Instructor Lab Manual

Machine Tool and Machining

Department of Mechanical Engineering
University of Engineering and technology
Lahore

Lab Session NO 1

To perform basic machining operations on Lathe machine.

Lathe:

A lathe is a machine tool that rotates a workpiece about an axis of rotation to perform various operations such as cutting, sanding, knurling, drilling, deformation, facing, and turning, with tools that are applied to the workpiece to create an object with symmetry about that axis.

Types of Lathe Machine Operations:

The lathe machine operations are classified into three main categories and are as follows.

Following are the Lathe machine operations done either by holding the workpiece between centers or by a chuck:

1. Turning Operation
   - Plain or Straight Turning
   - Rough Turning
   - Shoulder Turning
   - Taper Turning
   - Eccentric Turning
2. Facing Operation
3. Chamfering Operation
4. Knurling Operation
5. Thread cutting Operation
6. Filing Operation
7. Polishing Operation
8. Grooving Operation
9. Spinning Operation
10. Spring Winding
11. Forming

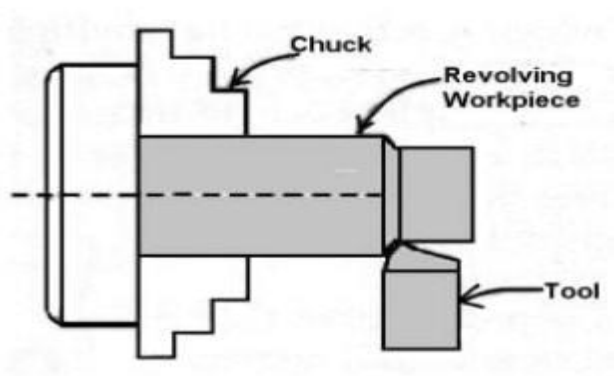
Lathe machine operations which are performed by holding the work by a chuck or a faceplate or an angle plate are:

1. Drilling
2. Reaming
3. Boring
4. Counter boring
5. Taper boring
6. Tapping
7. Undercutting
8. Internal thread cutting
9. Parting-off

1. Operations Done by Holding Workpiece Between Centres:

Turning:

It is the most common type of operation in all lathe machine operations. Turning is the operation of removing the excess material from the workpiece to produce a cylindrical surface to the desired length.



Turning Operation

The job held between the centre or a chuck and rotating at a required speed. The tool moves in a longitudinal direction to give the feed towards the headstock with proper depth of cut. The surface finish is very good.

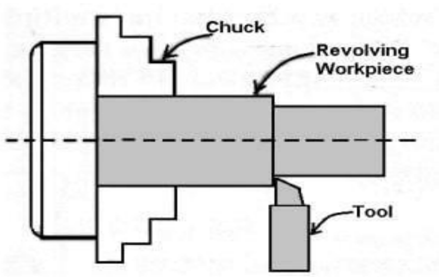
1. Straight Turning:

The workpiece is held on the chuck and it is made to rotate about the axis, and the tool is fed parallel to the lathe axis. The straight turning produces a cylindrical surface by removing excess metal from the workpiece.

2. Rough Turning:

It is the process of removal of excess material from the workpiece in minimum time by applying high rate feed and heavy depth of cut. in rough turning the average depth of cut 2mm to 4mm can be given and feed is from 0.3 to 1.5mm per revolution of the work.

3. Shoulder Turning:



Shoulder Turning

When a workpiece has different diameters and is to be turned, the surface forming steps from one diameter to the other is called the shoulder, and machining this part of the workpiece is called shoulder turning.

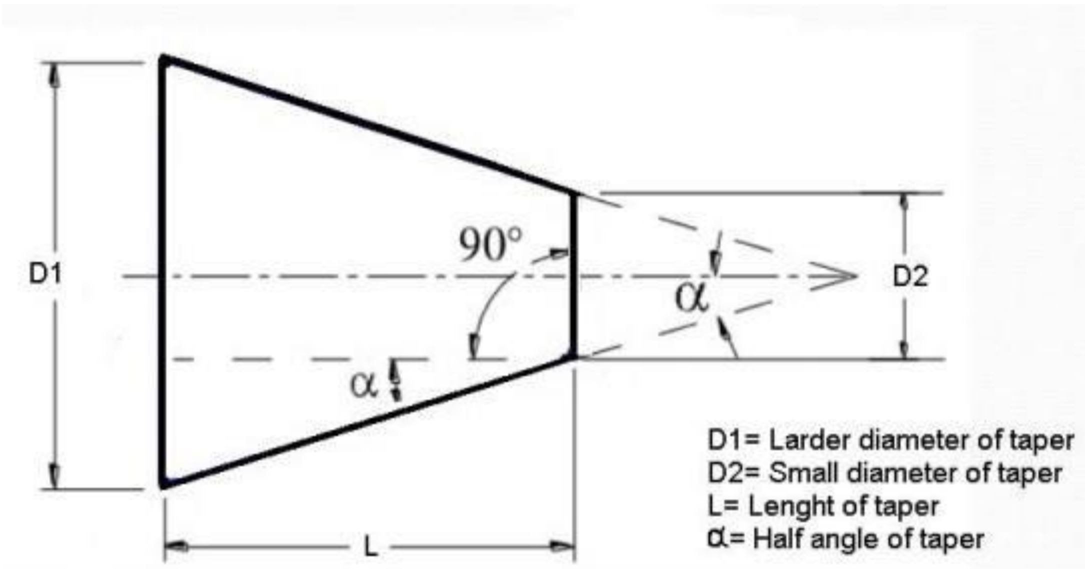
Eccentric turning:

When a cylindrical surface two separate axis of rotation, with the first axis, is offset to the other axis then such a workpiece is machined by the operation called eccentric turning. Here three sets of centre holes are drilled.

By holding the workpiece at these three centres the machining operation for each of the surface can be completed.

Taper Turning:

- A ”taper” is the uniform increase or decrease in the diameter of the workpiece and measured along with its length.
- Taper turning means to produce a conical shape by a gradual reduction in diameter from a cylindrical workpiece.



The amount of taper in the workpiece is usually specified on the basis of the difference in diameter of the taper to its length. It is known as a cone and it is indicated by the letter K.

It has the formula K = D-d / 1 to produce the taper on the workpiece.

- D = Larger diameter of taper.
- d = Small diameter of taper.

In the case of a lathe, the taper on a given workpiece is obtained by tuning the job and feeding the tool at an angle to produce a gradual increase or decrease in the diameter of the workpiece.

- The two important types of tapers are,

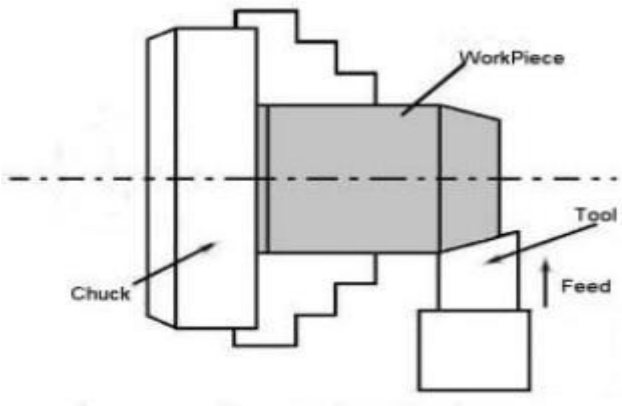
- ”More taper” here, the angle is very small and varies from 1.4 to 1.5°.
- ”Metric taper” is available in seven standard sizes with standard taper angles.

➢ Methods of taper turning,

- Form tool method
- Combined feeds method
- Compound rest method or swivelling compound rest method
- Tailstock set over method
- Taper turning attachment method

1. Form tool method

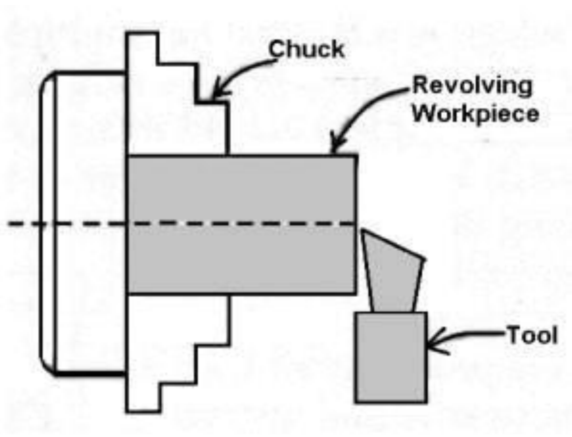
Here the taper length obtain is equal to the width of the form tool. To obtain the required size of the taper the form tool is fed slowly straight into the workpiece by operating the cross slide perpendicular to the lathe axis.



Taper Turning Using Form Tools.

This is the simplest method of taper turning. It is limited to obtain small taper length such as chamfering the side of the workpiece. The method is done at a faster rate.

Facing:

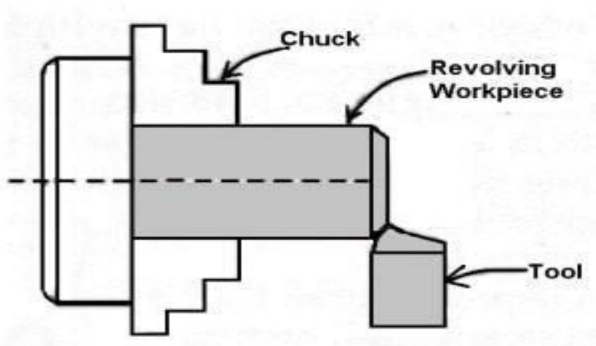


Facing Operation

It is an operation of reducing the length of the workpiece by feeding the perpendicular to the lathe axis. This operation of reducing a flat surface on the end of the workpiece. For this operation, regular turning tool or facing tool may use. The cutting edge of the tool should set to the same height as the centre of the workpiece.

- Facing consist of 2 operations
  - Roughing: Here the depth of cut is 1.3mm
  - Finishing: Here the depth of cut is 0.2-0.1mm.

Chamfering operation:

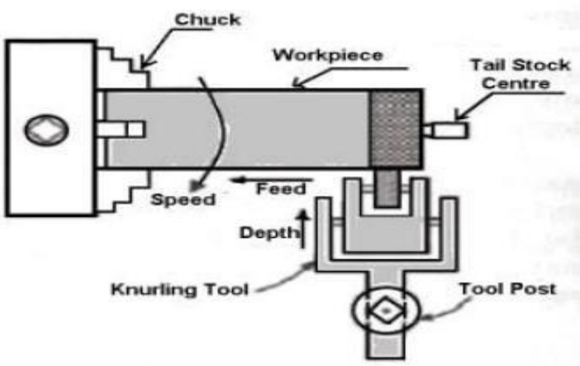


Chamfering

It is the operation of getting a bevelled surface at the edge of a cylindrical workpiece. This operation is done in case of bolt ends and shaft ends. Chamfering helps to avoid damage to the sharp edges and protect the operation getting hurt during other operations. Chamfering on bolt helps to screw the nut easily.

Knurling operation:

It is an operation of obtaining a diamond shape on the workpiece for the gripping purpose. This is done to provide a better gripping surface when operated by hands. It is done using a knurling tool. The tool consists of a set of hardened steel roller, and it is held rigidly on the toolpost.

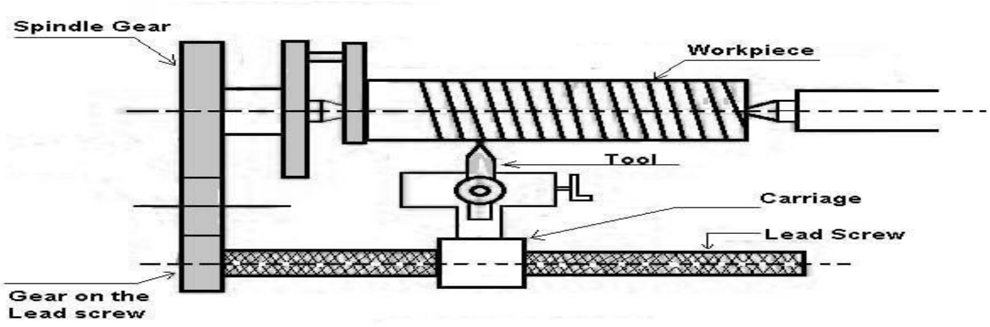


Knurling

Knurling is done at the lowest speed available on a lathe. It is done on the handles and also in case of ends of gauges. The feed varies from 1 to 2 mm per revolution. Two or three cuts may be necessary to give the full impression.

Thread cutting:

It is the important operation in the lathe to obtain the continuous "helical grooves" or "threads". When the threads or helical grooves are formed on the out surface of the workpiece is called external thread cutting. When the threads or helical grooves are formed on the inner surface of the workpiece is called internal thread cutting. The workpiece is rotating between the two centres i.e., live centre and dead centre os the lathe.



Thread Cutting

Here the tool is moved longitudinally to obtain the required type of the thread. When the tool is moved from right to the left we get the left-hand thread. Similarly, when the tool is moved from left to the right we get the right-hand thread.

Here the motion of the carriage is provided by the lead screw. A pair of change gears drives the lead screw and by rotating the handle the depth of cut can be controlled.

Filling:

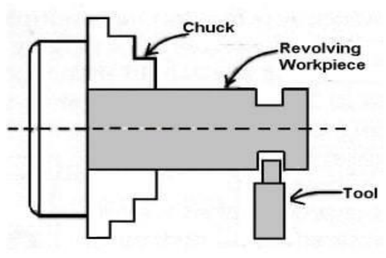
It is the finishing operation performed after turning. This is done on a lathe to remove burrs, sharp corners, and feed marks on a workpiece and also to bring it to the size by removing the very small amount of metal.

The operation consists of passing a flat single-cut file over the workpiece which revolves at a high speed. The speed is usually twice that of turning.

Polishing:

This operation is performed after filing to improve the surface quality of the workpiece. Polishing with successively finer grades of emery cloth after filing results in a very smooth, bright surface. The lathe is run at high speeds from 1500 to 1800m per min, and oil is used on the emery cloth.

Grooving:



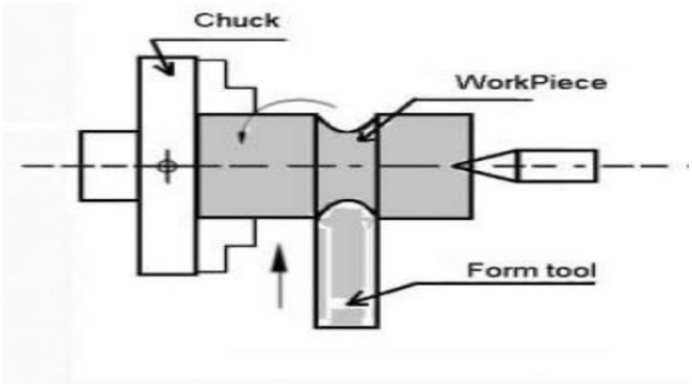
Grooving

It is the process of reducing the diameter of a workpiece over a very narrow surface. It is done by a groove tool. A grooving tool is similar to the parting-off tool. It is often done at the end of a thread or adjacent to a shoulder to leave a small margin.

Forming:

It is the process of turning a convex, concave or of any irregular shape. Form-turning may be accomplished by the following method:

1. Using a forming tool.
2. Combining cross and longitudinal feed.
3. Tracing or copying a template.



Forming Operation

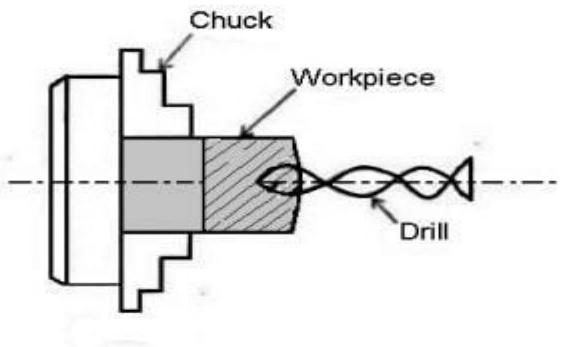
Forming tools are not supposed to remove much of the material and is used mainly for finishing formed surfaces. Generally, two types of forming tools are used straight and circular. Straight type is used for wider surface and the circular type for narrow surfaces.

2. Operations Done By Holding The Work By A Chuck

Lathe machine operations performed by holding the work by a chuck or a faceplate or an angle plate are:

Drilling:

Drilling is the operation of producing a cylindrical hole in a workpiece. It is done by a rotating tool, the rotating side of the cutter, known as drilling drill.In this operation, The workpiece is revolving in a chuck or a faceplate and the drill is held in the tailstock drill holder or drill chuck.

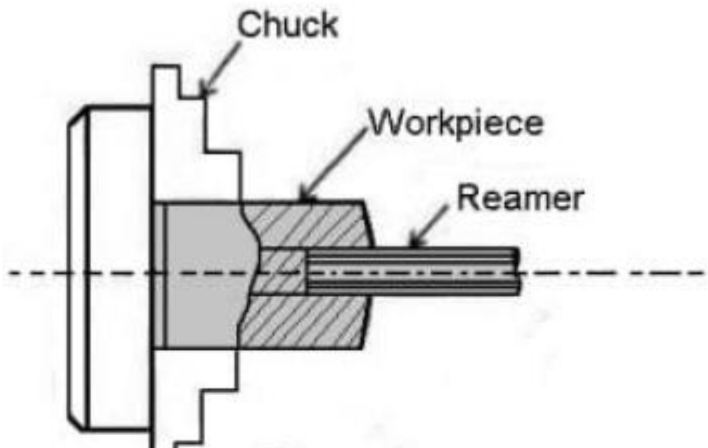


Drilling

The feeding is adopted is affected by the movement of the tailstock spindle. This method is adopted for the drilling regular shaped workpiece.

Reaming:

Reaming is the operation of finishing and sizing a hole which has been already drilled or bored. The tool is used is called the reamer, which has multi-plate cutting edges.

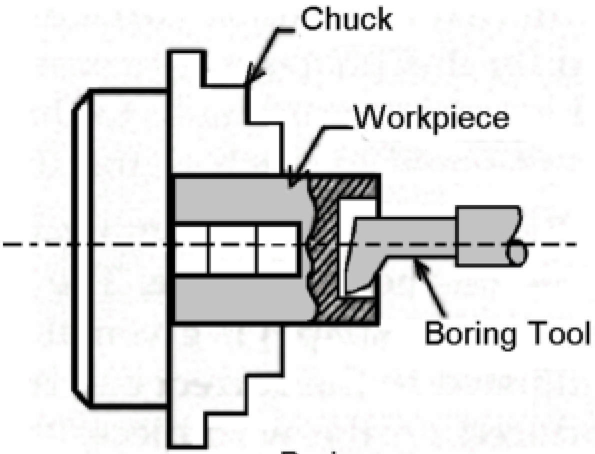


Reaming

The reamer is held on the tailstock spindle, either directly or through a drill chuck and is held stationary while the work is revolved at a very slow speed.

Boring:

Boring is the operation of enlarging the hole which is already drilled, punched or forged. It cannot produce a hole. Boring is similar to the external turning operation and can be performed in a lathe. In this operation, the workpiece is revolved in a chuck or a faceplate and the tools which are fitted to the tool post is fed into the work.



Boring

It consists of a boring bar having a single-point cutting tool which enlarges the hole. It also corrects out of roundness of a hole. This method adopted for boring small-sized works only. The speed of this process is slow.

Counter boring:

Counter boring is the operation of enlarging the end of the hole through a certain distance. It is similar to a shoulder work in external turning.

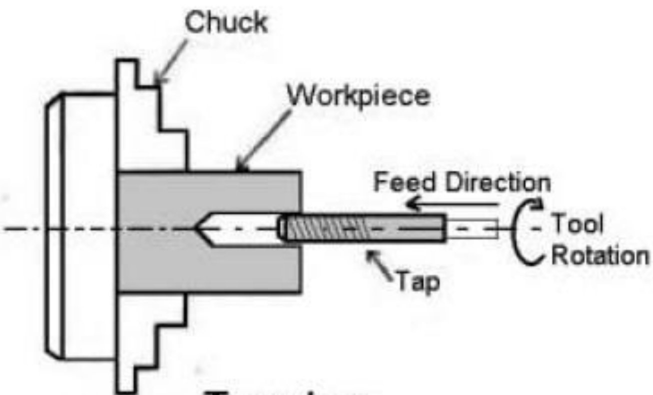
The operation is similar to boring and plain boring tools or a counter bore may be used. The tool is used called a counter bore. The speed is slightly less than drilling.

Taper Boring:

The principle of turning a tapered hole is similar to the external taper turning operation and is completed by rotating the work on a chuck or a faceplate. The feeding tool is at an angle to the axis of rotation of the workpiece.

A boring tool is mounted on the tool post and by swiveling the compound slide to the desired angle, a short taper hole is machined by hand feeding.

Tapping:



Tapping

Tapping is the operation of cutting internal threads of small diameter using a multipoint cutting tool called the tap. In a lathe, the work is mounted on a chuck or on a faceplate and revolved at a very slow speed. A tap of required size held on a special fixture is mounted on the tailstock spindle.

Undercutting:

Undercutting is similar to grooving operation when performed inside a hole. It is the process of boring a groove or a large hole at a fixed distance from the end of a hole.

This is similar to the boring operation, except that a square nose parting is used. Undercutting is done at the end of an internal thread or a counter bore to provide clearance for the tool or any part.

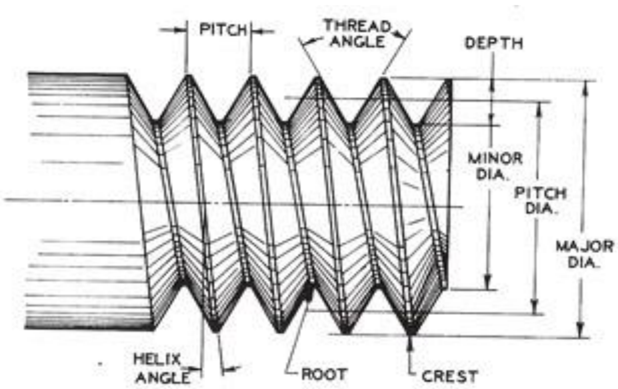
Lab Session No 2

To perform thread cutting operation on lathe in both with worth and metric system.

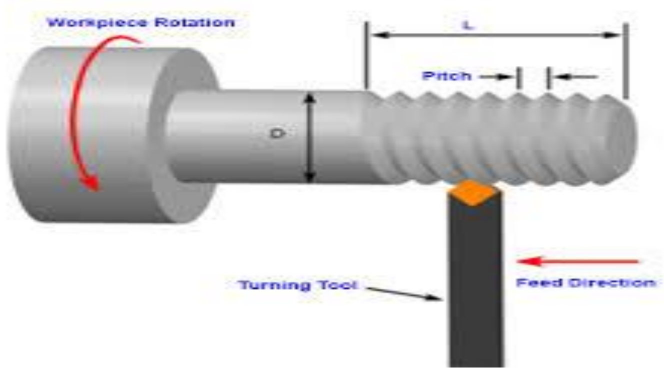
Theory:

As we became familiar with threads, let us talk about the lathe. A lathe is a machine which is used to shape wood, metal, etc. by removal of material. A lathe can perform various operations such as turning, knurling, facing, etc. In a lathe, a workpiece is being rotated along an axis of rotation and different machine tools are used to perform different operation on the workpiece.

Some basic terminologies of thread are:



- Pitch: It is defined as the distance between two points starting from one point on one thread to another point at a similar location on the consecutive thread, measured parallel to the axis of the cylinder.
- Lead: It is the distance moved by a thread in an axial direction on one complete revolution.
- Major Diameter: It is the largest diameter of the thread measured from one crest to the other crest in opposite direction.
- Minor Diameter: It is the smallest diameter of the thread measured from one root (trough) to another root in opposite direction.
- Threads per inch (tpi): As the name suggests it is the number of threads in an inch. It is being measured by placing scale alongside the thread and counting no. of pitches.
- Depth: The distance measured between a crest and a root in the perpendicular direction to the axis of the cylinder.



Metric Threads On Lathe:

Step #1. Set the compound slide to the same angle as the trailing flank of the thread to be cut. For a Unified National thread this angle is 30°, as shown in Illustration #1. (NOTE: This illustration is for cutting right hand external or left hand internal threads. For cutting left hand external or right hand internal threads, the compound should be set to 30° in the opposite direction.) Use the protractor markings on the compound slide for this.

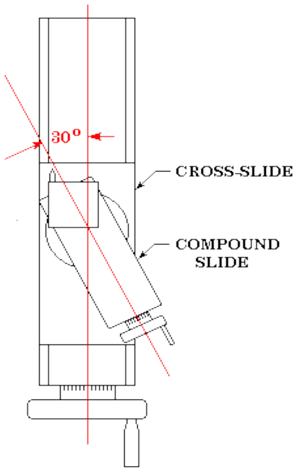


Illustration #1: Lathe Cross-Slide and Compound Slide Set Up for Threading

Step #2. Align the cutting tool using a threading tool gauge such as the one shown in Illustration #2. One edge of the threading tool gauge should be held up against any cylindrical surface of the workpiece. The tool holder should first be adjusted to the proper center height and then rotated left or right to visually align the cutting edges of the tool with one of the V-shaped slots of the gauge.

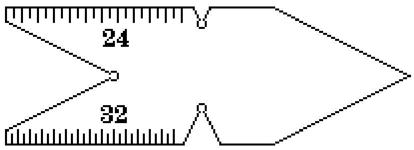


Illustration #2: Threading Tool Grinding and Alignment Gauge

Step #3. Use the headstock gear change levers to obtain the proper pitch for the thread to be cut. This is usually marked on a plate attached to the lathe headstock. Double check the change gears inside the headstock cover if you are unsure about which change gear ratio has been set up.

Step #4. Check to be sure the threading dial (normally found on the right hand side of the carriage, as shown in Illustration #3) is engaging the lead screw. (NOTE: To prevent excessive wear to the lead screw, it is considered good practice to disengage the threading dial from the lead screw when performing other lathe operations.)

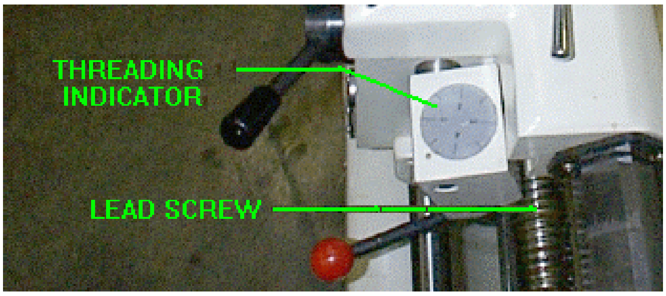


Illustration #3: Threading Dial

Step #5. Screw in the cross-slide until the tip of the cutting tool almost touches the surface to be threaded. Set the dial on both the cross-slide and the compound slide to zero.

Step #6. After making sure the workpiece is properly secured, start the spindle and workpiece rotating. The RPM should be set to a value considerably slower than that used for turning or facing. The surface finish, which can be poor if too low an RPM is used, can be improved by using an appropriate cutting fluid.

Step #7. With the tool at the right hand end of thread to be cut, move the tool in a few thousandths of an inch by screwing in the compound slide. Do not move the cross-slide in, leave it set to zero.

Step #8. Engage the thread engagement lever just as the number "1" on the threading dial comes around into alignment with the index mark on the frame. Be prepared for a much faster carriage traverse velocity than is normally encountered when turning or boring as the thread helix is generated.

Step #9. Disengage the thread engagement lever to stop the carriage when the tool exits the end of the cut and enters into the breakout area.
Step #10. Without moving the compound slide, withdraw the tool from the cut by moving the cross-slide away from the workpiece.
Step #11. Move the carriage back to the right hand end of the workpiece and screw the cross-slide back in to the previously established zero.
Step #12. Move the tool in toward the workpiece by screwing in the compound slide a few more thousandths of an inch.
Step #13. Repeat steps #8 through #12 until the thread is cut to size. Note that for some thread pitches, numbers on the thread dial other than "1" can also be used, but "1" will always work for all thread pitches.
Step #14. The thread can be checked by:
(a). using the three-wire method and a standard outside micrometer.
(b). using a special thread-type micrometer.
(c). screwing on a nut - this is the least accurate method but is suitable for most maintenance work if care is taken to remove all burrs on the threads.

Whitworth Threads On Lathe:

The process for single point turning threads in the design lab is as follows:

1. Clamp the part in the lathe using a live center if necessary.
2. Turn the OD to the target major diameter and include a chamfer on the end at least 0.020” smaller than the minor diameter of the thread profile to be cut.

3. If permissible, cut a thread relief using a grooving tool (as shown in Figure 10 and the two video thumbnail images above). The thread relief should be slightly less than the minor thread diameter.
4. Adjust the gearbox levers on the front of the headstock to cut the proper thread pitch.
5. Adjust the threading tool so it is aligned parallel to the X-axis.
6. Touch off on the part and zero the X-axis.
7. Cut a light (0.001 - 0.002”) scratch pass across the surface of the part to be checked with a thread gage for accuracy.
8. If the pitch of the scratch pass measures correctly, begin cutting the thread to depth; start with deeper depths of cut (.010” in aluminum, 0.005” in steel) and make progressively shallower cuts as the thread gets deeper and the threading tool begins to leave a worse finish)
9. As you approach final thread size, use a fine file to carefully debur the rough edges of the major diameter (unless using a full profile insert, which deburrs the major diameters automatically, as discussed above). The major diameter should end up a few thousandths of an inch under the nominal size, according to the tolerances listed in the Machinery Handbook (e.g. 0.4906-0.4987” for a ½-20 UNF 2A thread). You will know when you are close to the final size by keeping track of your X-infeed value, which will end up smaller than the equivalent internal thread’s minimum minor diameter by the noted allowance (e.g. 0.446” – 0.0013” = 0.4447” for the same ½-20 UNF 2A thread).
10. The procedure for making an actual cut is:
    a. Check the direction of the threading direction by engaging the half-nut with the tool a safe distance from the part; for this example, we will thread toward the chuck
    b. Adjust the spindle speed to a low setting (100-300 rpm) depending on how brave you are J
    c. Position the tool in a safe starting location to begin cutting the thread
    d. Advance the tool toward the part the distance (depth of cut) you wish to cut
    e. Engage the half-nut for threading (it’s safest to leave this engaged for the duration of the threading session)
    f. Turn the spindle ON in the FWD direction and allow the tool to make a cut
    g. Turn the spindle OFF before the tool reaches a shoulder (if not exists); you can use the foot brake to stop it quickly if needed; if you stop too early, simply bump the power switch to continue the cut or rotate the chuck by hand

h. Retract the tool a safe distance from the part in the X-direction

i. Turn the spindle ON in the REV direction to allow the tool to return to a safe starting location

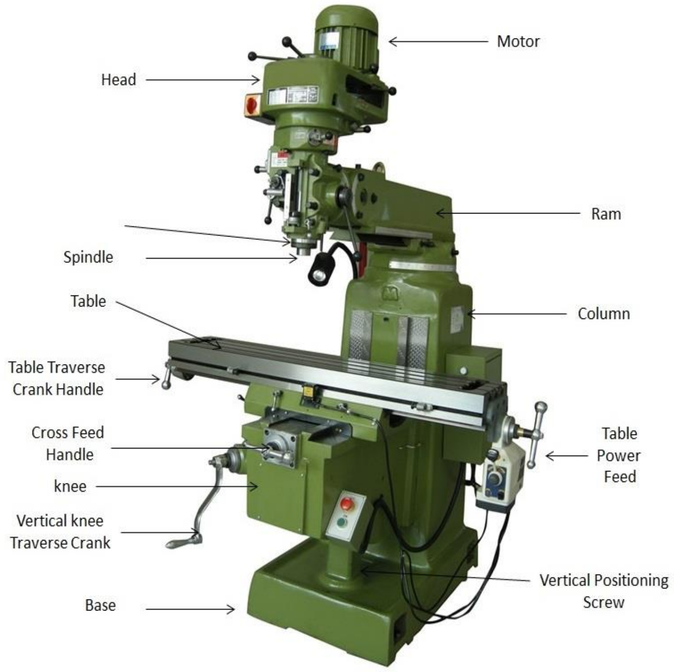
j. Repeat steps d. thru i. until the desired minor or pitch diameter is reached.

Lab Session No 3

Detail demonstration on executing milling functions(face & power) on a given metallic piece.

Milling Machine

Milling is a process performed with a machine in which the cutters rotate to remove the material from the work piece present in the direction of the angle with the tool axis. With the help of the milling machines one can perform many operations and functions starting from small objects to large ones.



Milling Machine Operations:

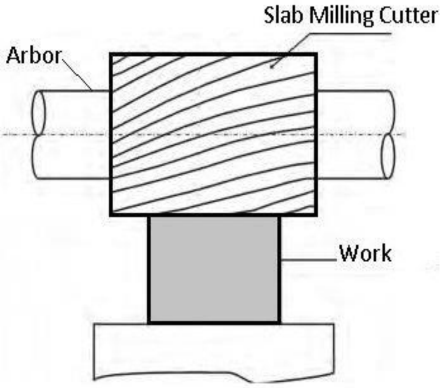
The 15 different types of milling machine operations are as follow:

1. Plain Milling Operation
2. Face Milling Operation
3. Side Milling Operation
4. Straddle Milling Operation
5. Angular Milling Operation
6. Gang Milling Operation
7. Form Milling Operation
8. Profile Milling Operation
9. End Milling Operation
10. Saw Milling Operation

11. Milling Keyways, Grooves and Slot
12. Gear Milling
13. Helical Milling
14. Cam Milling
15. Thread Milling

Types of Milling Machine Operations:

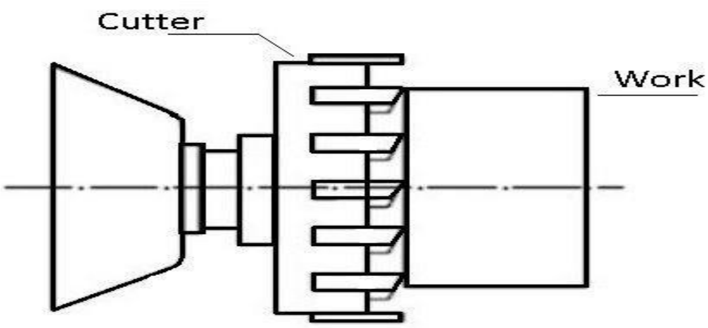
1-Plain Milling



Plain Milling Operation

- The plain milling is the most common types of milling machine operations.
- Plain milling is performed to produce a plain, flat, horizontal surface parallel to the axis of rotation of a plain milling cutter.
- The operation is also known as slab milling.
- To perform the operation, the work and the cutter are secured properly on the machine.
- The depth of cut is set by rotating the vertical feed screw of the table. And the machine is started after selecting the right speed and feed.

2-Face Milling



Face Milling Operation

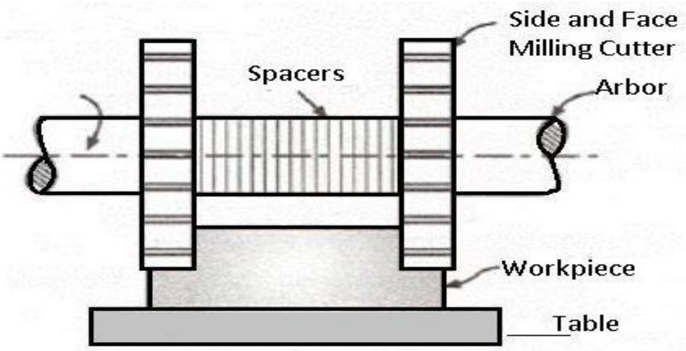
- The face milling is the simplest milling machine operations.
- This operation is performed by a face milling cutter rotated about an axis perpendicular to the work surface.

- The operation is carried in plain milling, and the cutter is mounted on a stub arbor to design a flat surface.
- The depth of cut is adjusted by rotating the crossfeed screw of the table.

3-Side Milling

- The side milling is the operation of producing a flat vertical surface on the side of a workpiece by using a side milling cutter.
- The depth of cut is set by rotating the vertical feed screw of the table.

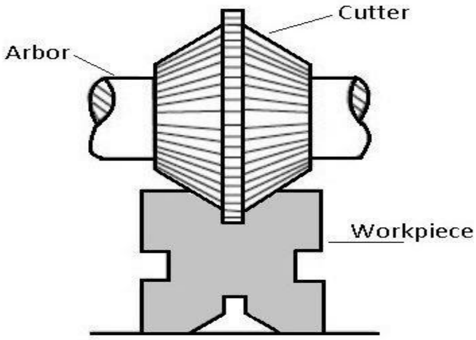
4-Straddle Milling



Straddle Milling Operation

- The straddle milling is the operation of producing a flat vertical surface on both sides of a workpiece by using two side milling cutters mounted on the same arbor.
- Distance between the two cutters is adjusted by using suitable spacing collars.
- The straddle milling is commonly used to design a square or hexagonal surfaces.

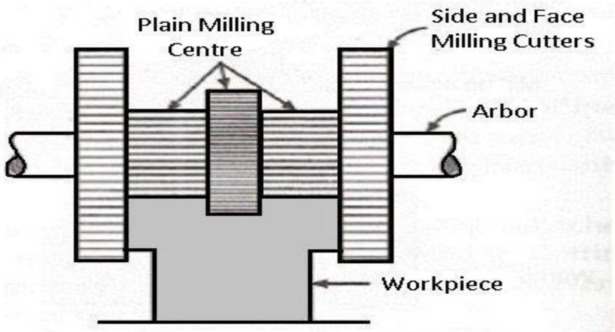
5-Angular Milling



Angular Milling

- The angular milling is the operation of producing an angular surface on a workpiece other than at right angles of the axis of the milling machine spindle.
- The angular groove may be single or double angle and may be of varying included angle according to the type and contour of the angular cutter used.
- One simple example of angular milling is the production of V-blocks.

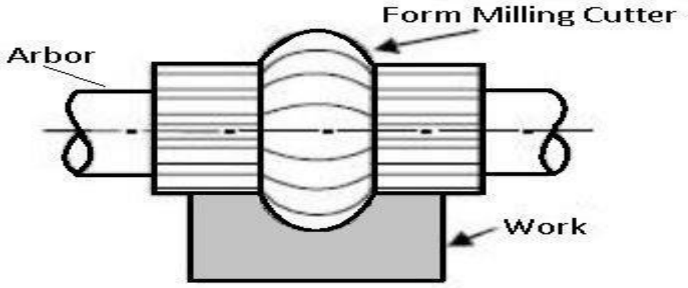
6-Gang Milling



Gang Milling Operation

- The gang milling is the operation of machining several surfaces of a workpiece simultaneously by feeding the table against a number of cutters having the same or different diameters mounted on the arbor of the machine.
- The method saves much of machining time and is widely used in repetitive work.
- Cutting speed of a gang of cutters is calculated from the cutter of the largest diameter.

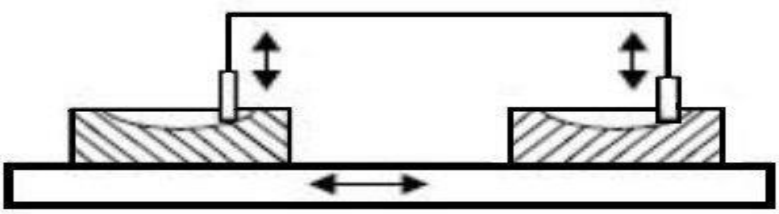
7-Form Milling



Form Milling Operation

- The form milling is the operation of producing the irregular contour by using form cutters.
- The irregular shape may be convex, concave, or of any other shape. After machining, the formed surface is inspected by a template gauge.
- Cutting rate for form milling is 20% to 30% less than that of the plain milling.

8-Profile Milling

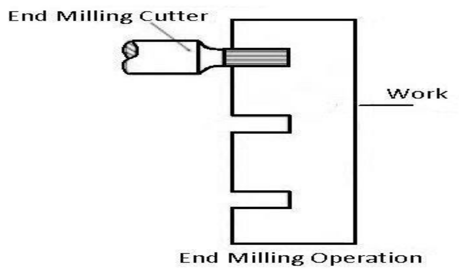


Profile Milling Operation

- The profile milling is the operation of reproduction an outline of a template or complex shape of a master dies on a workpiece.

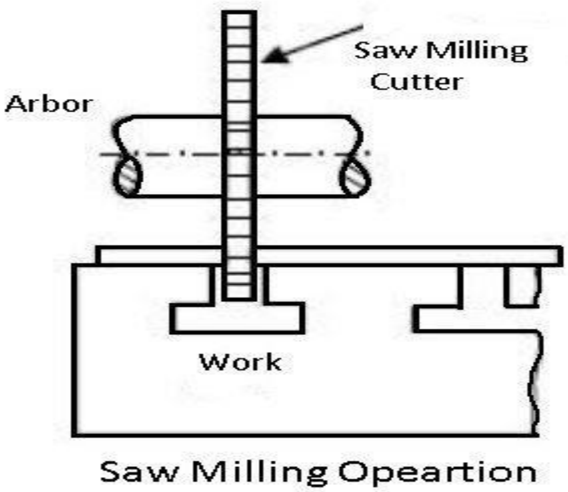
- Different cutters are used for profile milling. An end mill is one of the widely used milling cutters in profile milling work.

9-End Milling



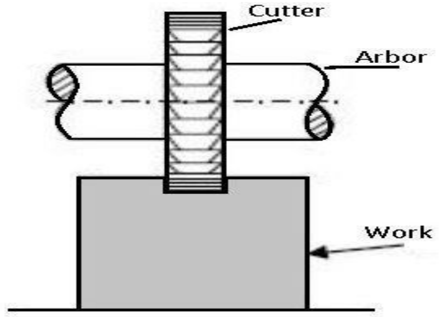
- The end milling is the operation of producing a flat surface which may be vertical, horizontal or at an angle in reference to the table surface.
- The cutter used is an end mill. The end milling cutters are also used for the production of slots, grooves or keyways.
- A vertical milling machine is more suitable for end milling operation.

10-Saw Milling



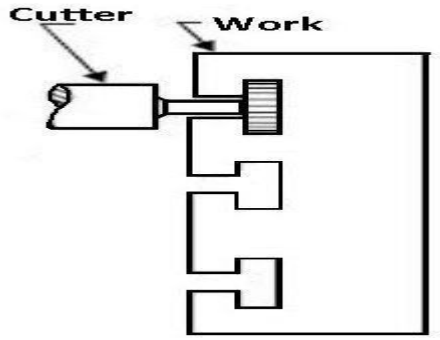
- Saw-milling is the operation of producing narrow slots or grooves on a workpiece by using a saw-milling cutter.
- The saw-milling also performed for complete parting-off operation.
- The cutter and the workpiece are set in a manner so that the cutter is directly placed over one of the T-slots of the table.

11-Milling Keyways, Grooves and Slots



Keyway Milling Operation

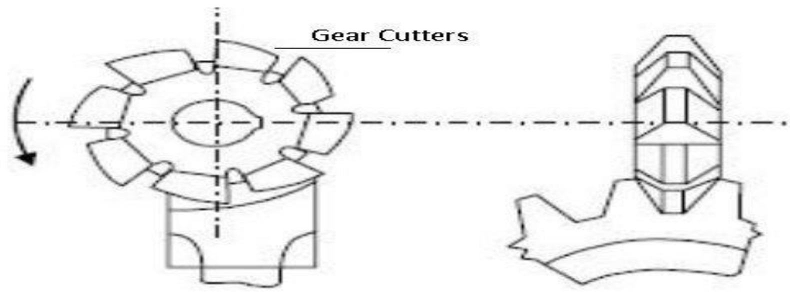
- The operation of producing of keyways, grooves and slots of varying shapes and sizes can be performed in a milling machine.
- It is done by using a plain milling cutter, a metal slitting saw, an end mill or by a side milling cutter.
- The open slots can be cut by a plain milling cutter, a metal slitting saw, or by a side milling cutter. The closed slots are produced by using endmills.



T-Slot Milling Operation

- A dovetail slot or T-slot is manufactured by using special types of cutters designed to give the required shape on the workpiece.
- The second slot is cut at right angles to the first slot by feeding the work past the cutter.
- A woodruff key is designed by using a woodruff key slot cutter.
- Standard keyways are cut on the shaft by using side milling cutters or end mills.
- The cutter is set exactly at the centre line of the workpiece and then the cut is taken.

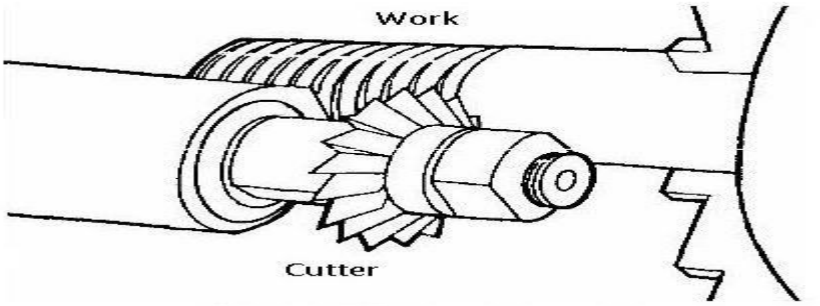
12-Gear Cutting



Gear Cutting Milling Operation

- The gear cutting operation is performed in a milling machine by using a form-relieved cutter. The cutter may be a cylindrical type or end mill type.
- The cutter profile fits exactly with the tooth space of the gear.
- Equally spaced gear teeth are cut on a gear blank by holding the work on a universal diving head and then indexing it.

13-Helical Milling



Helical Milling Operation

- The helical milling is the operation of producing helical flutes or grooves around the periphery of a cylindrical or conical workpiece.
- The operation is performed by rotating the table to the required helix angle. And then by rotating and feeding the workpiece against rotary cutting edges of a milling cutter.
- Production of the helical milling cutter, helical gears, cutting helical grooves or flutes on a drill blank or a reamer.

14-Cam Milling

The cam milling is the operation of producing cams in a milling machine by the use of universal dividing head and a vertical milling attachment. The cam blank is mounted at the end of the dividing head spindle and an end mill is held in the vertical milling attachment.

The axis of the cam blank and the end mill spindle should always remain parallel to each other when setting for cam milling. The dividing head is geared to the table feed screw so that the cam is rotated about its axis while it is fed against the end mill. The axis of the cam can be set from 0 to 90° in reference to the surface of the table for obtaining a different rise of the cam.

15-Thread Milling

The thread milling machine operations are used to produce threads by using a single or multiple thread milling cutter. Thread milling operation is performed in special thread milling machines to produce accurate threads in small or large quantities.

The operation requires three driving motions in the machine. One for the cutter, one for the work and the third for the longitudinal movement of the cutter.

When the operation is performed by a single thread milling cutter, the cutter head is swivelled to the exact helix angle of the thread. The cutter is rotated on the spindle and the workpiece is revolved slowly about its axis. The thread is completed in one cut by setting the cutter to the full depth of the thread and then feeding it along the entire length of the workpiece.

When the thread is cut by multiple thread milling cutter, the cutter axis and the work spindle are set parallel to each other after adjusting the depth of cut equal to the full depth of the thread. The thread is completed by simply feeding the revolving cutter longitudinal through a distance equal to the pitch length of the thread while the work is rotated through one complete revolution.

Lab Session No 4

To perform the drilling operations on conventional drilling machine.

Drilling Machine:

Drilling machine can be defined as a machine which makes a circular hole in the job by removing volume of the metal from it with the help of a cutting tool called drill bit. When drilling is performed by the drilling machine the drill bit i.e. the cutting tool is rotated along its own axis into the job.

A drilling machine consists of the following parts:

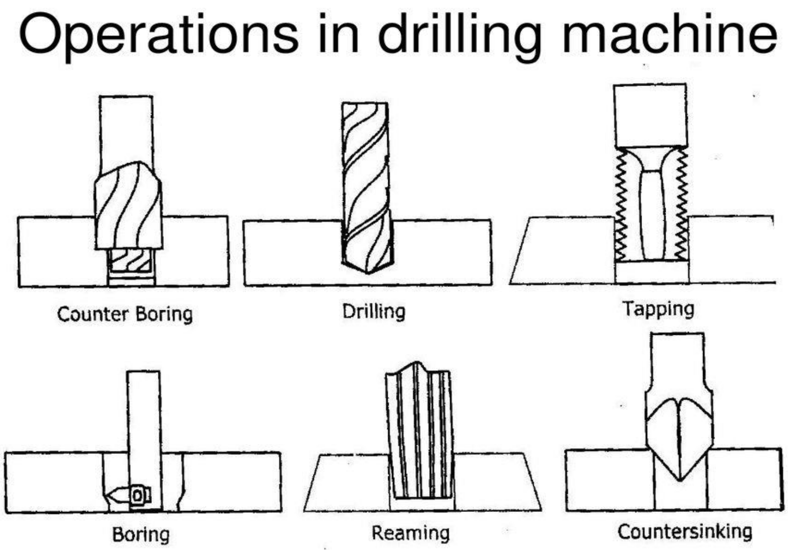
- Base.
- Column or Pillar.
- Arm.
- Worktable.
- Drill head.
- Feed Mechanism.
- Spindle.
- Drill jigs.

Process:

Drilled holes are characterized by their sharp edge on the entrance side and the presence of burrs on the exit side (unless they have been removed). Also, the inside of the hole usually has helical feed marks.

Drilling may affect the mechanical properties of the workpiece by creating low residual stresses around the hole opening and a very thin layer of highly stressed and disturbed material on the newly formed surface. This causes the workpiece to become more susceptible to corrosion and crack propagation at the stressed surface. A finish operation may be done to avoid these detrimental conditions.

For fluted drill bits, any chips are removed via the flutes. Chips may form long spirals or small flakes, depending on the material, and process parameters.The type of chips formed can be an indicator of the machinability of the material, with long chips suggesting good material machinability.



When possible drilled holes should be located perpendicular to the workpiece surface. This minimizes the drill bit's tendency to "walk", that is, to be deflected from the intended center-line of the bore, causing the hole to be misplaced. The higher the length-to-diameter ratio of the drill bit, the greater the tendency to walk. The tendency to walk is also preempted in various other ways, which include:

Surface finish produced by drilling may range from 32 to 500 microinches. Finish cuts will generate surfaces near 32 microinches, and roughing will be near 500 microinches.

Cutting fluid is commonly used to cool the drill bit, increase tool life, increase speeds and feeds, increase the surface finish, and aid in ejecting chips. Application of these fluids is usually done by flooding the workpiece with coolant and lubricant or by applying a spray mist.

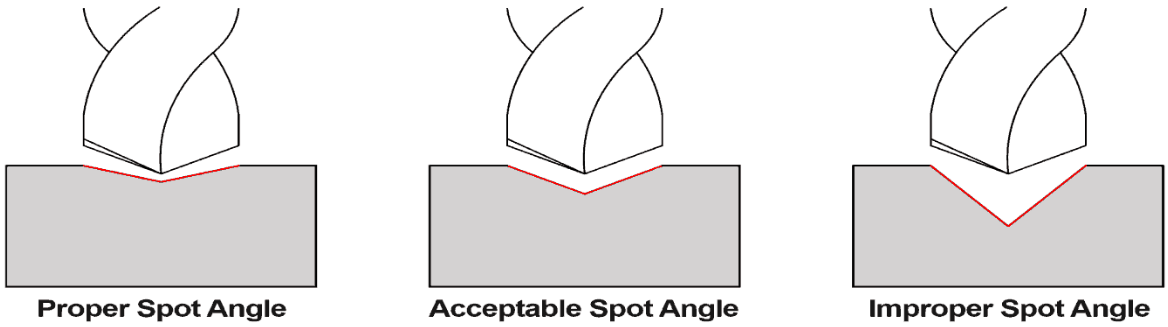
Drilling types:

- Establishing a centering mark or feature before drilling, such as by:
  - Casting, molding, or forging a mark into the workpiece
  - Center punching
  - Spot drilling (i.e., center drilling)
  - Spot facing, which is machining a certain area on a casting or forging to establish an accurately located face on an otherwise rough surface.
- Constraining the position of the drill bit using a drill jig with drill bushings

In deciding which drill(s) to use it is important to consider the task at hand and evaluate which drill would best accomplish the task. There are a variety of drill styles that each serve a different purpose. The subland drill is capable of drilling more than one diameter. The spade drill is used to drill larger hole sizes. The indexable drill is useful in managing chips.

Spot Drilling:

The purpose of spot drilling is to drill a hole that will act as a guide for drilling the final hole. The hole is only drilled part way into the workpiece because it is only used to guide the beginning of the next drilling process.



Center Drilling:

Centre drill is A two-fluted tool consisting of a twist drill with a 60° countersink; used to drill countersink center holes in a workpiece to be mounted between centers for turning or grinding.

Deep Hole Drilling:

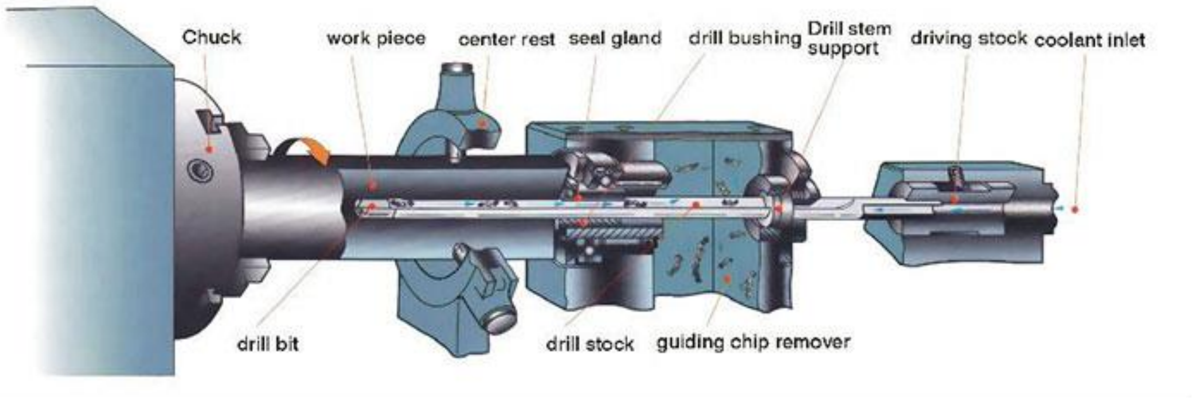
Deep hole drilling is defined as drilling a hole of depth greater than ten times the diameter of the hole. These types of holes require special equipment to maintain the straightness and tolerances. Other considerations are roundness and surface finish.

Deep hole drilling is generally achievable with a few tooling methods, usually gun drilling or BTA drilling. These are differentiated due to the coolant entry method (internal or external) and chip removal method (internal or external). Using methods such as a rotating tool and counter-rotating workpiece are common techniques to achieve required straightness tolerances. Secondary tooling methods include trepanning, skiving and burnishing, pull boring, or bottle boring. Finally, a new kind of drilling technology is available to face this issue: vibration drilling. This technology breaks up the chips by a small controlled axial vibration of the drill. The small chips are easily removed by the flutes of the drill.

A high tech monitoring system is used to control force, torque, vibrations, and acoustic emission. Vibration is considered a major defect in deep hole drilling which can often cause the drill to break. A special coolant is usually used to aid in this type of drilling.

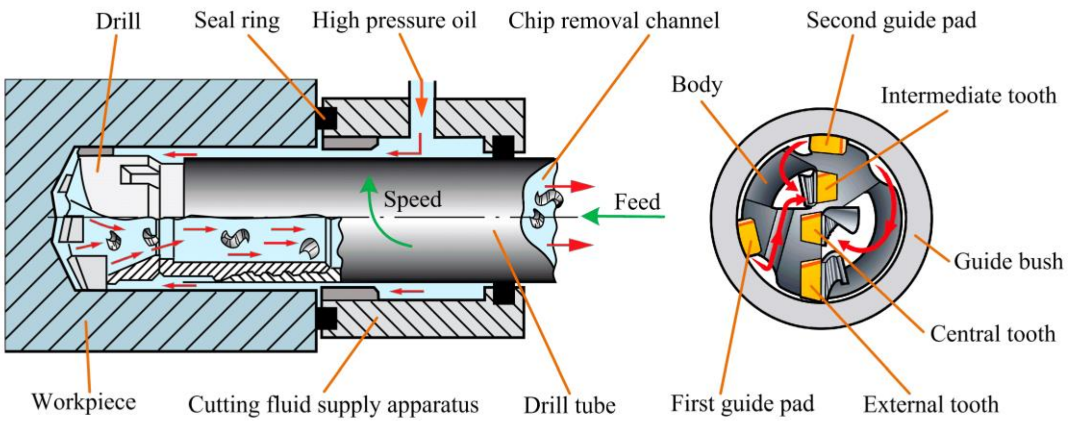
Gun Drilling:

Gun drilling was originally developed to drill out gun barrels and is used commonly for drilling smaller diameter deep holes. The depth-to-diameter ratio can be even greater than 300:1. The key feature of gun drilling is that the bits are self-centering; this is what allows for such deep accurate holes. The bits use a rotary motion similar to a twist drill; however, the bits are designed with bearing pads that slide along the surface of the hole keeping the drill bit on center. Gun drilling is usually done at high speeds and low feed rates.



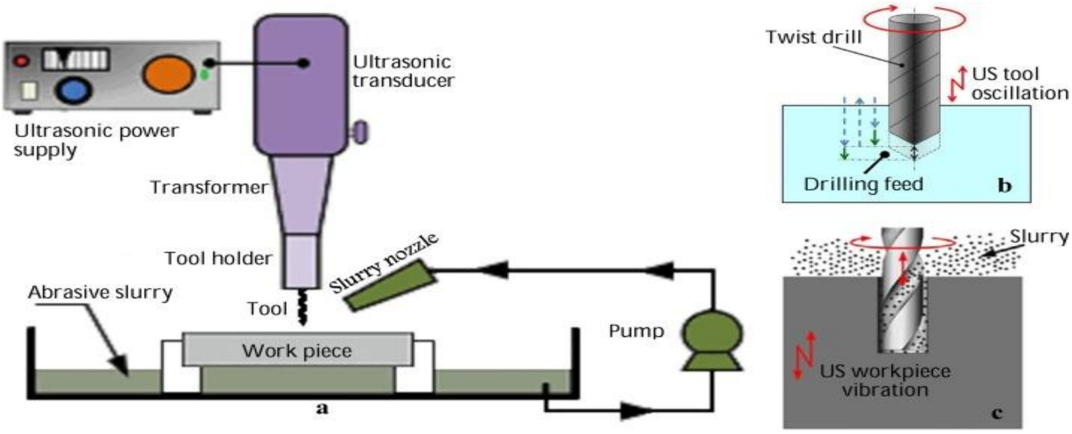
Trepanning:

Trepanning is commonly used for creating larger diameter holes (up to 915 mm (36.0 in)) where a standard drill bit is not feasible or economical. Trepanning removes the desired diameter by cutting out a solid disk similar to the workings of a drafting compass. Trepanning is performed on flat products such as sheet metal, granite (curling stone), plates, or structural members like I-beams. Trepanning can also be useful to make grooves for inserting seals, such as O-rings.



Microdrilling:

Microdrilling refers to the drilling of holes less than 0.5 mm (0.020 in). Drilling of holes at this small diameter presents greater problems since coolant fed drills cannot be used and high spindle speeds are required. High spindle speeds that exceed 10,000 RPM also require the use of balanced tool holders.


Ultrasonic power supply
Ultrasonic transducer
Transformer
Tool holder
Slurry nozzle
Abrasive slurry
Tool
Work piece
Pump
a
Twist drill
US tool oscillation
Drilling feed
b
Slurry
US workpiece vibration
c

Lab Session NO 5

Detail introduction of wire EDM and draw different shapes using controller**.**

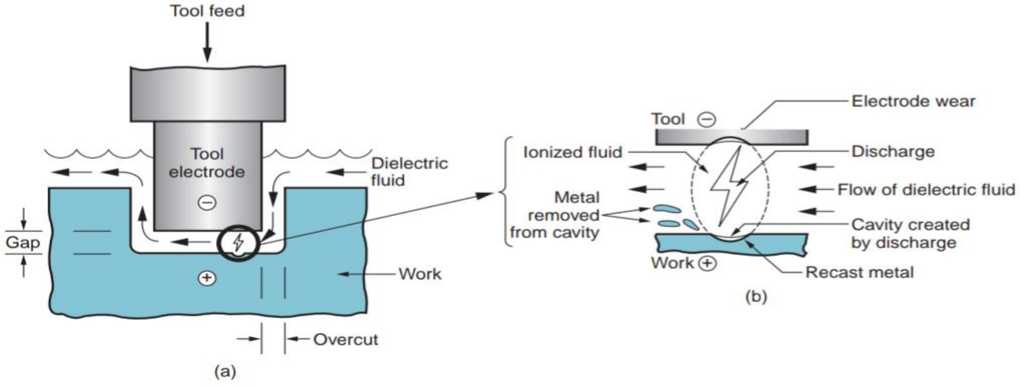
Wire EDM

Wire electrical discharge machining (EDM) is a non-traditional machining process that uses electricity to cut any conductive material precisely and accurately with a thin, electrically charged copper or brass wire as an electrode. During the wire EDM process, the wire carries one side of an electrical charge and the work piece carries the other side of the charge. When the wire gets close to the part, the attraction of electrical charges creates a controlled spark, melting and vaporizing microscopic particles of material. The spark also removes a miniscule chunk of the wire, so after the wire travels through the work piece one time, the machine discards the used wire and automatically advances new wire. The process takes place quickly—hundreds of thousands of sparks per second—but the wire never touches the work piece. The spark erosion occurs along the entire length of the wire adjacent to the work piece, so the result is a part with an excellent surface finish and no burrs regardless of how large or small the cut. Wire EDM machines use a dielectric solution of deionized water to continuously cool and flush the machining area while EDM is taking place. In many cases the entire part is submerged in the dielectric fluid, while high-pressure upper and lower flushing nozzles clear out microscopic debris from the surrounding area of the wire during the cutting process. The fluid also acts as a non-conductive barrier, preventing the formation of electrically conductive channels in the machining area. When the wire gets close to the part, the intensity of the electric field overcomes the barrier and dielectric breakdown occurs, allowing current to flow between the wire and the work piece, resulting in an electrical spark.

Figure 1 WIRE EDM

On most wire EDM machines, the path of the wire is controlled by computer numerically-controlled (CNC) diamond guides, which can move independently of each other on multiple axes for tapered cuts and complex shapes such as small-radius inside corners and narrow slots. Additionally, wire sizes vary from 0.012” diameter down to 0.004” for high-precision work. Wire EDM is capable of holding tolerances as tight as +/-0.0001”.Wire EDM provides a solution to the problems encountered when trying to machine materials that are normally difficult to work with, such as hardened steel, aerospace-grade titanium, high-alloy steel, tungsten carbide, Inconel, and even certain conductive ceramics.

One requirement of the wire EDM process is a start hole for threading the wire if the part’s features do not allow you to cut an edge. Wire EDM can only machine through features; however, we can quickly drill a hole in any conductive material using another type of EDM, small hole drilling or “hole pop” EDM.



Hole Pop EDM

Small hole drilling or “hole pop” EDM utilizes a hollow round electrode to EDM a hole through the work piece. Like wire EDM, hole pop EDM uses spark erosion to remove material; however, in hole pop EDM the size of the hole is controlled by the diameter of the electrode. Dielectric fluid is pumped through and around the electrode during the EDM process for cooling and flushing of eroded particles. CNC hole drilling EDM enables unattended drilling of multiple holes quickly and easily without the risks of manual positioning. Hole pop EDM is ideal for parts requiring start holes, vent holes, coolant holes, ejector pin holes or other blind holes, and is useful in removing broken taps and drill bits.

Sinker EDM

Sinker EDM, also known as ram EDM, conventional EDM or plunge EDM, employs machined electrodes of varying shapes, sizes and materials to remove material from the workpiece. Electrodes are usually made from graphite, but copper, tungsten or brass as well as combinations of these materials are also used, and the geometric features of the electrode can be customized to reach the required specifications. In the sinker EDM process, both the workpiece and the electrode are submerged in a dielectric fluid of oil or synthetic oil and the machine guides the electrode toward the workpiece automatically using CNC technology. As in wire EDM, as the electrode approaches the workpiece, the strength of the electrical charges breaks the barrier of dielectric fluid and a spark is created, eroding away a small amount of material by melting and vaporizing microscopic particles. The process repeats hundreds of thousands of times per second, and as material is removed, the machine continues to direct the movement of the electrode until the desired dimensions are achieved. After the initial “roughing” pass, during which the bulk of the material is removed, we may perform additional 3-D orbiting passes in order to achieve the best surface finish and tightest tolerances possible. Sinker EDM allows us to machine complex 3-D shapes, blind cavities, intricate internal features, threads into hardened parts, and much more. In addition, sinker EDM has the ability to use multi-up electrodes for multi-up parts production—in some cases, we can sinker EDM 100 pieces simultaneously.

Lab Session No 7

Tool adjustment of EDM die sinker machine to equalize the spark at every position of tool.

Apparatus:

EDM machine with tool to be adjusted.

Background information:

Electrical discharge machining (EDM) has long been the answer for high accuracy, demanding machining applications where conventional metal removal is difficult or impossible.

Known by many other names, including spark machining, arc machining and (inaccurately) burning, the EDM process is conceptually very simple: an electrical current pass between an electrode and a workpiece which are separated by a dielectric liquid. The dielectric fluid acts as an electrical insulator unless enough voltage is applied to bring it to its ionization point, when it becomes an electrical conductor. The resulting spark discharge erodes the workpiece to form a desired final shape. Like the electric telegraph and the jet engine, die sinker EDM was invented independently and almost simultaneously by more than one person. In 1941, Russian scientists Boris and Natalya Makarenko were tasked with finding a way to increase the lifespan of tungsten breaker points. In the course of their research, they discovered that they could control the erosion of tungsten electrical contacts by immersing them in a dielectric fluid. By 1943, the Makarenko's had developed a spark machining process based on this discovery that eventually became known as a resistor-capacitor (R-C) circuit for EDM.

At nearly the same time, an American engineering team—consisting of Harold Stark, Victor Harding and Jack Beaver—was working on a way to remove broken drills and taps. Harding, an electrical engineer, proposed using sparks to erode them away. The idea showed promise, but it wasn't until water was added as a coolant that this approach became practical. Stark, Harding and Beaver continued to refine their process, which eventually became the basis for the vacuum-tube EDM, which made it possible to increase spark frequency from 60 times per second to well over 1,000.

As it exists today, die sinker EDM is used to create complex cavity shapes in tool and die applications, such as metal stamping dies and plastic injection molds. The die sinker process begins with machining a graphite electrode to form a "positive" of the desired cavity.

This electrode is then carefully plunged into the workpiece, causing sparking over its surface as features close the sparking gap—the distance required for sparking.

EDM DIE SINKER

Tool adjustment in EDM DIE SINKER:

The prime role of EDM tool is to convey the electrical pulse to allow erosion of work piece with little or no tool wear rate. A lot of effort has gone into the EDM tooling problem regarding inexpensive tool materials, ease of manufacture, rapid work piece erosion, coupled with Electrode and the workpiece tool erosion. To improve machining efficiency, roughing, finishing and semi-finishing electrodes are used in EDM process. EDM is mostly employed in Recent Researches in Multimedia Systems, Signal Processing, Robotics, Control and Manufacturing Technology ISBN: 978-960-474-283-7 25 obtaining mold cavities, cylindrical hole machining and 3-dimensional cavity machining. In cylindrical hole machining, through holes and cavities are produced by electrodes of constant cross section. However, in 3- dimensional cavity machining any cavity is machined with one or more electrodes with varying cross section. The tool design procedure is approximately same for both the cases. We have discussed basic principles of electrode design. It has classified EDM tools depending upon the value of area of projection of electrode on the work piece plane. Two major factors governing the tool design are material selection and electrode wear.

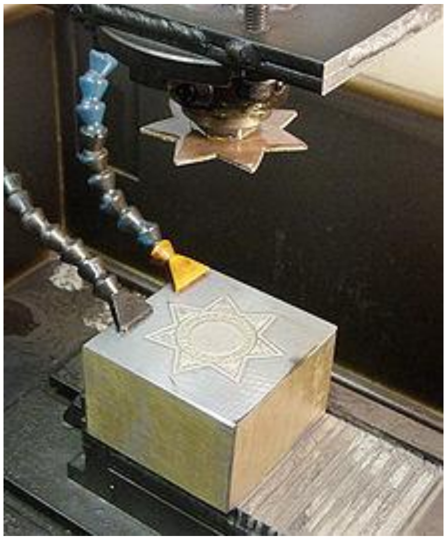
The tool is adjusted in such a way that there will be equalized sparking between tool and workpiece so that we have to follow following parameters:

Spark On-Time (Pulse Time or Ton):
It is the duration of time expressed in micro seconds in whom the peak current is ready to flow in every cycle. This is the time in which energy removes the metallic particles from the work piece. This energy is really controlled by the peak current and the length of the on-time.

Diameter of Electrode (D):
It is the diameter of electrode or tool material. Diameter of tool is one factor considered on machining. This experiment 10 mm
tool diameter is utilized.

Application of EDM DIE SINKER:

Coinage die making:



Coinage Die Making

Master at top, badge die workpiece at bottom, oil jets at left (oil has been drained). Initial flat stamping will be "dapped", see sinking (metalworking), to give a curved surface.

For the creation of dies for producing jewelry and badges, or blanking and piercing (through use of a pancake die) by the coinage (stamping) process, the positive master may be made from sterling silver, since (with appropriate machine settings) the master is significantly eroded and is used only once. The resultant negative die is then hardened and used in a drop hammer to produce stamped flats from cutout sheet blanks of bronze, silver, or low proof gold alloy. For badges these flats may be further shaped to a curved surface by another die. This type of EDM is usually performed submerged in an oil-based dielectric. The finished object may be further refined by hard (glass) or soft (paint) enameling, or electroplated with pure gold or nickel. Softer materials such as silver may be hand engraved as a refinement.

Lab Session No 8

Set up the desired value of current and create a 3D Impression on tool.

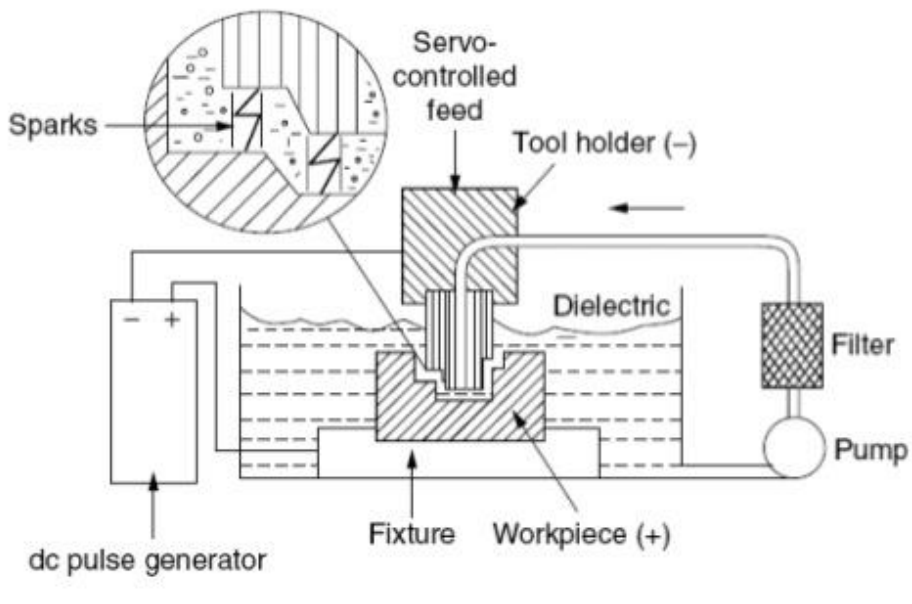
Apparatus:

EDM DIE SINKER, control unit.

Background information:

Two Russian scientists, B. R. Makarenko and N. I. Makarenko, were tasked in 1943 to investigate ways of preventing the erosion of tungsten electrical contacts due to sparking. They failed in this task but found that the erosion was more precisely controlled if the electrodes were immersed in a dielectric fluid. This led them to invent an EDM machine used for working difficult-to-machine materials such as tungsten. The Makarenko' machine is known as an R-C-type machine, after the resistor–capacitor circuit (RC circuit) used to charge the electrodes.[3]

Simultaneously but independently, an American team, Harold Stark, Victor Harding, and Jack Beaver, developed an EDM machine for removing broken drills and taps from aluminum castings. Initially constructing their machines from feeble electric-etching tools, they were not very successful. But more powerful sparking units, combined with automatic spark repetition and fluid replacement with an electromagnetic interrupter arrangement produced practical machines. Stark, Harding, and Beaver's machines were able to produce 60 sparks per second. Later machines based on their design used vacuum tube circuits that were able to produce thousands of sparks per second, significantly increasing the speed of cutting



Electro discharge machining (EDM) Diagram

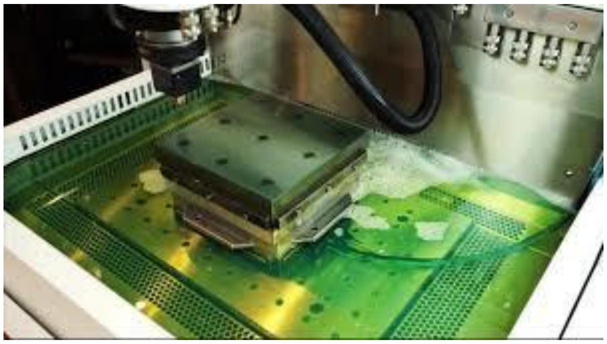
EDM DIE SINKER

Setting value of current:

If you now observe the charge's fade phase you will see that the removal from the workpiece is caused by the collapse of the plasma canal. The sudden drop in pressure triggered by switching off the power causes the evaporation and ejection of superheated material. The plasma canal has a very high temperature and pressure. The gradient of its collapse influences material removal. The more abruptly the energy disappears, the better the crater material will be ejected. In order to enhance this effect, a special trick is employed: before the pulse is interrupted, the current is increased briefly. The idea of increasing the pulse current is not new[6], the innovation is the definition of the point in time when this increase is to take place. The increase in the pulse current has no consequences for the roughness, wear or gap width, but does increase the removal. In addition, as the removal per pulse is greater, you need fewer pulses for the machining, and therefore the wear sinks.

Creating 3D object:

New wire EDM systems allow the importing of 3D CAD (Parasolid) files, and these 3D model contour files are then extracted via the on-board 3D CAM software. This feature provides the ability to bring a solid model file into the control and set the wire cut height to generate the NC cutting profile, including machining conditions, and then uses the 2D CAM software to create the NC program for the machine to follow.



EDM 3D System

Lab Session No 9

To develop CNC part program for a given drawing using G codes and M codes.

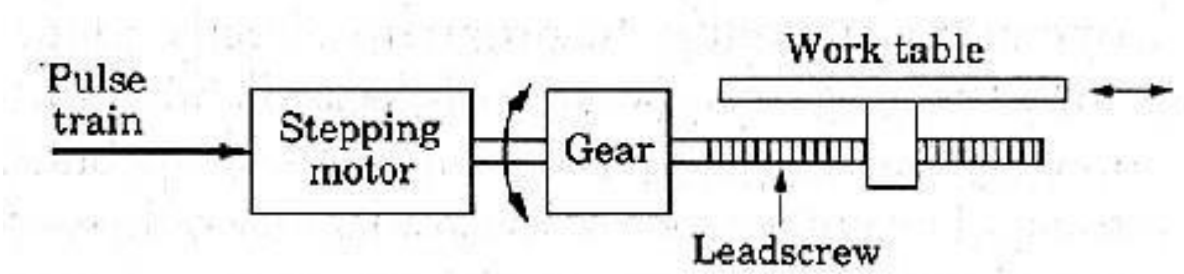
Introduction:

Computer Numerical Control (CNC) is a specialized and versatile form of Soft Automation and its applications cover many kinds, although it was initially developed to control the motion and operation of machine tools.
The definition of CNC given by Electronic Industry Association (EIA) is as follows:
"A system in which actions are controlled by the direct insertion of numerical data at some point. The system must automatically interpret at least some portion of this data".
In a simple word, a CNC system receives numerical data, interpret the data and then control the action accordingly.

Computer Numerical Control Fundamentals

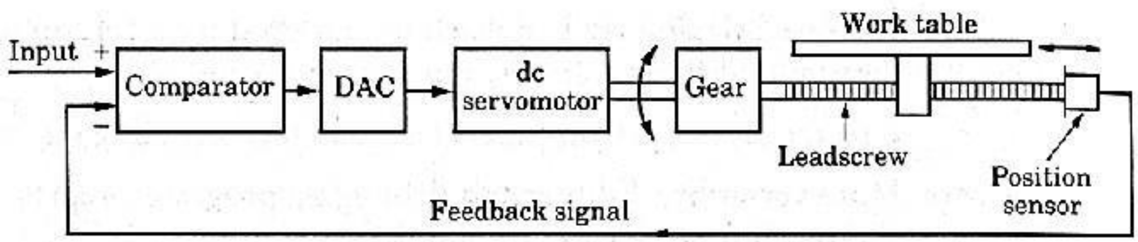
Control Systems

Open Loop Systems:
In an open loop control system, the electrical motor drives the slides as per the input signal or command. There is no monitoring of the actual displacement of the machine slide.



Close Loop Systems

In a closed loop system the displacement of slide is achieved to very high degree of accuracy by using monitoring devices. This feedback is compared with the input info & the slide position is regulated to agree with the desired positions.



From comparator, we get digital data and convert it to analog signal using DAC (digital to analog converter).

Elements of a CNC System:

A CNC system consists of the following 6 major elements:

- Input Device
- Machine control unit
- Machine tool
- driving system
- feedback devices
- display unit

Input Devices:

- USB Flash Drive
- Conversational Programming

Machine Control Unit (MCU):

The machine control unit is the heart of the CNC system. There are two sub-units in the machine control unit: the Data Processing Unit (DPU) and the Control Loop Unit (CLU).

Data Processing Unit:

On receiving a part program, the DPU firstly interprets and encodes the part programme into internal machine codes. The interpolator of the DPU then calculate the intermediate positions of the motion in terms of BLU (basic length unit) which is the smallest unit length that can be handled by the controller. The calculated data are passed to CLU for further action.

Control Loop Unit:

The data from the DPU (data processing unit) are converted into electrical signals to control the driving system to perform the required motions. Other functions such as machine spindle ON/OFF, coolant ON/OFF, tool clamp ON/OFF are also controlled by this unit according to the internal machine codes. Machine Tool:

This can be any type of machine tool or equipment. Required to perform desired work.

Driving System

The driving system has to response accurately according to the programmed instructions. This system usually uses electric motors although hydraulic motors are sometimes used for large machine tools. The motor is coupled either directly or through a gear box to the machine lead screw to moves the machine slide or the spindle. Three types of electrical motors are commonly used.

- ✓ DC Servo Motor
- ✓ AC Servo Motor
- ✓ Stepping Motor

Feedback Device

In order to have a CNC machine operating accurately, the positional values and speed of the axes need to be constantly updated. Two types of feedback devices are normally used positional feedback device and velocity feedback device.

- ✓ Positional Feed Back Devices
- ✓ Velocity Feedback Device

Display Unit:

The Display Unit serves as an interactive device between the machine and the operator. When the machine is running, the Display Unit displays the present status such as the position of the machine slide, the spindle RPM, the feed rate, the part programs, etc.

CNC PART PROGRAMMING

Axis of motion:

In general, all motions have 6 degrees of freedom. In other words, motion can be resolved into 6 axes, namely, 3 linear axes (X, Y and Z axis) and 3 rotational axes (A, B, and C axis).

Dimension Systems:

Incremental System:

This type of control always uses as a reference to the preceding point in a sequence of points. The disadvantage of this system is that if an error occurs, it will be accumulated.

Absolute System:

In an absolute system all references are made to the origin of the coordinate system. All commands of motion are defined by the absolute coordinate referred to the origin.

NC Programming:

NC programming is where all the machining data are compiled and where the data are translated into a language which can be understood by the control system of the machine tool.

Program Structure:

Codes used for CNC:

- ✓ G Codes
- ✓ M Codes

Preparatory Function (G Address):

A preparatory function determines how the tool is to move to the programmed target. The most common G addresses are listed below:

Code	Function
G00	Point to point position at rapid feed
G01	Linear interpolation
G02	Circular interpolation, clockwise
G03	Circular interpolation, anti clockwise
G40	Cutter compensation cancel
G41	Cutter compensation, Left
G42	Cutter compensation, Right
G45 - G48	Other cutter compensation, if used
G70 - G79	Milling and turning cycle
G80 - G89	Drilling and tapping cycle
G90	Absolute dimensioning
G91	Incremental dimensioning

Miscellaneous Function (M Address):

The miscellaneous function is programmed to control the machine operation other than for coordinate movement. The most common M functions are as follows:

Code	Function
M00	Programme stop
M03	Spindle rotation clockwise
M04	Spindle rotation counterclockwise
M05	Spindle STOP
M06	Change of Tool
M08	Coolant ON
M09	Coolant OFF
M10	Clamp
M11	Unclamp
M30	Programme end and ready for another start

Lab Session No 10

To demonstrate the Milling operations perform on CNC milling machine

The Milling Operation:

CNC stands for computer numerically controlled. As a milling technique, this means that a design can be specified on a computer using CAD tools, and that a computer can handle the milling process. And the program is able to specify the movements that the mill and table must make.

CNC milling is a specific form of computer numerical controlled (CNC) machining consisting of both the drilling and cutting. Mill uses a rotating cylindrical cutting tool. However, the cutter in a milling machine is able to move along multiple axes, and can create a variety of shapes, slots and holes. In addition, the work-piece is often moved across the milling tool in different directions, unlike the single axis motion of a drill.

Introduction to The Machine:

In the machine, the labelled axes, X and Y designate horizontal movement of the work-piece (forward-and-back and side-to-side on a flat plane). And Z represents vertical, or up-and-down movement. CNC milling machine also integrate a device for pumping cutting fluid to the cutting tool during machining.

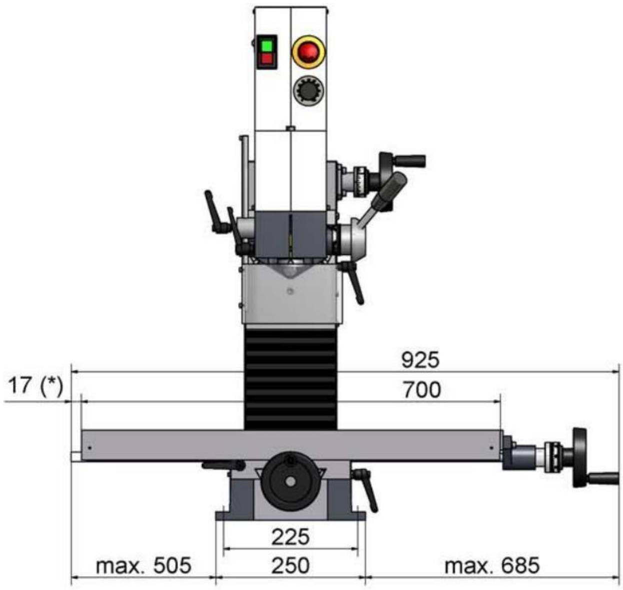
The software used for this machine is NCCAD for which the latest version is 9.5. But currently NCCAD 7.5 is running on the attached computer. This machine can be used to produce a wide range of components, and tooling costs involved have continued to become more affordable day by day. It provides ideal solutions to everything ranging from prototyping and short-run production of complex parts to the fabrication of unique precision components.

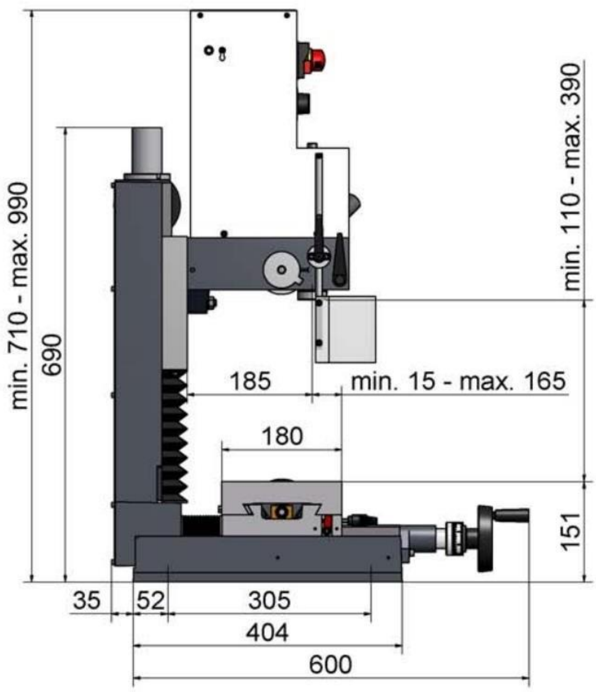
Machinable Materials:

Virtually every type of material that can be drilled or cut can be machined by a CNC mill, although most of the work performed is done in metal. As with drilling and cutting, the proper machine tools must be selected for each material in order to avert potential problems. The hardness of the work-piece material, as well as the rotation of the cutting tool must all be factored before beginning the machining process.

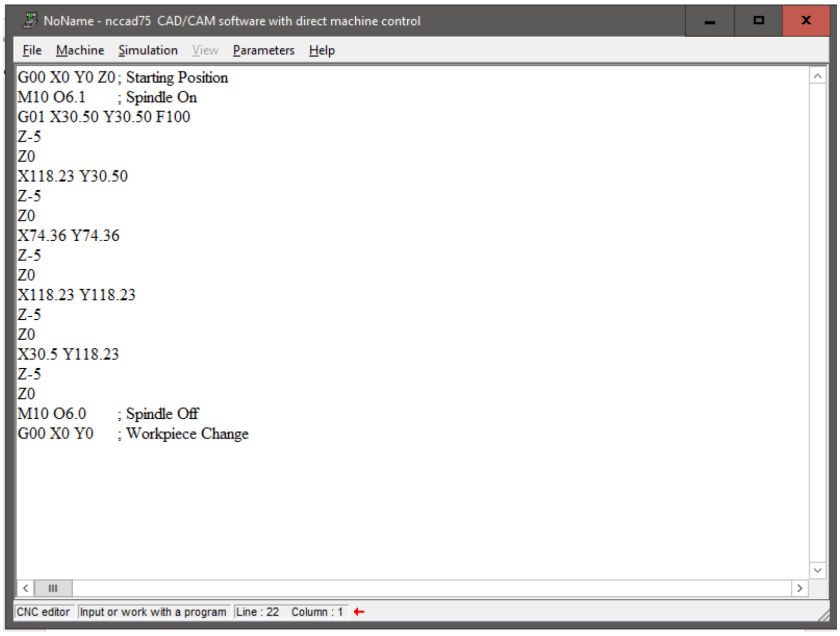
MACHINE AND CONTROLLER:

DIMENSIONS OF MACHINE:

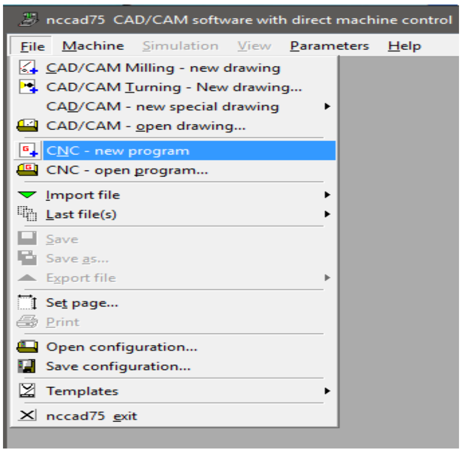




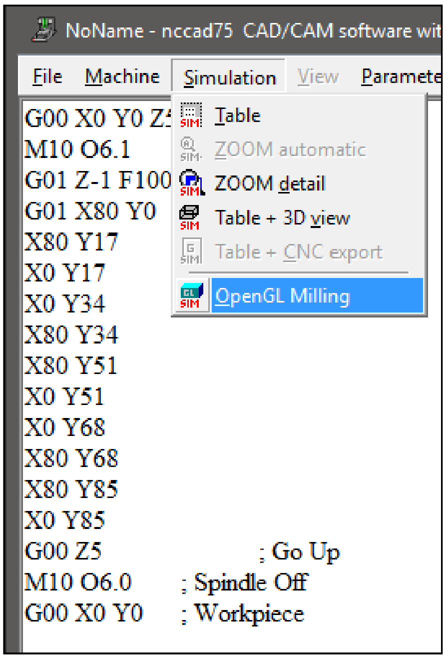
For version 7.5, Go to File and click the option “CNC New Program”. A new window will appear. Here we can write or paste our program code for milling operation. After

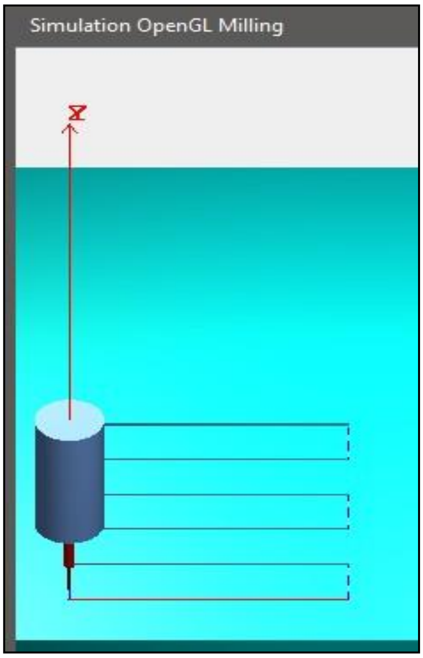


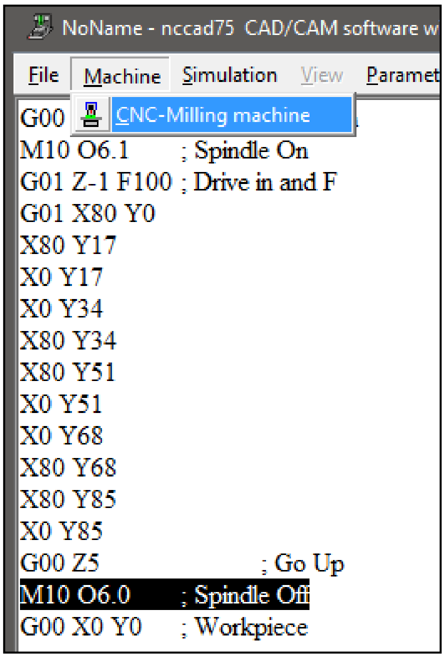
generating the code, Multiply All the X, Y and Z coordinates to 1.9831 (correcting factor) to achieve the Precision as shown in the figure below.



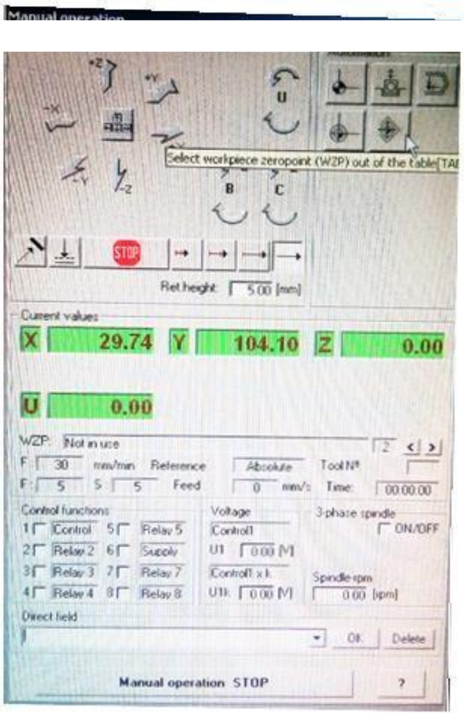
Go to Simulation and observe the simulation of this code to see any errors, by clicking OpenGL Milling. A new window will open showing the simulation. Now Go to Machine Tab and Click the name of Milling Machine WABECO 7.5.

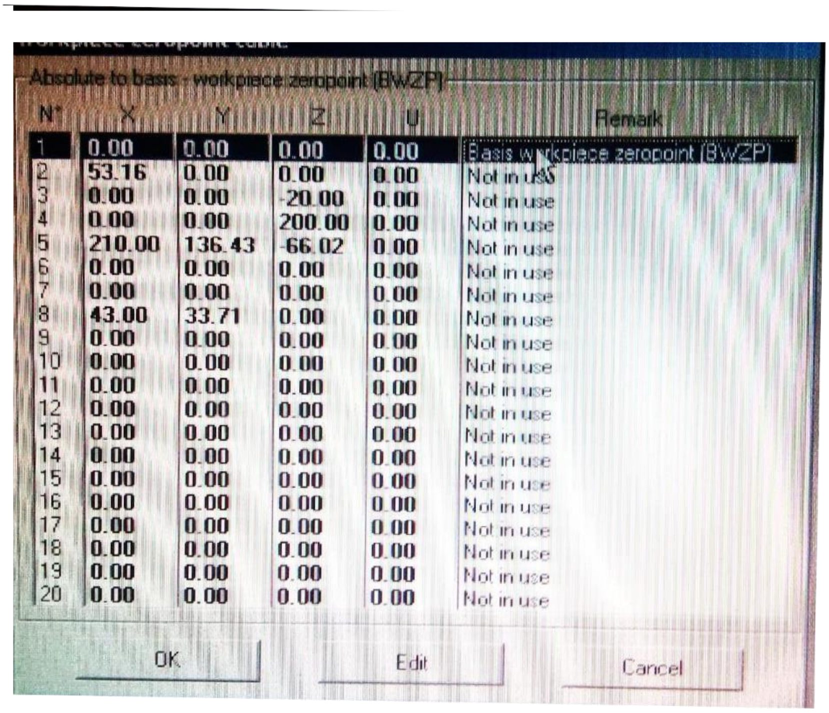






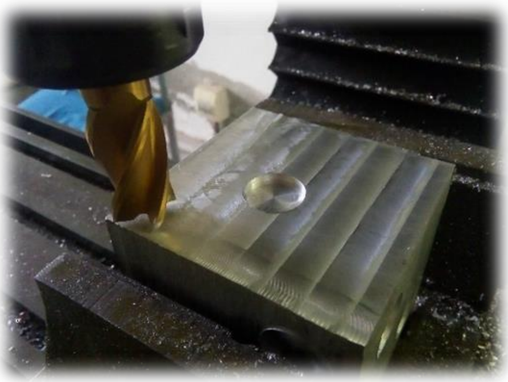
A Window will be displayed with the title manual operation, Click the "Select workpiece zero point (WZP) out of the table". Click the basic work zero point of the table (BWZP) and then click OK button.





N°	X	Y	Z	U	Remark
1	0.00	0.00	0.00	0.00	Basis workpiece zeropoint (BWZP)
2	53.16	0.00	0.00	0.00	Not in use
3	0.00	0.00	-20.00	0.00	Not in use
4	0.00	0.00	200.00	0.00	Not in use
5	210.00	136.43	-66.02	0.00	Not in use
6	0.00	0.00	0.00	0.00	Not in use
7	0.00	0.00	0.00	0.00	Not in use
8	43.00	33.71	0.00	0.00	Not in use
9	0.00	0.00	0.00	0.00	Not in use
10	0.00	0.00	0.00	0.00	Not in use
11	0.00	0.00	0.00	0.00	Not in use
12	0.00	0.00	0.00	0.00	Not in use
13	0.00	0.00	0.00	0.00	Not in use
14	0.00	0.00	0.00	0.00	Not in use
15	0.00	0.00	0.00	0.00	Not in use
16	0.00	0.00	0.00	0.00	Not in use
17	0.00	0.00	0.00	0.00	Not in use
18	0.00	0.00	0.00	0.00	Not in use
19	0.00	0.00	0.00	0.00	Not in use
20	0.00	0.00	0.00	0.00	Not in use

<u>Now Clamp the Tool on the Spindle and Locate it manually with respect to the origin of the workpiece.</u>



While Locating its position please sure that the value of x, y and z should be zero. Position the tool 5 to 10 mm above the work surface because in this machine tool only can travel on negative direction of z axis so we can't put the value of z axis greater then 0.

SAMPLE PROGRAMS OF MILLING:

PLANE MILLING OPERATION:

The following code will work for plane milling on 75*75 dimensions.

```
G00 X0 Y0 Z0
M10 O6.1
G01 Z-1 F25
G01 X160 Y0
X160 Y20
X0 Y20
X0 Y40
X160 Y40
X160 Y60
X0 Y60
X0 Y80
X160 Y80
X160 Y100
X0 Y100
X0 Y120
X160 Y120
X160 Y140
X0 Y140
X0 Y160
X160 Y160
G00 Z0
M10 O6.0
G00 X0 Y0
```

CREATING 6 CIRCULAR IMPRESSIONS FACE MILLING:

The following code will work for plane milling on 75*75 dimensions.

```
G00 X0 Y0 Z0
M10 O6.1
G01 X29.74 Y44.61 F25
Z-5
Z0
X74.36 Y44.61
Z-5
Z0
X118.98 Y44.61
Z-5
Z0
X118.8 Y104.10
Z-5
Z0
X74.36 Y104.10
Z-5
Z0
X29.74 Y104.10
Z-5
```

```
Z0
G00 Z0
M10 O6.0
```

OPERATING INSTRUCTIONS:

- The zero point for the workpiece and the reference axis in the program should be same.
- The tool must be selected according to the dimensions of our workpiece and the specified requirement of operation.
- The dimension of our workpiece in millimeter must be multiplied with 1.93 before using these dimensions in program to convert the dimension into the equivalent dimension in the units of machine.
- The maximum range of the CNC in x-axis and y-axis is 210mm, while it is 70mm in z-axis.
- Feed rate should not be a high to avoid tool wear

Lab Session No 11

To perform Job setting, tool compensation setting and tool changing on CNC Machining Center

CNC Machining Center

A sophisticated CNC machine that can perform milling, drilling, tapping, and boring operations at the same location with a variety of tools is called CNC machining center.

CNC Machining Center:

Manual mills require that the operator/machinist set all the required parameters, change tools, and manually direct all table movement. However, with CNC capability, work is performed much faster, with exceptional repeatability. In addition, CNC computer programs can be verified and completed graphically before actual metal cutting begins. A machining center is a machine for both milling and holes making on a variety of non-round or prismatic shapes.

Types of CNC Machining Center:

Machining centers are either vertical or horizontal. The vertical type is often preferred when work is done on a single face. With the use of rotary tables, more than one side of a work piece, or several work pieces, can be machined without operator intervention. Vertical machining centers using a rotary table have four axes of motion. Three are lineal motions of the table while the fourth is the table's rotary axis. Horizontal centers with their horizontal spindles are better suited to larger, boxy work pieces.

With a horizontal spindle, a wider variety of work piece shapes are easier to mount and chips fall out of the way better. Like vertical machining centers, horizontal centers have multiple-axis table movements. Typically, the horizontal center's table rotates to present all four sides of a work piece to the tooling.

Parts and their functions:

Work holding and Work changing:

In all kinds of milling, a critical component is the work holding device and the ability to be changed over quickly to present new work or work surfaces to the tooling. Machining centers can utilize long machine beds, pallet changers and multi-sided "tombstone" fixtures to enable new work to be set up and positioned while previously setup work pieces are being milled.

Lubrication system:

The lubrication system is used to lubricate the moving parts of CNC machining center like rail lead screw and other moving parts.

Lubrication pump is used to force the lubrication oil to circulate in the machine.

Coolant system:

Coolant tank store the coolant or cutting oil and pump is used to pump the coolant from tank to machining area.

Turret:

Turret is the part where different tools are mounted according to requirement different turret have different capacity to holdings tools normally 12 to 22 tools are hold in turret.

Spindle:

Spindle is the part where tool is grape for machining purpose

Control panel:

From control panel we can control all movements of the machine and feed program according to our requirement different type of keys are placed on control panel like

1. Mode selecting knob
2. Setting keys
3. Soft keys
4. Keyboard

Mode Selecting Knob:

The mode selecting knob includes the following feature:

Fig.6.1: Knob

- Edit Program: If we want to edit some program which is already existing we have to turn the knob to this position edit program and then we can edit the program.

  Memory: Memory contains all the save programs and several programs can be save in memory.
- DNC: If we want to contact more than one machine with one computer than this mode is activated.
- Handle: Handle will not work until we set the knob to this position.
- Jog: To operate the machine manually or to set reference X, Y, Z positions manually.

  Step: To read single block after reading each block machine will stop.
- ZRN: To return the machine to zero position the knob should be at that point.

Setting Keys:

- It includes the following keys.
- Alter: To change any command.
- Delete: To delete any command.
- Cancel: To cancel any command.
- Insert: To insert any new command.
- Message: If machine gives any warning message then to control that we have to press “Msg” key.
- End of Block: It is used to end one block while writing program.
- Position: To see the value of X, Y and Z in absolute positions.

Soft Keys:

These are the keys which are provided on the bottom of the screen.

Keyboard:

All instruction is entered from key board are appears on control panel display screen

Power ON and OFF key:

This key used for on and off power of machine

Emergency stop button:

Use for stop the machine in emergency condition like accident ect.

Cycle start:

When the program is ready the press this button for machining cycle start.

Feed rate overdrive:

For control the feed rate according to requirement.

Spindle overdrive:

For control the speed of the spindle according to requirement.

Rapid traverse:

To control the fast and slow motion of spindle in x.y.z direction.

Offset:

This button is used when the tool wears and is used to give compensation of tool wear. If tool wears than negative value is given while setting the reference.

Handel:

Handel is used to control the movements of machine in X,Y,Z direction.

Machine operating procedure for cutting:

On the machine power when machine display is illuminate then release the emergency stop button

- Check the air pressure level it should be 0.4 bars
- Machine compressor breaker in on position
- Select the work coordinate system G54-G59
- Set the reference according to work piece edge reference or center point reference
- Move the X,Y,Z axis according to reference point by handle
- When the tool is reached the final reference point press the button position
- Read all values of X,Y,X in the block of absolute reading values
- Entre all the absolute X,Y,Z values in work coordinate system which is selected before G54-G59
- Select edit mode to write program
- Use different keys from control panel to write a program like edit, alter, insert according to requirement
- When program is completed it save automatically
- Select memory from mode knob
- Press the button of cycle start
- Close the door of machine
- Machining will be start
- After completing the machining machine will be stop
- Dry run mode option is also available for check the program without cutting