming quite cheaper.

Three basic components of an NC system

1. Input medium
2. Machine control unit (MCU)
3. Machine tool



Types of NC (Numerical Control) Machine

There are 3 types of NC machines and are as follows.

- Traditional Numerical Control (NC Machine)
- Computer Numerical Control (CNC Machine)
- Distributed Numerical Control (DNC Machine)

Traditional Numerical Control (NC Machine):

NC machines are the evolution after Conventional machines. They can run with the help of a tape reader system i.e. whatever the operation you want to perform, you can punch it on the tape, and thereby the NC machine can perform that operation.

Computer Numerical Control (CNC Machine)

The Evolution of the CNC machine takes place after the evolution of NC machines. To overcome the limitation of the NC machine, the CNC machine has come into the picture.

In the case of NC machines, the Tape Reader system is used, which after several usages, the wear and tear of the tape take place and the operator has to punch again on the new tape to carry out the operation.

In order to avoid this limitation of NC Machine, the CNC machine uses a computer-generated file to store the program which was written by the usage of G-Codes and M-Codes.

Whatever operation you need to change like speed, feed, depth of cut, etc. can be changed in the program instantly and there is no damage to the file as of tape reader. This is the reason, CNC machines are used which are highly accurate compared to NC Machines.

Distributed Numerical Control (DNC Machine)

The DNC Machine is similar to CNC Machine, except a remote computer is used to control no. of machines that can perform no. of operations at a time. Here the central computer or the remote computer communicates with the local CNC computers to do the operation.

Problems Associated with the Conventional NC Machines

- **Mistakes related with part programming** (programming for the parts to be manufactured): When the programs of instructions related to the particular part to be manufactured are written on the punched tape, the syntax or numerical mistakes are quite common. The NC tape is not completed correctly in a single pass and at least three passes are required to get the correct program written. Another major problem with the part programming is achieving the best sequence of steps required for the machining the part.
- **Non optimal speed and feeds**: For most economic manufacturing of the object from the raw material it should be given optimum speed and feeds during manufacturing. The conventional numerical control does not provide opportunity to change the speeds and feeds during the cutting operations, so the programmer is compelled to set the speeds and feeds for the worst-case conditions that can result in highly expensive manufacturing due to wastages, and low quality jobs. This also results in manufacturing of the jobs at lower than optimum productivity.
- **Punched tape**: The punched, which is made up of paper and on which the program is written is the problem in itself. This tape is fragile and susceptible to wear and tear so it has short life and cannot be reliable enough for the repeated use. Instead of paper, other media like Mylar can be used for writing the program of instructions, but these materials are quite expensive.
- **Unreliable tape reader**: The tape reader reads the program of instructions from the punched tape, but it is considered to be highly unreliable hardware component of the NC machine. When the NC machine breaks down the first thing the maintenance personnel checks is the tape reader.
- **The inflexible controller**: The conventional NC machine has the controller unit which is hard wired and the making the changes in the controls of the machines is a tough task. The controller used in the CNC machines is the computer, which is highly flexible.
- Important information: The conventional NC machine cannot provide crucial information to the operator and the supervisor like the number of pieces manufactured, tools changes and others.

Advantages of the CNC Machines

- **Part program tape and tape reader**: In the older CNC machines the part program tape and the tape reader is still required, but they are used only for feeding the program into the memory of the computer. Once the program is saved into the memory, the tape is no more required and the program stored in the memory can be used repeatedly. Thus the tape and the tape reader that poses the major maintenance problems are done away with. In fact the latest CNC machine don't even require the tape and tape reader, for the program of instructions are fed directly into the mini or microcomputer via the control panel of the computer.

- **Editing the program**: Since the program of instructions is saved in the computer memory, they can be edited and changed as per the requirements. Thus the CNC system is highly flexible. One can also make necessary changes in the program for providing variable speeds and feeds for the manufacture of the jobs resulting in economic manufacturing. Even the NC tape used for the programming in CNC machines can be corrected and optimized since it allows changes in the tool path, speed, feed etc.
- **Metric conversion**: The CNC machine allows the conversion of tapes prepared in the metric system into the SI system of measurements. Thus programmer does not have to re-enter the whole program of instructions merely because of the different units of measurements used in the program.
- **Highly flexible**: The CNC machines are highly flexible. One can easily make the changes in the program and store them as the new program. One can also introduce new control options like the new interpolation scheme quite easily. It is easier to make updates in the CNC machines with lesser cost; hence risk of the obsolescence of the CNC machine is reduced.
- **Easier programming**: The programs are written in the CNC machine using language which has statements similar to the ordinary English language statements. The programmer can easily master the CNC programming language and use it for the wide range of the machining operations of the job. The programmer can set the various dimension of the job, the machining operations to be carried out and their sequence, the amount of metal to be removed in each cutting operation, the speed of cutting, etc. The program of instructions is written as per the available size of the raw materials and also the surface finish required for the final finished job. Some of the programs take the form of the macro subroutines stored in the memory of the CNC machine and the programmer can use them frequently whenever required. Some of the programs are stored in the library and they can be used wherever required completely or as a small part of the big program.

CNC MACHINING CENTRES

The term "machining centres" describes almost any CNC milling and drilling machine that includes an automatic tool changer and a table that clamps the workpiece in place.

CNC machine centre is a advance manufacturing machine tool which performs wide range of machining operation with accuracy and good quality surface finish.

The orientation of the spindle is the most fundamental defining characteristic of a CNC machining centres.

Various mechanisms used in CNC machining centers, there main aim is to reduce the production time and gives the best quality results.

- ATC (Automatic tool changer)
- APC (Automatic Pallet changer)
- Feedback systems
- Servo motors systems
- Re-circulating Ball screw and Nut

CNC machining centers can further be classified based on the rotation of either the work piece Or the rotation of the tool as:

- CNC turning machines
- CNC Milling machines

CNC Turning

- The primary function of a CNC Turning Center is that it rotates (or "turns") your workpiece.
- CNC Turning Machines are one of the oldest and simplest forms of machining parts, called "lathes," .
- Can be either horizontal or vertical depending on the weight and tolerance of the workpiece.
- Workpieces for this process are usually round, but can be other shapes — like squares or hexagons.
- The workpiece is held in place by an instrument known as the "chuck." The chuck then spins at various RPMs (depending on the capability of your machine).
- When this occurs, the machine's tool moves into the rotating workpiece and begins to shave away material to create the desired shape.

CNC Milling machines

- The primary function of a CNC Milling Machine is that your tool will be doing the rotating and moving while your workpiece stays in one spot (generally).
- Milling is a more specific process that is similar to drilling and cutting.
- These machines can also be either horizontal or vertical, again depending on the tolerance and weight of your workpiece.
- This process has many axes that allow for a variety of shapes, holes, and slots to be cut into the workpiece at many angles.
- These axes provide many different manoeuvres, either by the spindle or the bed, to cut the part desired to the exact specifications.

CNC FIXTURES

- CNC fixture, also called jig, is a kind of device used to clamp workpiece and guide cutting tool on CNC machine tool.
- CNC fixture can adapt to the high precision, high efficiency, multi-directional processing simultaneously, digital program control and single piece small batch production of computer numerical control machine tools.

- Lathe fixture is to secure the workpiece on the machine through holding in the chuck jaws or fixing to a face plate, and can clamp it quickly and reliably to get into machining or testing, ensure the machining accuracy of CNC parts. The components of a lathe fixture usually including a base, location and clamping devices.

Lathe fixture

The main function of the lathe fixture is for Workholding (from location to clamping).

- **Positioning**: make the workpiece fixed on the correct machining position relative to the cutter and machine tool, and ensure that the machined surface meets the requirements.
- **Clamping**: after the material block is positioned, the force is applied to the workpiece by the clamping device, and the part is fixed and clamped firmly, so as to maintain the correct position during the processing.

Types of CNC Fixtures or Jigs

Based on the applicable machines

- Turning fixtures
- Milling fixtures
- Drilling fixtures
- Boring fixtures
- Grinding fixtures

According to the uses

- Universal fixture
- Special fixture
- Adjustable fixture
- Modular fixture
- Combination fixture

By power source

- Manual fixture
- Pneumatic fixture
- Hydraulic fixture
- Electric fixture
- Magnetic fixture
- Vacuum fixture

CNC Lathe Workholding Methods

There are several ways for holding workpiece on the CNC turning lathe.

- **Chucks**: extensively used for holding CNC parts of various shapes (round, square, hexagonal, and irregular) in machining operations, most commonly applied lathe chucks are three-jaw universal, four-jaw independent and collect chuck.

- **Face plate**: basic lathe workholding accessory, a circular metal plate to be fixed to the end of the lathe spindle, then clamp the CNC lathe parts, generally use T-nuts in slots or threaded holes in the faceplate. This CNC workholding solution is suitable for large part that can't be held in a chuck or between centers.
- **Mandrel**: clamp the internal machined workpiece between centers for further machining operation that concentric with bore, the mandrel types including plain mandrel, expanding mandrel, gang mandrel, stub mandrel.

MODULAR TOOLING

Dedicated fixtures for each workpiece are not desired. As for storing large quantities of dedicated fixtures for possible reuse in the future, there is no practical solution. Either they are scrapped out and new ones are built as needed, or they are stored in space that could be better used for producing parts. Modular tooling systems were designed to solve both of these problems at the same time. They are all kits of tooling components that can be used together in various combinations to locate and clamp workpieces for machining, assembly, and inspection operations.

A modular tool kit consists of mounting plates, angle plates, locators, clamps, and mounting accessories. Adapters also are available to permit the use of many standard and power-workholding devices.

Fixtures made from modular tooling kits can be used on standard machines and numerically controlled (NC) machines as well as CNC machining centers.

Modular tooling systems are valuable when confronted with a short lead time or small production quantities that do not warrant the design and construction of a special jig or fixture.

Construction

- The first step in assembling a jig or fixture is to select a base large enough to handle the workpiece.
- Next, the main structure is constructed with riser blocks and reinforced with stop-thrust elements.
- Finally, the more specialized elements are added to properly locate and clamp the workpiece for machining.

Benefits of modular tooling

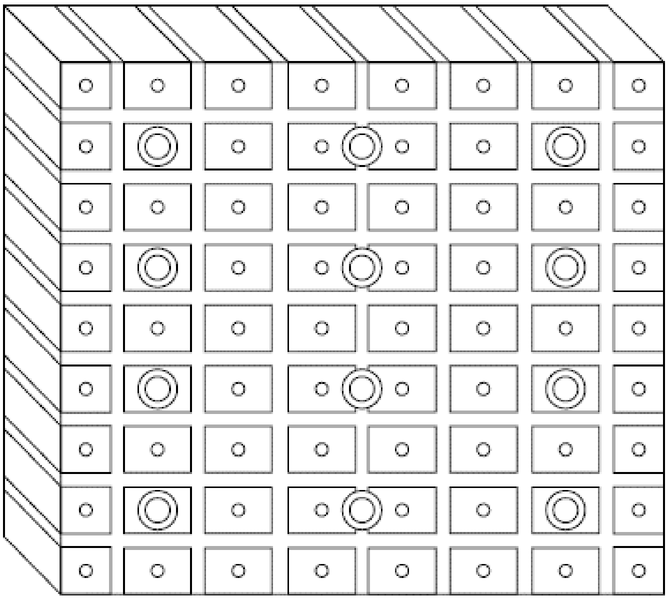
- **Fast changeouts** For traditional cutting tools attached by a square shank tool, a changeout may take as long as two minutes, including measuring to ensure the correct insert position. A quick-change modular cutting tool-holder accessory offers much faster changeouts — sometimes as quick as 20 seconds with the half-turn of a clamp.
- **High repeatability** Accurate tooling solutions should always be a top priority, and a quick-change tooling system shouldn't sacrifice accuracy and repeatability.
- **Versatility in tooling** Between the options in cutting tools, holder extensions and the number of machine adapters, you can load on the tool-holder assembly, modular cutting

tool platforms offer a wide range of tooling task options. This capability to mix and match while maintaining the rigidity of the tool makes it worth adopting a modular tooling strategy. With this versatility, you can take on more jobs and complete them faster through greater machine uptime.

- **Quick ROI** Another critical benefit to modular cutting tools is their relatively quick ROI. With the time savings and increase in production capability through quick change, modular cutting tools, the investment in the system can be earned back quickly — often in a few months.
- **Shop flexibility** With a modular cutting tool platform tool in your shop, you gain a great deal of flexibility. Your sales team can confidently bid on almost any project because of the added capabilities that modular cutting tools add for performing almost any tooling task.

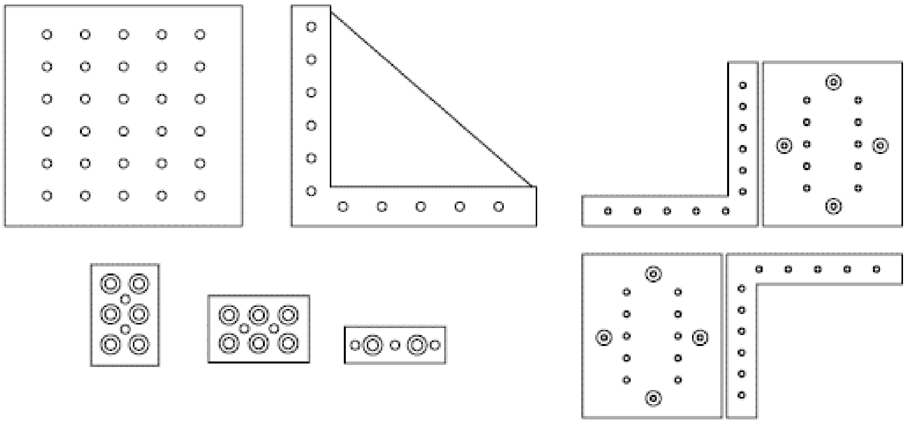
MODULAR FIXTURES

Modular fixturing is used for quickly rigging up a fixture for a specific component using the off-the-shelf components. Modular elements are available such as grid plates shown in following figure.



Grid plates are generally used as one of the fixturing bases. The grid plates are provided with precisely drilled and tapped holes to facilitate the clamping operation. Since the holes on these grid plates are made at precise positions, the operator would know the exact location of the component depending upon where he is clamping. These grid plates can be permanently clamped on the machine tool table if necessary.

In addition to these standard fixture bases, a large number of fixture elements such as angle blocks, base elements, locators, and clamping elements (see figure) are available for assembling a fixture.



Depending upon the outer contour of the component, it is possible to identify the various elements required from the stock elements and assemble a fixture. Since these elements are reusable, the overall cost of fixturing is less.

ASSIGNMENT

CUTTING TOOLS

Q.1.(AMIE S15, 19, 20, W18, 5 marks): What are the basic requirements of a cutting tool? What are the purposes of rake and relief angles? Name and describe three basic types of chips.

Q.2.(AMIE S15, 5 marks): Why does a built-up edge (BUE) chip produce poor surface finish?

Q.3.(AMIE S15, 5 marks): Why is it possible to remove metal more efficiently at heavy feeds heavy depth of cut?

Q.4.(AMIE W17, 8 marks): Sketch a single-point turning tool and show its rake angles, clearance angles and cutting angles.

BROACHING

Q.5.(AMIE S17, 18, 12 marks): Discuss the design parameters for designing a broach by rising-tooth method.

HOLE OPERATION

Q.6.(AMIE S15, 5 marks): What is the major disadvantages in using toolmaker's buttons for precision hole location?

Q.7.(AMIE S15, W19, 5 marks): Draw a drill bit neatly showing its important features on it.

Q.8.(AMIE S17, 18, 8 marks): What are the major problems associated with boring operation? Discuss the different types of standard boring bars available, mentioning the specific usage, advantages and disadvantages of each one of them.

Q.9.(AMIE W19, 4 marks): Distinguish between boring and external turning.

MOULDING

Q.10.(AMIE W15, 6 marks): What are the advantages of injection moulding?

Q.11.(AMIE W15, 6 marks): Describe any three types of gates used in injection moulding with a neat sketch.

Q.12.(AMIE W15, 8 marks): Discuss the following with reference to injection moulding:

(i) Mould filling

(ii) Venting

(iii) Mould cooling

(iv) Distance between holes

EDM

Q.13.(AMIE S16, 10 marks): What is electric discharge machining? What is the electrode material generally used in EDM process?

CNC

Q.14.(AMIE S19, 10 marks): Explain the two basic systems of numerical control in use today. What are the factors considered when designing fixtures for NC machine tools?

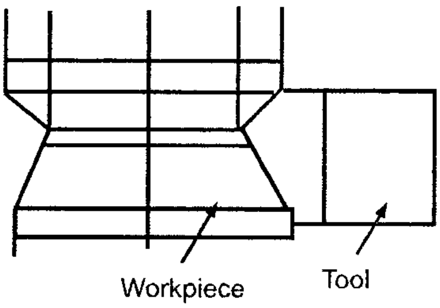
Q.15.(AMIE S17, W17, 8 marks): State the advantages of numerically controlled machine tools with respect to conventional machine tools.

Q.16.(AMIE W18, 10 marks): What is a N/C machining centre? How are lathe chucks used as holding fixtures in N/C machine tools?

Q.17.(AMIE S20, 6 marks): How are lathe chucks used as holding fixtures in N/C machine tools?

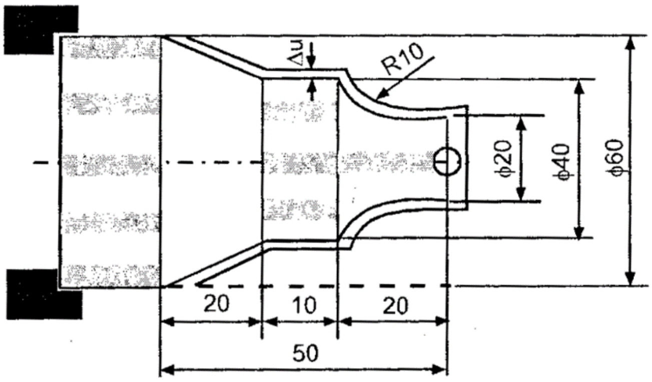
Q.18.(AMIE W15, 10 marks): What is modular tooling? List the benefits gained by using modular tooling.

Q.19.(AMIE S16, 5 marks): Design a circular form tool for the component shown in figure given. Assume suitable dimension.



Q.20.(AMIE S16, 15 marks): Select suitable turning inserts and tool holders for CNC applications for the component shown in figure. Workpiece: medium carbon steel.

The specific cutting resistance of the material is 3000 N/mm^2. Specify the tool shank dimension, geometry and ISO designation of tool holder shown in figure. Take cutting speed as 300 m/min. Take E = 2.1 x 10^5 N/mm^2.



Q.21.(AMIE W17, 6 marks): How are the grooves (circular) produced in the outer and inner races of single row ball bearings? Describe briefly with the help of suitable sketches.

Universal internal grinder is used to generate circular grooves in the outer and inner races of single row ball bearings.

JIGS & FIXTURES

Q.22.(AMIE W15, S20, 10 marks): What are the differences between jigs and fixtures and their advantages in mass production?

Q.23.(AMIE W17, 8 marks): Describe basic principles of jig and fixture design.

Q.24.(AMIE S20, 6 marks): How was the term "fixture" derived?

Q.25.(AMIE S17, 4 marks): What basic features of the part are considered for jig design?

Q.26.(AMIE S17, 6 marks): Mention any five standards jigs used in industry? Explain the design features and specific applications of a box jig.

Q.27.(AMIE S15, 18, W15, 5 marks): What are the basic rules for location? Explain the 3-2-1 location principle.

Q.28.(AMIE W15, 5 marks): What are the basic rules for applying classing forces?

Q.29.(AMIE S15, 19, 5 marks): What is the advantages of a toolmaker's clamp compared to a C-clamp.

Q.30.(AMIE S15, 17, 20, W18, 19, 10 marks): Define locating and clamping. Explain the locating and clamping principles.

Q.31.(AMIE S20, 8 marks): What are diamond pins, and how are they used?

Q.32.(AMIE S19, 10 marks): What are the major factors that determine how a workpiece will be located? What are the essential requirements of clamps and clamping devices?

Q.33.(AMIE S20, 6 marks): Why should complicated clamping devices be avoided?

Q.34.(AMIE W15, 10 marks): Explain briefly the essentials of welding fixtures and inspection fixtures.

Q.35.(AMIE S16, 5 marks): What is meant by degree of freedom and how are they arrested with reference to jig and fixtures?

Q.36.(AMIE W18, 6 marks): How is a face plate fixture located on the lathe faceplate?

Q.37.(AMIE W19, 8 marks): Explain the use of face plate as a turning fixture with a suitable example.

Q.38.(AMIE S16, 15 marks): Enumerate the design principles for drilling jigs. What are the main types of jigs? Discuss these with the help of suitable sketches.

Q.39.(AMIE W19, 4 marks): What are the requirements of a good drilling jig?

Q.40.(AMIE W17, 6 marks): What rules should be followed during designing a drilling jig for location and supporting?

Q.41.(AMIE S17, 4 marks): Why should a drill jig stand on four legs rather than three?

Q.42.(AMIE S17, 18, 12 marks): Explain the construction of the back plate for turning fixtures with their specific applications. Also discuss various chucks used as turning fixtures with their specific applications.

Q.43.(AMIE W17, 19, S18, 7 marks): Discuss the characteristics and uses of modular fixture and tools in manufacturing.

Q.44.(AMIE S18, 6 marks): Explain the design features and specific applications of a box jig.

Q.45.(AMIE W18, 10 marks): What is the advantage of tumble jigs? What are the three ways chips removed from drill jigs? What is the advantage of a cast constructed jig body?

Q.46.(AMIE S19, 10 marks): Describe the steps involved in designing a drill jig to drill 6 equidistant holes in a disk of width 10 mm. All other dimensions can be assumed as per the requirement.

Q.47.(AMIE S20, 6 marks): How is the grinding spindle of a jig grinder powered?

Q.48.(AMIE S16, 5 marks): Name the essential features of a milling fixture.

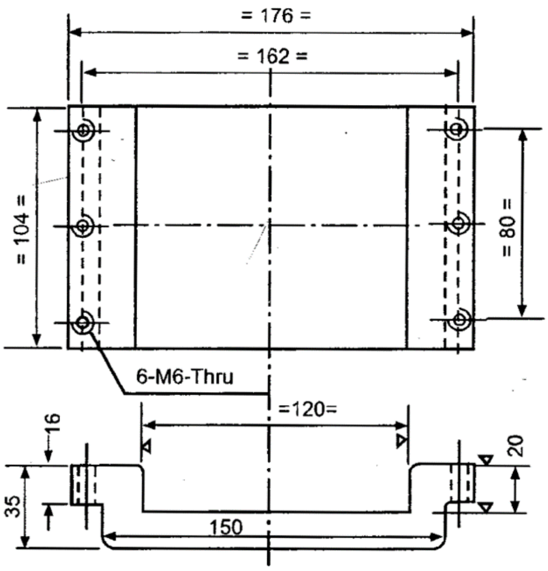
Q.49.(AMIE W19, 6 marks): What are the major advantages of magnetic and vacuum milling fixtures?

Q.50.(AMIE S17, 18, 6 marks): Explain the steps followed in the production of precision dowel holes.

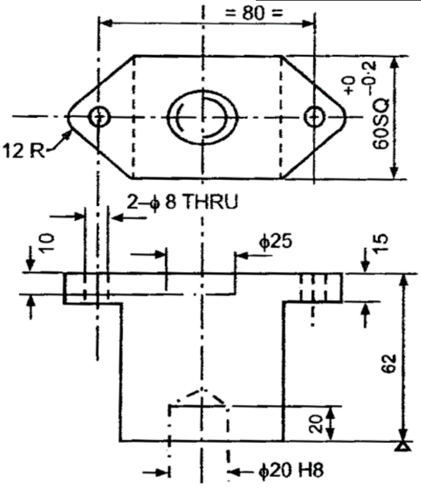
Q.51.(AMIE W18, 8 marks): Why dowel holes are generally made "through holes"?

Q.52.(AMIE S16, 15 marks): Design a suitable milling fixture to cut a keyway of size 4 mm x 4 mm on a solid shaft of ϕ 30 mm diameter and 100 mm length. Explain the use of tenon and setting block with sketch in milling fixture.

Q.53.(AMIE S15, 10 marks): Design a milling fixture for the component shown in figure below.



Q.54.(AMIE S15, 10 marks): Design a drilling jig for the component shown in figure below.



*(For online support such as eBooks, video lectures, unsolved papers, online objective questions, test series etc., visit **www.amiestudycircle.com**)*