Participant Handbook

Machining Technician (Lathe)

Skill India
कौशल भारत-कुशल भारत

सत्यमेव जयते
GOVERNMENT OF INDIA
MINISTRY OF SKILL DEVELOPMENT
& ENTREPRENEURSHIP

N·S·D·C
National
Skill Development
Corporation
Transforming the skill landscape

Table of Contents

CNC Turning

Symbols Used in This Manual

Symbol	Meaning
	Objectives
	Theory
	Procedure
	Key Learnings
	Worksheets
	Notes

Unit Conversion

At the end of this module you will be able to:

- explain measurements terms (length, weight, time, etc.),
- explain the different measurement systems;
- convert values between the measurement systems.

Session Plan	
1	Measurements
2	Length
3	Weight
4	Time
5	Different Measuring Unit
6	Unit Conversion
7	Key Learnings
8	Worksheet

Measurements

Shailesh has just started his apprenticeship as a computer numerical control (CNC) lathe operator. Today is his first day at the site and he is excited to start operating a CNC machine. But, he is left perplexed when his workshop supervisor, Mohan asks him to read up on the different measurement units. With a disappointed look on his face he asks Mohan:

Sailesh: Why do I need to know about measurements units? Isn't my job, simply to operate the machine?

Mohan: Well yes, but to complete a task you need to take critical measurements based on dimensions set forth in the Engineering Drawings. In addition to this, you need to implement any needed adjustments. Thus, it is extremely important for you to understand how to measure things.

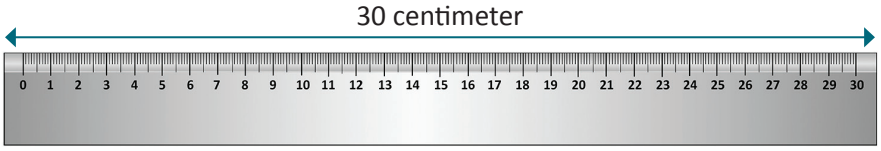
Sailesh: Oh! I see. So, brushing up the measurement units will help me make accurate measurements.

Mohan: That's right! This way you can maintain quality by controlling the work being manufactured to exact tolerances. Let me start by explaining what a measuring unit is.

A measurement unit shows the size or amount of something. In order to complete your job you need to know all about the measurements such as width, length, height, weight and time.

Measurements

In order to measure something you need to assign a number to that entity. For Instance, when I say the scale is 30 centimeter long, I am assigning a number to length of the scale. 30 in this example is the numerical value and centimeter is the unit for measurement.



No value is complete if we don't add the unit of measurement along with it. Let us learn more about the units.

Length

Mohan: The first kind of measurement we are going to explore is length. This is important for lots of different reasons. Measurement of length helps you measure the distance between any two points, or how long and tall things are.

Can you recall the most common units of length?

Sailesh: Well of course, the most common measurements of length are centimeters, Kilometers, inches, feet, yards and miles.

Mohan: That's absolutely correct Sailesh. Now, tell me what unit of length will you use to measure thickness of this handbook?

Sailesh: The thickness of this handbook is quite less so, it would be appropriate to measure it in millimeter or centimeters.

Mohan: Bingo! How about when we have to measure the length of a long Ribbon?

Sailesh: The length of the Ribbon will be measured in meters.

Mohan: Correct Sailesh! Similarly, to measure long distances like the distance between two towns, we use kilometers. 100 centimeters add up to make one meter and once you have 1000 meters you will have one kilometer.

Sailesh: I see.

Weight

Mohan: Next, Let's talk about Weight. Weight of an object means how heavy the object is.

Sailesh: Oh yes! Bigger the size more the weight, right?

Mohan: Well, not always Sailesh. The weight of an object does not depend on its size. For instance, a large plastic ball may be lighter than a small ball made of iron.

Sailesh: Ahh yes… Of course! But, don't we always measure weight in Kilograms?

Mohan: Well, Kilograms is one of the common measurements of weight, but we use grams to weigh lighter things. For instance, hold one small paperclip in your hand. Does it weigh a lot? No! A gram is very light. That is why you often see things measured in hundreds of grams.

Once you have 1,000 grams, you have 1 kilogram. When you weigh yourself on a weighing machine, you would use kilograms. I weigh about 70 kg. How much do you weigh?

Sailesh: I'm slightly heavier. I weigh around 80 kg.

Mohan: There, now you're getting a hang of it. While grams are used to weigh lighter things, we can use tonnes to measure extremely heavy things like trucks, elephants etc.

Time

Mohan: I should be rushing now. It's almost time for the lunch break.

Sailesh: What about time? Can you tell me how can I measure time? Isn't it an intangible thing?

Mohan: I guess I still have time to tell you about time measurement. Look at this watch on my hand. Clocks and watches use hands to show us the hours and minutes. The little hand shows the hours and is called the hour hand. While the big hand shows the minutes and the biggest hand shows us the seconds. These are called the minute and second hand respectively.

When the second's hand makes a full round that is 60 seconds the minute hand moves one point. So, 60 seconds is equal to one minute. When the minutes hand makes a full round that is 60 minutes, it makes 1 hour. So, 60 minutes is equal to 1 hour. Thus, 15 minutes is one quarter of an hour and 30 minutes is half an hour.

Sailesh: This watch shows number only till 12. Does that mean there are only 12 hours in a day?

Mohan: Not really Sailesh. There are 24 Hours in a Day. There are two major ways to show the time: "24 Hour Clock" or "AM/PM".

Whereas with AM/PM (or "12 Hour Clock") the day is split into the 12 Hours running from Midnight to Noon (the AM hours) and the other 12 Hours running from Noon to Midnight (the PM hours). For instance, it is nearly 1 pm now that is 1 hour past midnight.

With the 24 Hour Clock the time is shown as how many hours and minutes have passed since midnight. So, if I were to convert 1 pm to the 24 Hour Clock, I would say it is 13 hundred hours (13:00).

Sailesh: Wow! Now that I know how to use a watch I can get a new watch of my own.

Different Measuring Unit

Sailesh has just begun reading about the common systems of measurement, but he is totally confused between the three common systems of measurement:

- The centimeter–gram–second system (CGS system)
- The meter-kilogram-second (MKS system)
- The foot–pound–second system (FPS system)

What do all these things mean? They sound scary! He decides to ask his supervisor for help.

Sailesh: Mohan...I'm trying to understand the systems of measurement but I am confused. There are too many systems! What do they mean? Why don't we have one system?

Mohan: Relax Sailesh! These systems may fluster you initially but they are quite simple! These systems of measurement use standard units to take accurate measurements. Let's look at these one by one.

As the name suggests, the centimeter–gram–second system (CGS system), is a system of units built on the three fundamental units - centimeter as the unit of length, gram as a unit of weight, and second as a unit of time.

Similarly, the meter-kilogram-second (MKS system) is a physical system of units that expresses any given measurement using fundamental units of the meter, kilogram, and second.

The third common measurement system is the foot–pound–second system (FPS system) is based on foot for length, pound for weight and second for time. Pound can be used instead of kilogram to

measure weight.

Sailesh: You made that sound quite easy. But, what if we want to talk about really big or really small things?

Mohan: In that case we can use prefixes in front of the standard unit. For example, a long rope measures one thousand meters. It is easier to say it is 1 kilometer long, and even easier to write it down as 1 km.

But you have to be careful this may not be applicable in some cases. In the FPS system the unit of length is foot or mile. A foot is used by construction workers and architects to measure walls, floors, and ceilings. Whereas, a mile is used to measure big distances.

Likewise, to measure longer periods of time, minutes or hours are used.

Sailesh: Oho! Now this makes sense!

Unit Conversion

Mohan: Now that you have learned about different measurements units, you should learn to convert units to one another.

Sailesh: Isn't it enough to know the different measurement units? Why will I need to convert one measurement unit to another?

Mohan: That's where you're wrong Sailesh. Let's say you were going to purchase new curtains for your living room. First you would get out your measuring tape and measure the length and width of the window. This would probably be done in inches. Then you would take these measurements to the store and "OUCH" you see that all of the curtains in the store are being sold by the "meter"!" Being unable to convert the feet and inches measurements into meter means that, you really don't know whether you are getting the best deal for your money.

Sailesh: Ahh!! I never thought of it this way. But how can I convert one measurement unit to another?

Mohan: Just like you now know that 1 meter has 100 centimeters, you will have to memorize other conversion values.

Sailesh: Alright, so once I know the different conversion values I can easily convert one measurement value to another.

Mohan: That's right Sailesh. There are many measurement units for length and weight that are used globally. I suggest you chart these out. In fact, while I was starting out my career as a CNC Machine operator I made a table of the most commonly used units and wrote down their conversion value. I still have that table...here you go!

Mohan gave Sailesh a unit conversion table, which turned out to be really helpful. Why don't you take a look at the chart too!

CGS / MKS	FPS
1 centimeter (cm)	0.4 inch
1 meter (100 cm)	3.3 feet
1 kilometer (1000 m)	0.6 mile
1 gram (gm)	0. 002 ounce
1 kilogram (1000 g)	2.2 pounds

FPS	CGS/MKS
1 inch	2.5 centimeters
1 foot	0.3 meter
1 mile	1.6 kilometers
1 ounce (oz)	28 grams
1 pound (lb)	0.5 kilogram

Sailesh: Hey friends, thanks to the chart I can now easily convert units. Do you want to see how I put these to use? Well, here is an example:

Example 1: There are 720 centimeters of tape on a roll. How many inches of tape is this?

The conversion for inches into centimeters is:

2.5 centimeters= 1 Inch (as given in the chart)

Therefore 720 centimeters will be divided by 2.5

720 /2.5 = 288 Inches

So, the length of the tape is 288 Inches.

Pretty neat huh! Here is another example for you:

Example 2: The road is 5 kilometers long? How many miles will it be?

The conversion for kilometers into miles is:

1 km = 0.6 mile (as given in the chart)

Therefore, the 5 kilometers is multiplied by 0.6

5 × 0.6214 = 3.107

So, 5 km of road will be equal to 3.1 miles

I hope you enjoyed learning all about measurement and unit conversion. Now I must return home. Keep measuring until I see you again!

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. List some of the common units of length and weight.

2. There are two major ways to show the time: "24 Hour Clock" or "AM/PM". What is the difference between the two?

3. Name the three common measurement systems. How are they different from one another?

Worksheet

1. Help Sailesh by picking the correct answer from the options given below:

1. What is the unit of length in the CGS system?
   - a. Feet ☐
   - b. Inch ☐
   - c. Centimeter ☐
   - d. Kilometer ☐
2. What is the unit of weight in the FPS system?
   - a. Kilogram ☐
   - b. Gram ☐
   - c. Mile ☐
   - d. Pound ☐
3. How many hours are there in 120 minutes?
   - a. 3 hours ☐
   - b. 2 hours ☐
   - c. 1 hour ☐
   - d. None of the above ☐

4. A cylinder weighs ½ kilo. How many such cylinders will weigh 1 pound?
   a. Three ☐
   b. Two ☐
   c. One ☐
   d. Half ☐
5. How many meters are there in 50¼ km ?
   a. 5,025 ☐
   b. 5,250 ☐
   c. 50,025 ☐
   d. 50,250 ☐
6. Which one of the following has the greatest weight?
   a. A bookshelf that has weighs 12 kg ☐
   b. 25 bags of salt that each has a weight of 500g ☐
   c. A bicycle that has a weight of 0.015 tonne ☐
   d. Two Iron balls each with a weight of 5.5 kg ☐
7. Which of the following is the least distance?
   a. The height of Mount Everest above sea level, which is 8,848 m ☐
   b. The distance of a race track, which is 42.195 km ☐
   c. 100,000 rulers, each of length 30 cm, joined end to end ☐
   d. The distance of one airport to another which is 52 km ☐
8. One week while he was training for a marathon, Jagdeesh ran a total of 131 kilometers. If he ran every day of the week, how many meters on average did he run each day?
   a. Jagdeesh ran an average of 18714.285 meters each day ☐
   b. Jagdeesh ran an average of 17814.285 meters each day ☐
   c. Jagdeesh ran an average of 14817.285 meters each day ☐
   d. Jagdeesh ran an average of 18417.285 meters each day ☐
9. What is 18:35 on the AM/PM clock?
   a. 6:35 AM ☐
   b. 18:35 PM ☐
   c. 6:35 PM ☐
   d. 18:35 AM ☐
10. What is 11:28 AM on the 24 hour clock?
   a. 11:28 ☐
   b. 23:28 ☐
   c. 20:28 ☐
   d. 1:28 ☐

Notes

Answers:

1.

1.	c	2.	d	3.	b
4.	c	5.	d	6.	c
7.	a	8.	a	9.	c
10.	a				

Basic Geometry – 2D

At the end of this module you will be able to:

- identify the different 2D geometrical shapes.

Session Plan	
1	Line, Line Segment and Ray
2	Angle
3	Triangle
4	Quadrilaterals
5	Circle, Elliptical Shapes
6	Key Learnings
7	Worksheet

Line, Line Segment and Ray

It's Sailesh's second day at the workshop. He walks in to find Mohan studying a component drawing. His interest is further piqued when he sees a number of geometrical shapes printed on the paper. He asks Mohan:

Sailesh: What is this drawing?

Mohan: Oh this… this is a component drawing. You could say it is a blueprint of the desired product. Before we begin, it is very important to define the geometry of the part to be manufactured. These geometrical shapes on the component drawing will provide us with the details of the design – like the exact shape and dimensions.

Sailesh: That's interesting! geometry was always one of my favorite subjects.

Mohan: That's good! Be sure to revisit the basics of geometry before you delve into a component drawing.

Here, you can get started with this simple chart that distinguishes a line, ray and Line segment.

Please

Diagram	Definition
A B	**Line** - Infinetly continuous in both directions
O B	**Ray** - With one end point and other end extends in one direction
O C	**Line segment -** Has two end points

Sailesh always thought a line was a straight stroke, but when he studies the chart in detail he learns that a line in geometry extends in both directions without end. A ray on the other hand, has one end point and extends only in on direction. Whereas, a line segment has two end points.

Next, he goes on to learn about the various angles.

Angle

Sailesh has just begun reading about the different angles in geometry. He starts with the definition of an angle.

The amount of turn between two straight lines that have a common vertex

He understand what lines are, but what is a vertex. Confounded with the jargon, he decides to seek Mohan's help.

Sailesh: Mohan, I have understood all about lines, but what is a vertex and what does it have to with angles. Why can't I simply use a protector to measure angles? It has all the angles marked on it… why do I need to study them?

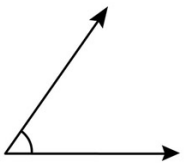
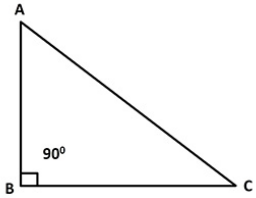
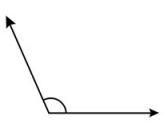
Mohan: Oh Sailesh do not get so confused. Let me explain you…. An angle is the space between two lines, line segments or rays with a common point. The common end point where both lines meet is called the vertex. The two straight sides are called arms.

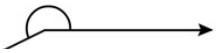
Simply said, the angle is the amount of turn between each arm. We can measure angles in degrees.

There are 360 degrees in one full rotation i.e. one complete circle around.

Sailesh: Oh I see but, what about these different angles? They are so confusing?

Mohan: Relax Sailesh! Some common angles have been given names. As the angles increase, the names also change. Take a look at this chart. It will help you understand the different types of angles. Here the angle is represented by AOB. The point where the two line segments meet at a common point is called angle vertex. In the above diagram, O is the angle vertex.

Type of Angle	Diagram	Description
Acute Angle		An angle that is less than 90°
Right Angle		An angle that is 90° exactly
Obtuse Angle		An angle that is greater than 90° but less than 180°

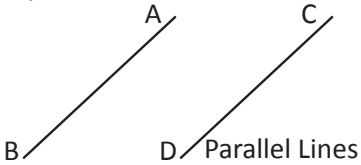
Straight Angle		An angle that is 180° exactly
Reflex Angle		An angle that is greater than 180°

Types of Angles

Sailesh: Now I get it. These are just names of the various kinds of angles. Each angle can be easily distinguished.

Mohan: That's right Sailesh!

Sailesh: Mohan, I still have a slight confusion! You said that an angle is the space between two lines, line segments or rays with a common point but how about when two line segments do not intersect each other…obviously no angle is formed then!

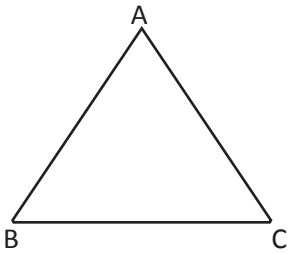


Mohan: You're right Sailesh… when two line segments do not intersect each other they are called parallel lines. For instance, take this diagram, AB is parallel to CD. It is represented as AB//CD.

Triangle

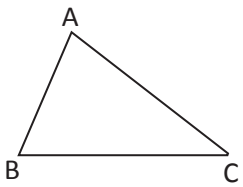
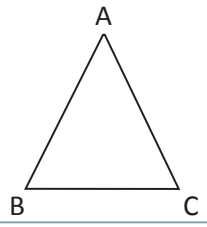
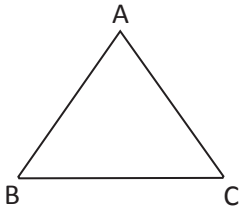
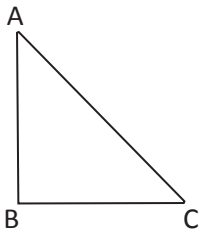
Sailesh: Hello everybody, I must say learning about the various types of angles was quite interesting.

Next, I'm going to learn about the various types of triangles. As most of us will know, a triangle has three sides and three angles. Take a look at this triangle I have drawn here. It has three sides and angles. A, B and C are the three vertices in the triangle. AB, BC and CA are the sides of the triangle.



Representation: △ ABC

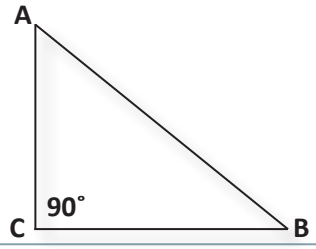
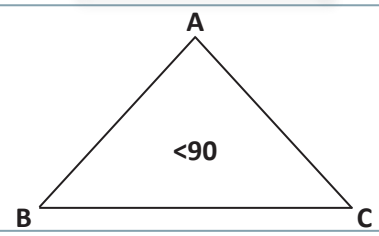
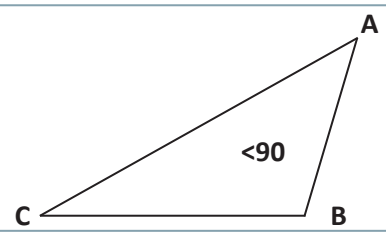
Did you know there are three special names given to triangles that tell how many sides are equal? Here, take a look.

	
Scalene Triangle: All the sides of the triangle are not equal	**Isosceles Triangle**: Two sides of the triangle are equal
	

Equilateral Triangle: All the sides of the triangle are equal	**Right Angle Triangle**: Triangle including a 900. AB is called perpendicular. BC is called base. AC is called Hypotenuse.

Types of Triangles

Triangles can also have names that tell you what type of angle is contained inside. Now that we are acquainted with the classifications of triangles, we can begin our extensive study of the angles of triangles. I have listed these in the chart below.

	**Right Angle Triangle:** Has a right angle(90°) AB is called perpendicular. BC is called base. AC is called Hypotenuse.
	
Acute Triangle All angles are less than 90°	Obtuse Triangle Has an angle more than 90°

Angles of Triangles

Sometimes a triangle may even have two names. For instance, a triangle that has a right angle (90°) and also two equal angles is called a **Right Isosceles Triangle.**

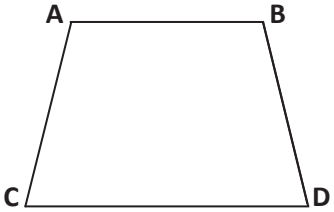
Can you guess what the equal angles are?

Quadrilaterals

Working with triangles was fairly easy... but what about quadrilaterals. Sailesh is totally flustered –what is a quadrilateral? Is it just a skewed square?

That is when Mohan steps in to help him learn about the quadrilateral shapes.

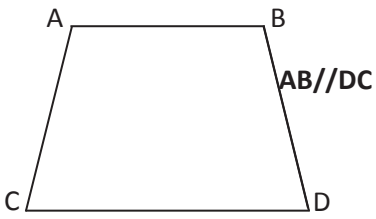
Mohan: Sailesh, don't look so confused. Quadrilateral just means "four sides", wherein quad means four, lateral means side.



Sailesh: Oh I see...any four-sided shape is a Quadrilateral.

Mohan: Yes, But remember the sides have to be straight, and it has to be 2-dimensional. See this diagram I have drawn below:

It can be represented by ▽ ABCD.



Trapezium:

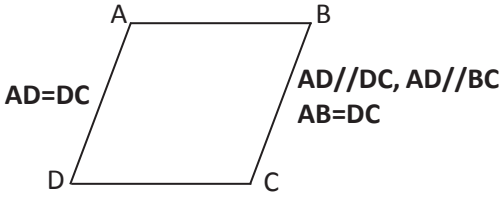
A trapezium is a quadrilateral with one pair of sides parallel to each other.

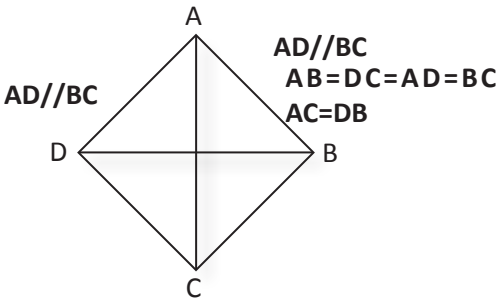
The parallel sides that are AB and CD are called bases of the trapezium.

The non-parallel sides are called legs of the trapezium.

Parallelogram:

The opposite pair of the quadrilateral is parallel and equal to each other.



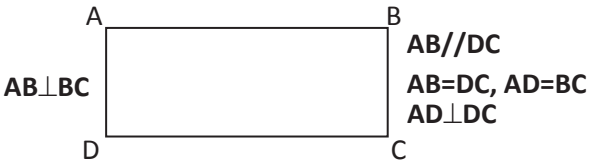


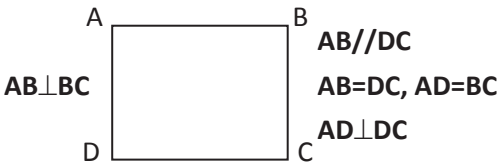
Rhombus:

It is a parallelogram where all the sides are equal to each other. The diagonals are also equal to each other.

Rectangle:

It is a parallelogram with opposite sides equal and adjacent sides perpendicular to each other.

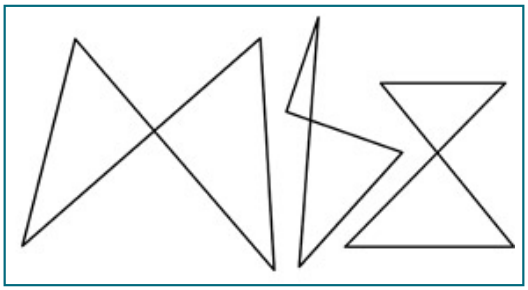




Square:

It is a parallelogram with all sides equal. The adjacent sides are perpendicular to each other.

Oh Yes! One other thing you must bear in mind is - When two sides cross over, you call it a "Complex" or "Self-Intersecting" quadrilateral like these:



They still have 4 sides, but two sides cross over.

That's it for quadrilaterals! Next, let's learn about some other polygons.

Sailesh: Polygons? What are those?

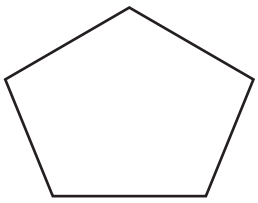
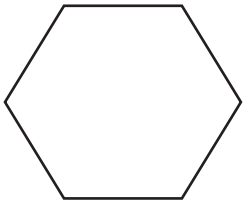
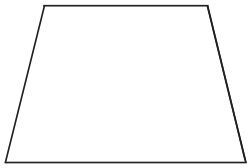
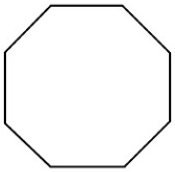
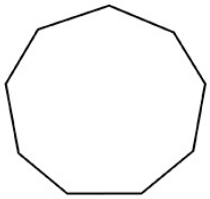
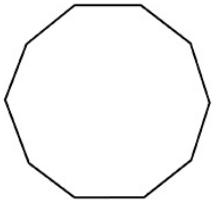
Mohan: Polygons are 2-dimensional shapes. They are made of straight lines, and the shape is closed.

Sailesh: closed?

Mohan: It means all the lines connect.

Sailesh: That means that all the shapes we learned about today, like the triangle and the various quadrilaterals are all polygons.

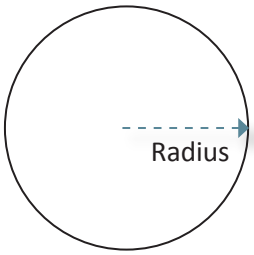
Mohan: That's right Sailesh! Some other polygons you should know about are:

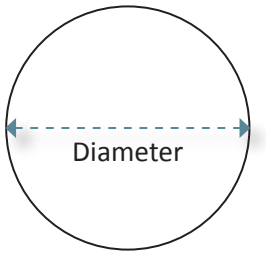
		
Pentagon: Shape with 5 sides	**Hexagon:** Shape with 6 sides	**Heptagon**: Shape with 7 sides
		
Octagon: Shape with 8 sides	**Nonagon**: Shape with 9 sides	**Decagon**: Shape with 10 sides

Circles / Elliptical Shapes

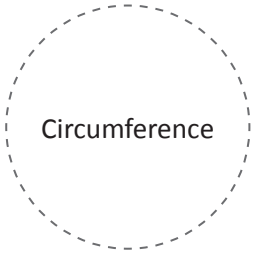
Sailesh: So we have learned a great deal about the various 2 dimensional shapes. However there is one more shape I would like to learn about. Most of us already know how to draw a circle, but did you know that in a perfect circle all its points are at the same distance from the center? When I began reading about the circle, there were a lot of confusing terms related to a circle. Mohan then simplified these for me. I have penned these down for you. Here take a peek!

- The Radius is the distance from the center to the edge.
- The Diameter starts at one side of the circle, goes through the center and ends on the other side.





- The distance all around the circle is called the Circumference of the circle.



And friends.here is the really cool thing: When you divide the circumference by the diameter you get 3.141592654. which is the number π (Pi). Simply put, when the diameter is 1, the circumference is 3.141592654.

With that we come to a close of this interesting session. I will see you soon with more information on shapes. Meanwhile, why don't you practice drawing different shapes? Have Fun!

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. Differentiate between line, line segment and ray.

2. Define the terms given below in your own language:
   a. Diameter
   b. Heptagon
   c. Polygons
   d. Rhombus
   e. Obtuse Triangle

3. Name the various types of angles you have studied about in this session. Describe them in one or two lines.

Worksheet

1. *Answer these fun riddles for Sailesh and learn more about 2D shapes and their attributes.*

   a. I am any closed shape with straight sides. Who am I? _____

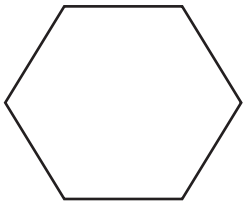
   b. I have three sides and the same amount of angles. What shape am I? _____

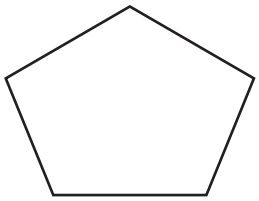
   c. I am a quadrilateral. All my sides are always the same length. I have all right angles. Who do you think I am? _____

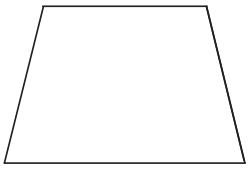
   d. Look at my angles! There are three and two of them are equal. Who am I? _____

   e. I have more sides than a quadrilateral, but fewer sides than a hexagon. Who could I be? _____

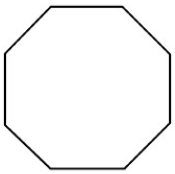
2. *Sailesh has muddled all the shapes in haste. Can you match the shape pictures to the correct names?*

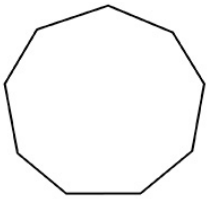






Pentagon	Hexagon	Trapezium





Heptagon	Rhombus

3. *1.If a circle has a diameter of 14 inches, what will its circumference be? Use 22/7 as an approximation for π.*

_____

_____

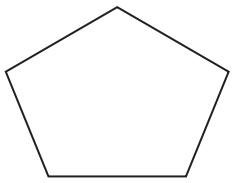
Notes

Answers

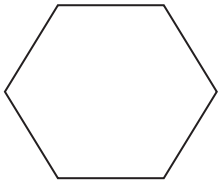
1.

a. Polygons	b. Equilateral Triangle	c. Square	d. Isosceles Triangle
e. Pentagon			

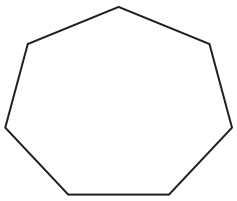
2.



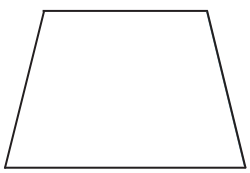
Hexagon



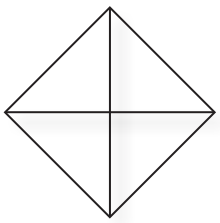
Pentagon



Rhombus



Heptagon



Trapezium

3.

44 inches

Basic Geometry-3D

At the end of this module you will be able to:

- identify the different 3D geometrical shapes;
- differentiate between 2D and 3D shapes.

Session Plan	
1	2D and 3D
2	X,Y and Z Axis
3	3 Dimensional Shapes
4	Key Learnings
5	Worksheet

2D and 3D

Shailesh has just entered the workshop and finds Mohan working with a cube. Excited to show Mohan how much he has learned about shapes he picks up the cube and says:

Sailesh: This is a square, isn't it?

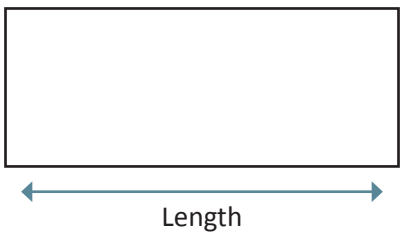
Mohan: Not quite Sailesh! This is a cube.

Shailesh: Aren't they the same thing?

Mohan: Not really! A square is a 2D (two dimensional) shape, whereas a cube is a 3D (three dimensional) shape.

Sailesh: 3D shape! What is a 3D shape? ... How is it different from a 2D shape? I'm so confused!

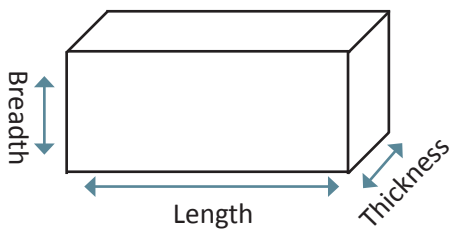
Mohan: Relax Shailesh, don't get confused. Let me explain you the difference between 2D and 3D shapes in geometry.



All 2D shapes are those which can be made on a plane surface. A 2D shape has no thickness. Like this rectangle I have drawn on a plane piece of paper.

Shailesh: Oh! That makes it clearer. Simply said, if you add thickness to any 2D shape it becomes a 3D shape.

Mohan: Perfect!



X, Y and Z Axis

Mohan: Now that you have learned about the 2D and 3D shapes. Let me explain to you about the X, Y and Z Axis.

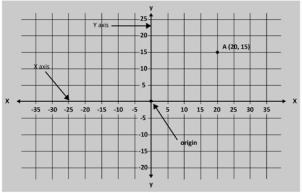
Shailesh: Axis? What's that?

Mohan: Well, 2D shapes can be plotted on X and Y axis. Axis is a reference line drawn on a graph. Here is a graph with an X Axis and a Y Axis. You can measure from it to find values.

The left-right (horizontal) direction is commonly called X. The up-down (vertical) direction is commonly called Y. The point where X and Y meet is called origin and generally marked as O. You measure everything from there. As you can see each axis has marking on it. Both the axis run infinitely and includes both positive and negative values.

Shailesh: You mean both X and Y can have 'n' number of values?

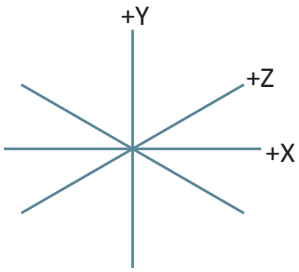
Mohan: Yes definitely! As X increases, the point moves further right. When X decreases, the point moves further to the left. As Y increases, the point moves further up. When Y decreases, the point moves further down.



Sailesh: Oh I see! But Mohan, why do we use axis?

Mohan: Ahh...That's an important question Sailesh...The values both negative and positive are used to determine the distance between two points.

Shailesh: What about the 3D shape? How do we use axis in 3D shape?



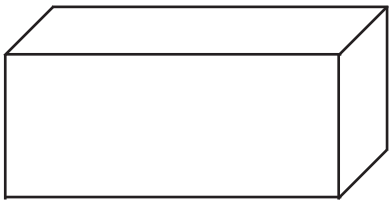
Mohan: Good going Sailesh this goes on to show how keen you are to learn. Well, when we plot a 3D shape on the axis then we need a third axis called Z axis to accommodate the third dimension of a 3D shape.

Shailesh: Let us learn more about three dimensional objects.

3 Dimensional Shapes

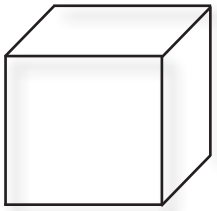
Sailesh has just learned about some of three dimensional objects. He is excited to share his knowledge with you.

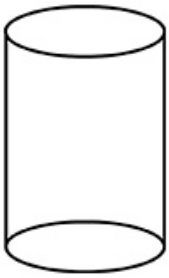
Sailesh: Hey friends, guess what? I have learned some interesting facts about three dimensional shapes. I have penned down my thoughts in the chart below, here take a look.



Cuboid – It is the three dimensional figure of a rectangle. The three dimensions are length, breadth and height

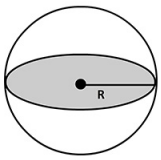
Cube - It is the three dimensional figure of a square. All the sides of a cube are equal

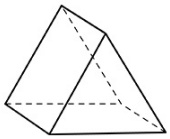




Cylinder - Cylinder is a three dimensional object with a circular base and straight parallel sides that looks similar to a can of cool drink; Pipes, Solid rods are examples.

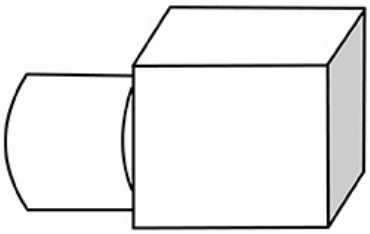
Sphere - Sphere is a perfectly round three dimensional shape similar to a round ball you might play sport with.





Prism - Prism is the solid shape you get if you move a flat polygon through space. A prism has five planes: a top, a bottom, and three sides. The most common kind of prism is a triangular prism, made by moving a triangle through space.

Multidimensional 3D - Sometimes a 3 D object items a single piece combining a cylinder and a cube.



That was easy! 3D shapes are very common in our world. From cardboard boxes to buildings we see them everywhere. Why don't you explore these shapes further, while I go check in with Mohan.

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. How is a two dimensional shape different from a three dimensional shape? Substantiate your answer with an example.

2. What is an axis? Chart out a graph showing all three axes.

3. Give some examples three-dimensional geometric shapes from the world around you.

Worksheet

1. *Help Sailesh in choosing the correct answer from the options given below:*

1. A triangle is a closed planar shape with
   a. 2 sides
   b. 4 sides
   c. 3 sides
   d. 5 sides

2. Which of the statements below best describes a square?
   a. A square has 4 equal sides and 4 right angles
   b. A square has 4 equal sides
   c. A square has 4 right angles
   d. 2 pairs of parallel sides

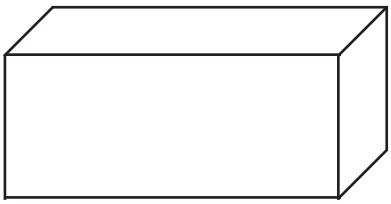
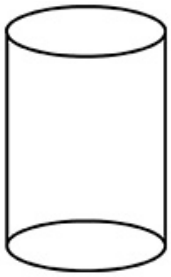
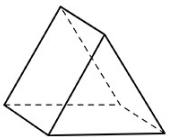
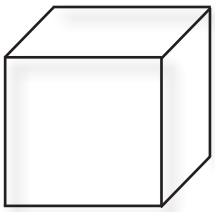
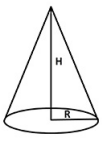
4. Which shape is like a can of cool drink?
   a. A cylinder
   b. A prism
   c. A cube
   d. A cuboid

5. Which 3D shape looks like a playing ball?
   a. Square
   b. Sphere
   c. Cuboid
   d. Prism

6. What is the distance from the center of a circle to its edge called?

a. Radius
b. Diameter
c. Length
d. Breadth

2. *Sailesh is confused about the various kinds of geometrical shapes. Help him by matching the correct image of the shape from column A with the correct shape name from column B.*

Column A	Column B
	Cylinder
	Triangle
	Cube
	Cone
	Prism

Notes

Answers:

1.

1.	c	2.	a	3.	a	4.	b	5.	a

2.

a.	iii	b.	i	c.	ii	d.	v	e.	iv

Reading Engineering Drawing

At the end of this session, you will be able to:

- read engineering drawings.

Session Plan	
1	What is an Engineering Drawing
2	Methods of Representation
3	Lining
4	Dimensioning
5	Scale Specification
6	Sectioning
7	Projection
8	Key Learnings
9	Worksheet

What is an Engineering Drawing

As Sailesh settles down for another day at the workshop, he sees Mohan leaf through a sheaf of papers. He wonders what's in the papers. He peeps over Mohan's shoulders and sees a drawing – More like a blueprint.

Sailesh: What are these papers Mohan? What do these drawings signify?

Mohan: (Pointing at the drawing) this is an engineering drawing.

Sailesh: An engineering drawing?

Mohan: An engineering drawing is the technical term that is used to communicate size, shape and features of a desired object. The purpose of engineering drawing is to graphically convey the ideas and information necessary for the construction or analysis of component or work piece. Simply said, it is a blueprint of the work piece you will create. It gives you the exact dimensions and tolerances.

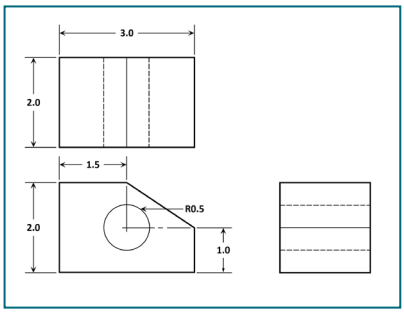
Sailesh: Oh I see... So before I start creating a work piece, I must understand how to interpret this drawing.

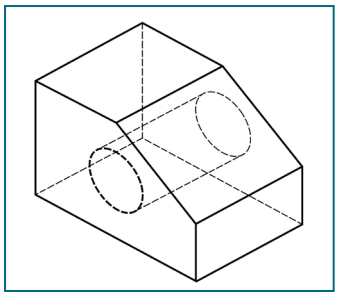
Mohan: Exactly Sailesh! The drawings prepared by any technical person must be clear and leave no scope for misinterpretation, or else confusion may arise. On the other hand, a machinist should be able to read the drawing carefully. Let me show you how to interpret an engineering drawing.

Methods of Representation

Mohan: There are two ways an engineering drawing can be represented:

1. **2D Drawing**: - Two dimensional is a concept that describes anything that composes of length and width. An object is regarded as two dimensional if it has two coordinates, which define the points within it. This is a mathematical and physics concept.

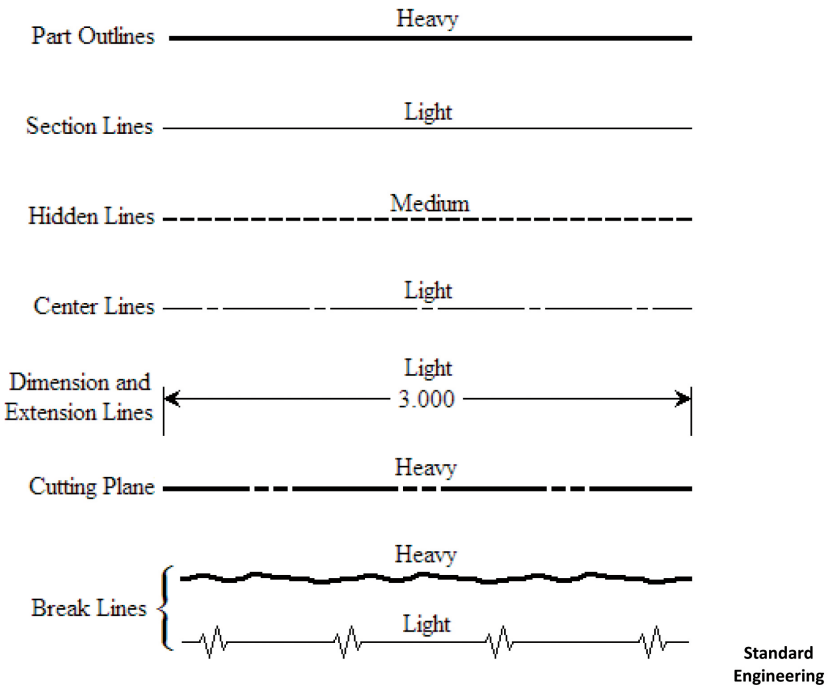




2. **3D Drawing**: - Three dimensional is a concept that describes anything that composes of length, width, breadth. An object is regarded as three dimensional if it has three coordinates, which define the points within it. This is a mathematical and physics concept.

Lining

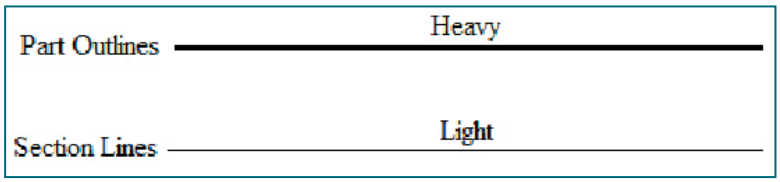
Mohan: There are many type of lines used in engineering presentation. These have a great importance as as they give different message as we givesimilar to the different expression in our voice during conversation. Let us look at some examples for your better understanding:



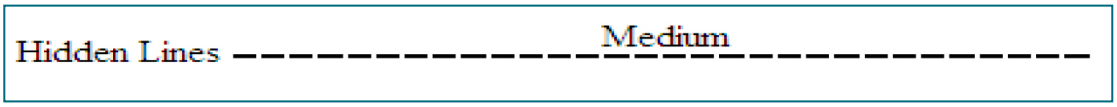
Standard Engineering Drawing Line Types

A variety of line styles graphically represent physical objects. Let's look at these in more detail.

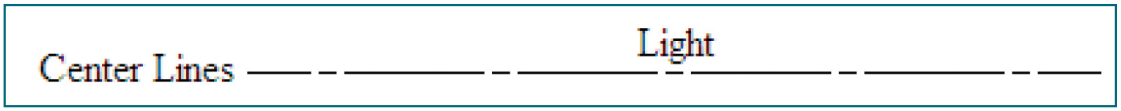
Visible Line: - A visible line is a continuous line used to depict edges directly visible from a particular angle. For instance part outlines and section lines.



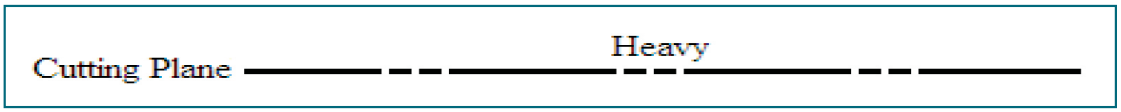
Hidden Line: - A hidden line is short-dashed lines that may be used to represent edges that are not directly visible.



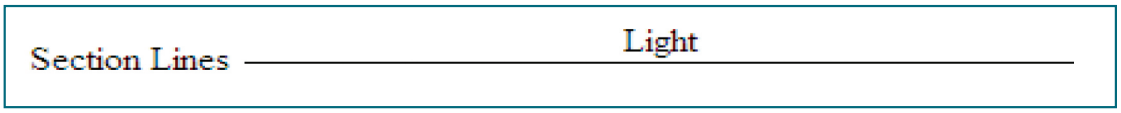
Center Line: - A center line is alternately long- and short-dashed lines that may be used to represent the axes axis of circular features.



Cutting Plane Line: - Cutting plane lines are thin, medium-dashed lines, or thick alternately long- and double short-dashed that may be used to define sections for section views.



Section Line: - Section lines are thin lines in a pattern (pattern determined by the material being "cut" or "sectioned") used to indicate surfaces in section views resulting from "cutting." Section lines are commonly referred to as "cross-hatching."



Lines can also be classified by a letter classification in which each line is given a letter.

Type A lines show the outline of the feature of an object. They are the thickest lines on a drawing and done with a pencil softer than HB.

Type B lines are dimension lines and are used for dimensioning, projecting, extending, or leaders. A harder pencil should be used, such as a 2H.

Type C lines are used for breaks when the whole object is not shown. They are freehand drawn and only for short breaks. 2H pencil.

Type D lines are similar to Type C, except they are zigzagged and only for longer breaks. 2H pencil

Type E lines indicate hidden outlines of internal features of an object. They are dotted lines. 2H pencil.

Type F lines are Type F[typo] lines, except they are used for drawings in electro technology. 2H pencil.

Type G lines are used for centre lines. They are dotted lines, but a long line of 10–20 mm, then a gap, then a small line of 2 mm. 2H pencil.

Type H lines are the same as Type G, except that every second long line is thicker. They indicate the cutting plane of an object. 2H pencil.

Type K lines indicate the alternate positions of an object and the line taken by that object. They are drawn with a long line of 10–20 mm, then a small gap, then a small line of 2 mm, then a gap, then another small line. 2H pencil.

Dimensioning

Sailesh: Now I know what the different lines mean but, what about the dimensions. How will I know what width and thickness to give?

Mohan: That's a good question Mohan. In addition to a complete shape description of an object, a drawing of the design must also give a complete size description. The process of writing or giving measurement of the object on a prepared drawing is known as dimensioning. Every engineering drawing should show complete size descriptions stating length, width, thickness, diameter of holes grooves etc and such other detail related to its construction.

The correct placement of dimensions is strictly prescribed by an extensive list of drawing conventions.

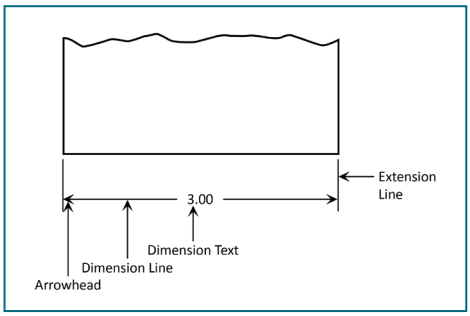
Let's look at the lines used in dimensioning.

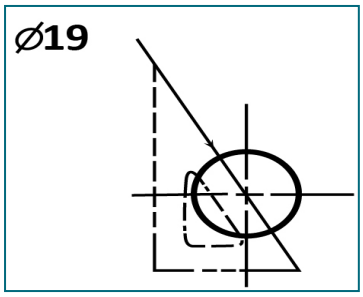
Dimension line: A dimension line is a thin, dark line broken in the middle to allow the placement of the dimension value, with arrowheads at each end.

Dimension text: The dimension text denotes the dimension value.

Extension line: An extension line is a thin, dark line that extends from a point on the drawing to which a dimension refers.

Arrowhead: An arrowhead indicates the extent of dimensions. It is approximately 3 mm long and 1 mm wide. That is, the length is roughly three times the width.





Leader: A leader is a thin, dark, solid line that leads from a note of dimension and terminates in an arrowhead on a line or edge of a hole.

Scale Specification

Sailesh: Mohan, these drawing are so tiny. How can one make an actual object just reading and interpreting these?

Mohan: Drawing of every object cannot be prepared in full scale because they would be too big to accommodate on the drawing sheet. Drawing of too small objects also cannot be prepared in full size because they would be too small to draw and to read. Hence, a convenient scale is chosen to prepare

the drawing of big as small object in proportionality smaller or larger in size. So the scales are used to prepare a drawing at a full size or enlarged size.

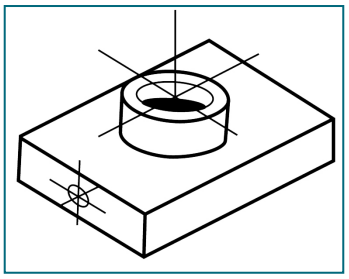
The ratio of the drawing to the object is called representative fraction. It is denoted by R.F.

R.F. = Length of an object in drawing / Actual length of the object

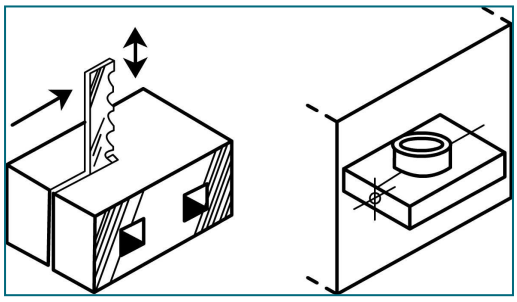
Sectioning

Sailesh: What about the interior details? How does one show these on an engineering drawing?

Mohan: I'm glad you asked…There are many times when the interior details of an object cannot be seen from the outside. For instance, this isometric drawing below does not show all details.



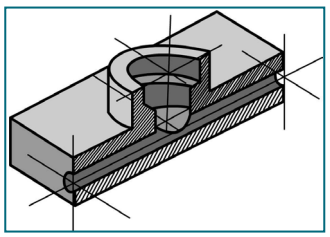
An Isometric Drawing



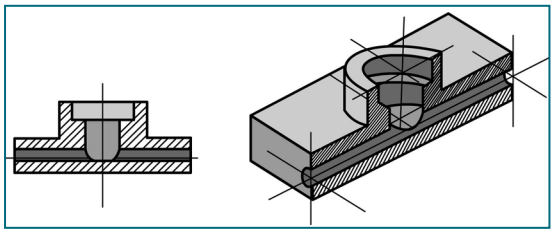
"Sectioning" an object

We can get around this by pretending to cut the object on a plane and showing the "sectional view". The sectional view is applicable to objects like engine blocks, where the interior details are intricate and would be very difficult to understand through the use of "hidden" lines (hidden lines are, by convention, dotted) on an orthographic or isometric drawing. Imagine slicing the object in the middle as shown below:

Once the object is sectioned it will look something like this:



Section of the object

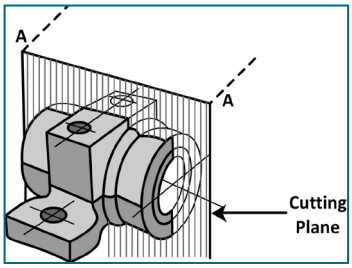


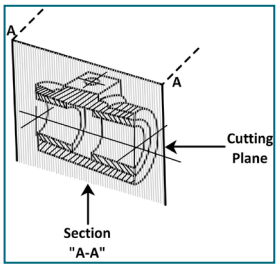
Take away the front half and what you have is a full section view. The cross-section looks like the figure given below when it is viewed from straight ahea

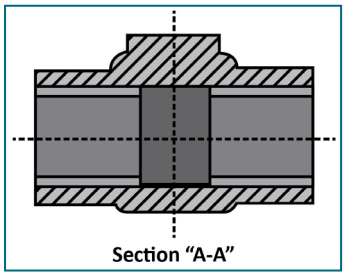
There are many types of sectional views. Let's look at these one by one:

Cross-Section View: A cross-sectional view portrays a cut-away portion of the object and is another way to show hidden components in a device.

Imagine a plane that cuts vertically through the center of the pillow block as shown in figure 15. Then imagine removing the material from the front of this plane, as shown below:





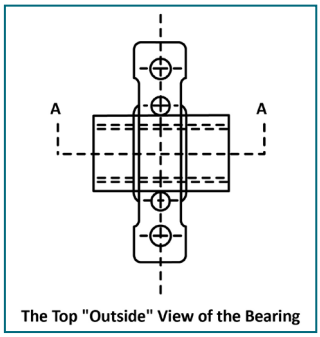


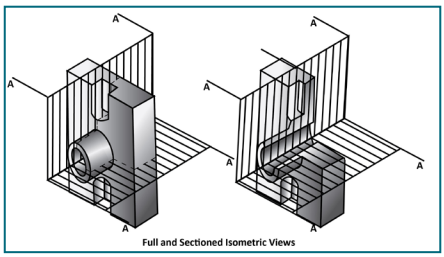
This is how the remaining rear section would look. Diagonal lines (cross-hatches) show regions where materials have been cut by the cutting plane.

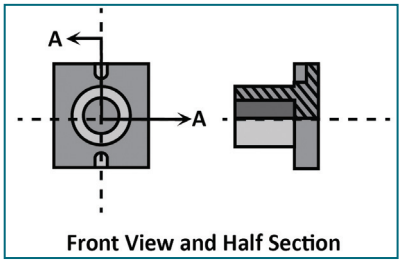
This cross-sectional view (section A-A one that is orthogonal to the viewing direction), shows the relationships of lengths and diameters better. These drawings are easier to make than isometric drawings. Seasoned engineers can interpret orthogonal drawings without needing an isometric drawing, but this takes a bit of practice.

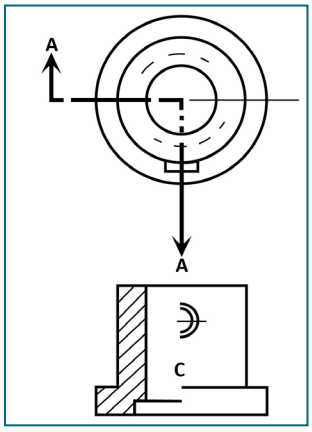
The top "outside" view of the bearing is shown in the figure given below. It is an orthogonal (perpendicular) projection. Notice the direction of the arrows for the "A-A" cutting plane.

Half-Section View: A half-section is a view of an object showing one-half of the view in section, as in figure 19 and 20.vt









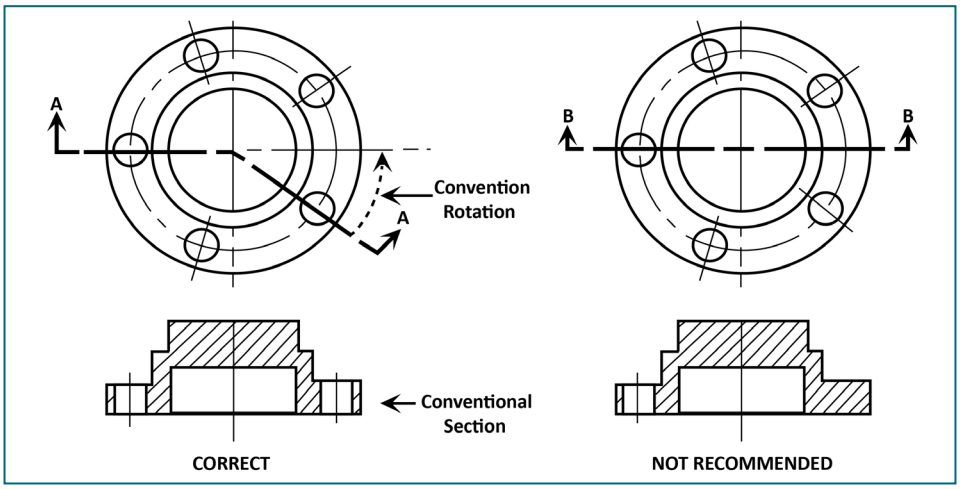
The diagonal lines on the section drawing are used to indicate the area that has been theoretically cut. These lines are called section lining or cross-hatching. The lines are thin and are usually drawn at a 45-degree angle to the major outline of the object. The spacing between lines should be uniform.

A second, rarer, use of cross-hatching is to indicate the material of the object. One form of cross-hatching may be used for cast iron, another for bronze, and so forth. More usually, the type of material is indicated elsewhere on the drawing, making the use of different types of cross-hatching unnecessary.

Usually hidden (dotted) lines are not used on the cross-section unless they are needed for dimensioning purposes. Also, some hidden lines on the non-sectioned part of the drawings are not needed (figure 12) since they become redundant information and may clutter the drawing.

Sectioning Objects with Holes, Ribs, Etc.

The cross-section on the right of figure given below is technically correct. However, the convention in a drawing is to show the view on the left as the preferred method for sectioning this type of object.



Projection

Mohan: Engineers are confronted with the task of communicating the design, development and structures of machines to manufacturers and builders. The shape and size of various parts of a machine and its structure must be recorded on plane sheets in a systematic way for communication. The pictorial view of the object does not carry all the details, especially the inner details and correct shape of complicated parts. Different methods, therefore, are implied for describing the exact shape based on the 'projections' drawn by engineers.

Sailesh: Projections? What are those?

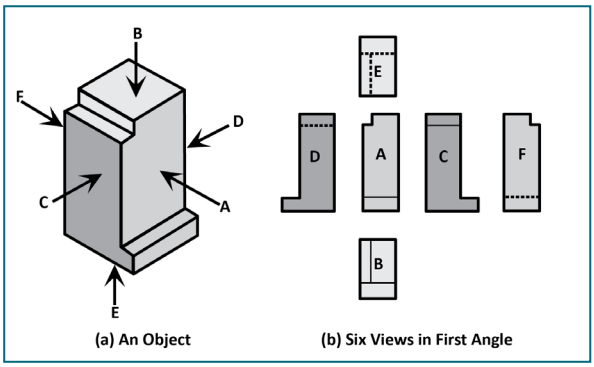
Mohan: Let me explain it to you…let's take this workpiece (image given below). It is a solid piece, with six sides. Now, if you have to make a sketch and send it to the manufacturer, so that he can create hundred such numbers. What will you do?

Sailesh: I would send him a 3D figure of course.

Mohan: Yes, but, that itself would show only three dimensions.

Sailesh: Oh yes… But, Mohan how else would we do it?

Mohan: That's where projections come handy. One has to show the drawing from all six sides for the manufacturer to understand the shapes required by us in the finished piece. This six sided view are known as projections.



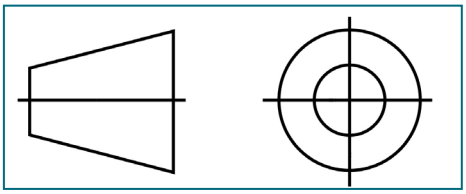
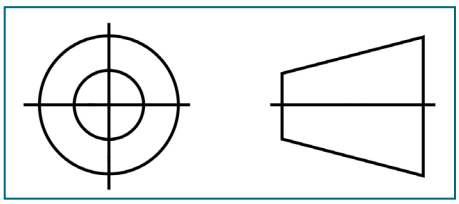
The designer first imagines the geometry of the piece required and then creates the projection by imagination. When these projections are put together we can very well imagine how the finished piece would look like. To accurately describe even the simplest object we need at least two Orthographic views. Typically an elevation and a plan are used. The Elevation typically gives us heights and hidths, the plan gives us widths and depths of the object. With these two views we can imagine the front and top of the object.

There are two common ways to project a drawing, keeping in mind the angles.

First Angle Projection Method: This method of projection is now recommended by the "Bureau of Indian Standards" from 1991. In this method, when the views are drawn in their relative positions, Plan (Top view) comes below the elevation (Front view), the view of the object as observed from the left- side is drawn to the right of elevation. The object lies between the observer and the plane of projection. The plane of projection is assumed to be non- transparent.

Third Angle Projection Method: This method of projection is used in U.S.A and also in other countries. In this method, when the views are drawn in their relative positions, the Pplan comes above the elevation; left hand side view is drawn to the left hand side of the elevation. The plane of projection lies between the observer and the object. The plane of projection is assumed to be transparent.

Because of the difference between projection methods, it may be difficult for Lathe operators to interpret drawings. It is important to indicate on the drawing which system we are using. The symbols for First Angle and Third Angle projection are shown below. The symbols are derived from a cut cone which has been projected in First or Third Angle respectively:

Projection	Symbol
First Angle Projection	
Third Angle Projection	

Sailesh: Friends, after completing this session I feel fairly competent to read and interpret technical drawings. A little more practice – and I will be able to determine the exact machining operations. Why don't you take a look at a few drawings while I go study a detailed component drawing with Mohan?

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

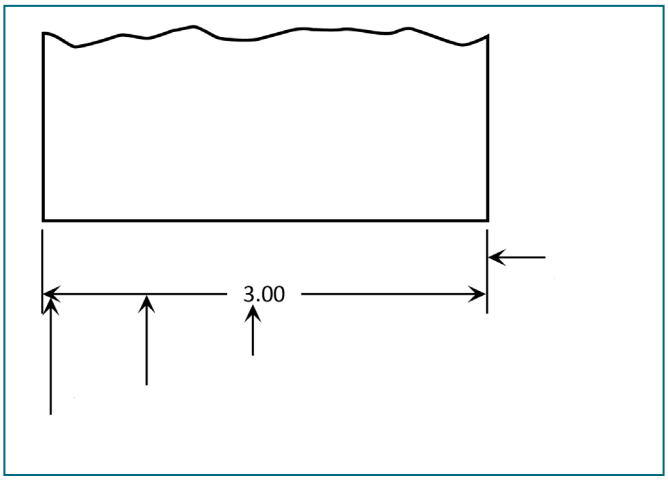
1. What is an engineering drawing? Why is it important for a Lathe operator to know how to read an engineering drawing?

2. Write a detailed note on the types of sectional views.

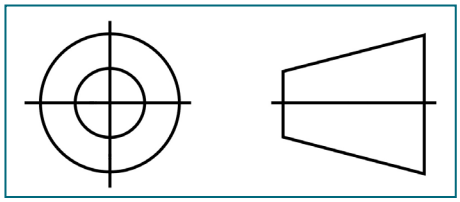
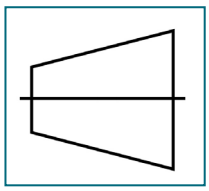
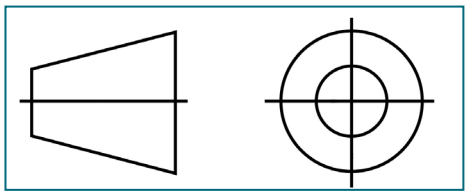
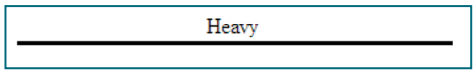
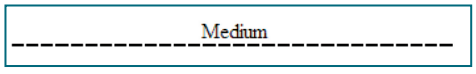
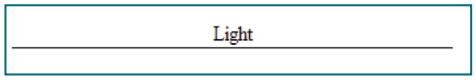
3. Chart out the difference between First Angle Projection and Third Angle Projection.

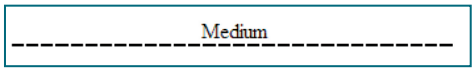
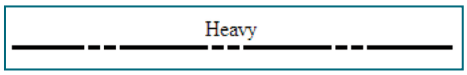
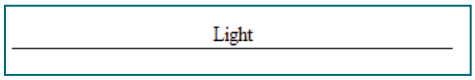
Worksheet

1. *Sailesh seems to have forgotten all about drawing conventions used in an engineering drawing. Help him out by labeling the lines used in dimensioning*



2. *Tick the correct Option:*

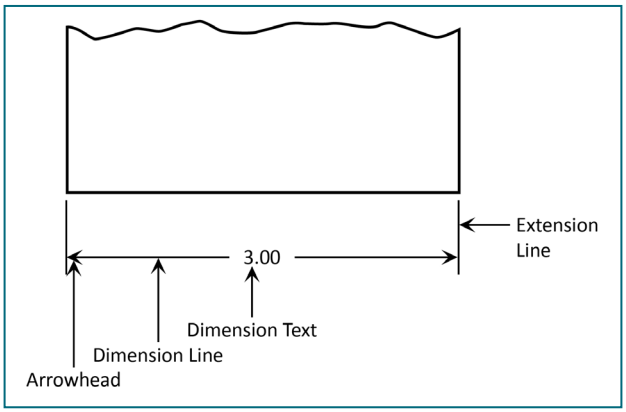
Projection	Symbol
a. The symbols for First Angle Projection is:	i. 
	ii. 
	iii. 
b. Sailesh wants to draw a part of the object that is not directly visible. Which of these lines should he use to represent the edges?	i. 
	ii. 
	iii. 

c.	A Cutting plane line is represented by:	i. 
		ii. 
		iii. 
d.	Type A lines show the:	iv. hidden outlines of internal features of an object.
		v. breaks when the whole object is not shown
		vi. outline of the feature of an object

Notes

Answers:

1.



2.

a. iii

b. ii

c. ii

d. iii

Precision Measurement – Vernier Calliper

At the end of this module you will be able to:

- explain the working principle;
- measure the different dimensions of using Vernier Calliper.

Session Plan	
1	Vernier Calliper
2	Parts of Vernier Calliper
3	Least Count
4	Reading Calculation Procedure
5	Zero Error
6	Key Learnings
7	Worksheet

Vernier Calliper

Mohan has just entered the workshop and finds Sailesh fiddling with the Vernier Calliper. Worried that Sailesh might impair the measurement instrument he asks Sailesh to keep it back.

Mohan: Shailesh, Is that a Vernier Calliper in your hand?

Sailesh: Vernier Calliper??? Oh! Is this scale called a Vernier Calliper?

Mohan: Yes, that a Vernier Calliper. It is used for measurement purposes.

Sailesh: Oh! I see.

Mohan: It helps you in getting accurate measurements. A Vernier Calliper consists of two scales - one is fixed while the other one is movable. The fixed scale is called a main scale which is calibrated on L-shaped frame and carries fixed jaws. The movable scale which is also called vernier Scale slides over the main scale and carries a moveable jaw.

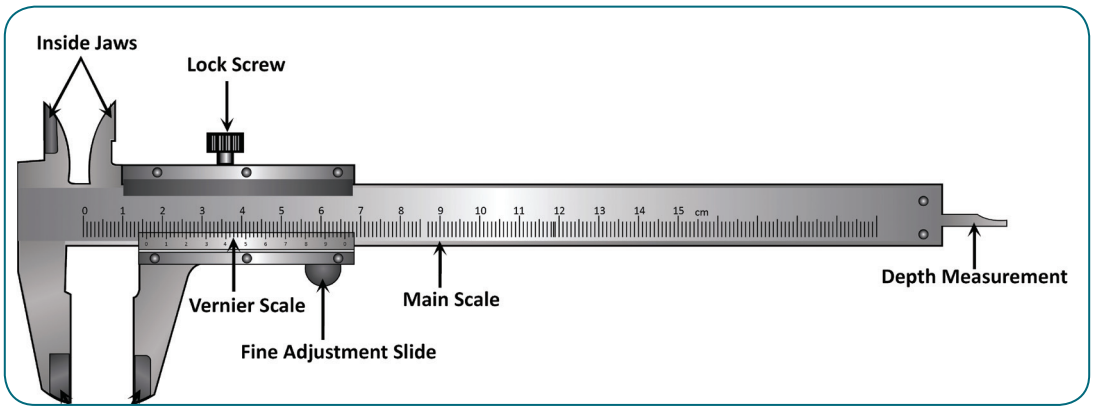
Sailesh: Ok, but I still can't see how we can take measurements with it.

Mohan: Sailesh, see on the vernier calliper, the movable and fixed jaws carries a measuring tip. When the two jaws are closed the zero of vernier scale coincides with the zero of main scale. A lock nut is provided to lock the sliding scale on the fixed main scale.

Parts of a Vernier Calliper

Sailesh: Hey Friends, I have learned about the various parts of the Vernier Calliper. For you to understand better, I have labeled these parts in the image given. Why don't you take a look!

1. Vernier Scale
2. Lock Screw
3. Fine Adjustment Slide
4. Lock Screw
5. Beam
6. Fine Adjustment Device
7. Movable Jaw
8. Stationary Jaw



Least Count

Mohan: Now that you have learnt about the various parts of a Vernier Calliper are you excited to see how it actually functions?

Sailesh: Yes!! Of course!

Mohan: First let's understand how accurate a Vernier calliper is…Have you heard about least count?

Sailesh: Least count? What is that?

Mohan: The least count is the smallest distance that a vernier calliper can measure accurately. The least count is the difference between the value of the main scale division and vernier scale division. Thus, least count of a vernier instrument = (value of the smallest division on the main scale)-(the value of the smallest division on the vernier scale).

For instance, 49 main scale divisions are divided into 50 vernier scale divisions

Value of 1VSD=49/50mm

Least count = 1MSD-1VSD

=1-49/50

=1/50=0.02mm

Sailesh: Of I see. So the least count in this case will be 0.02mm.

Mohan: Precisely!

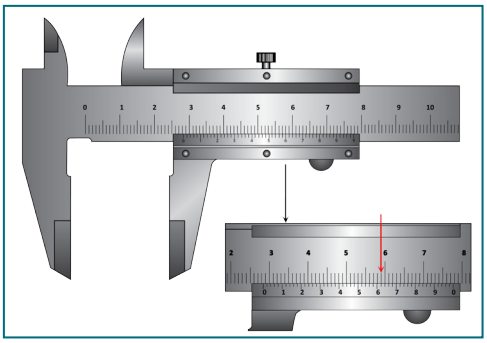
Reading Calculation Procedure

Mohan: Moving on, let's understand the procedure of reading the measurement value of Vernier Calliper.

For reading the measured value of Vernier Caliper you can use the statement I have written below:

The nth number of MSD that the zero of the vernier scale has crossed + the n'th number of VSD coinciding with the MSD X (the least count of the instrument)

For instance, in the figure given below the zero of vernier crossed 34 divisions in main scales so main scale reading is 34 and the 54th line coincides with the main scale. Thus, the Vernier scale reading will be:



54 x 0.02 = 1.08

The total reading is 34 + 1.08 = 35.08 mm

Sailesh: Ok. That was tricky, but I think once I get a hang of it I will be able to measure accurately.

Mohan: You're right Sailesh! It's quite easy but you must take some precautions. Ensure that the line of measurement coincides with the line of scale. In addition to this you must also hold the Vernier Calliper straight and view it perpendicularly.

Sailesh: Don't worry Mohan. I will heed your advice.

Zero Error

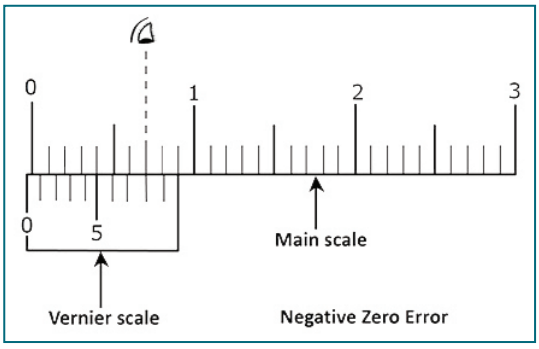
Sailesh has been practicing with the Vernier Calliper but is unable to fathom why his measurements are inaccurate. Confused, he seeks out Mohan's help. After working on the instrument for a while, Mohan deems that the Vernier Calliper has a defect of zero error.

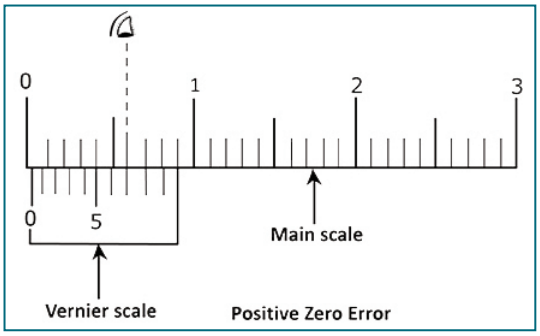
Mohan: Sailesh! This Vernier Caliper has a defect of zero error.

Sailesh: Zero error! What's that?

Mohan: When the zero of the main scale and the zero of the vernier scale doesn't coincide when the jaws of the Vernier Calipers is completely closed, it is said the instrument has defect of zero error. The zero error is of mainly two types:

1. Positive zero error: The zero of the vernier scale lie right to the zero of the main scale.
2. Negative zero error: The zero of the vernier scale lie let to the zero of the main scale.





Sailesh: So does that mean the Vernier calliper is faulty and is rendered useless?

Mohan: Not really, Sailesh. There is a process to correct the zero error.

Say the VSD=2

Then the error = VSD x LC = 2 x 0.02mm = 0.04mm

So, in case of Positive Zero Error

The actual reading = Observed value – Error value = 35.08 – 0.04 = 35.04 mm

In case of negative Zero Error

The actual reading = Observed value + Error value = 35.08 + 0.04 = 36.02 mm

Sailesh: Brilliant!

So friends, that was all about Vernier Calliper. I hope you're as excited as I am to get some practice on a Vernier Calliper. So while I get some practice here in the workshop why don't you people head out to your lab for some real action!

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. What is a Vernier Calliper? Name its various parts.

2. Describe Zero error. How to calculate zero error?

3. What is least count and give the formulae to calculate least count?

Worksheet

1. *Fill in the blanks:*

   a. The zero type error are of mainly two types_____ and _____ .

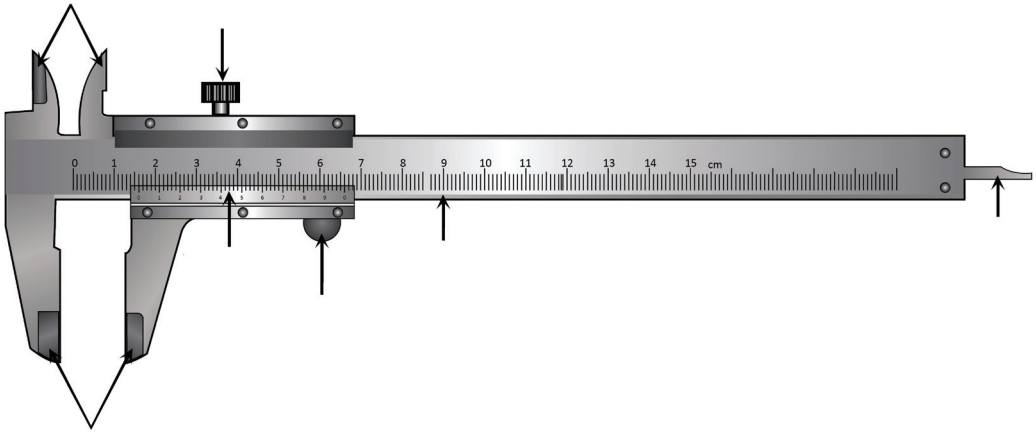
   b. The_____is the smallest distance that a vernier calliper can measure accurately.

c. _____ is used to get accurate measurements.

d. The fixed scale is called a _____ which is calibrated on _____ and carries fixed jaws.

e. The least count is the difference between the value of the _____ and _____ .

2. *Given below is the image of vernier calliper. Help Sailesh to mark various parts of vernier calliper in the image to guide.*



Mark these parts:

a. Jaws for measuring inner dimensions

b. Jaws for measuring outer dimensions

c. Screw clamp

d. Stem for measuring depths

e. Vernier scale

f. Main scale

Notes

Answers:

1.

a. positive zero error and negative zero error

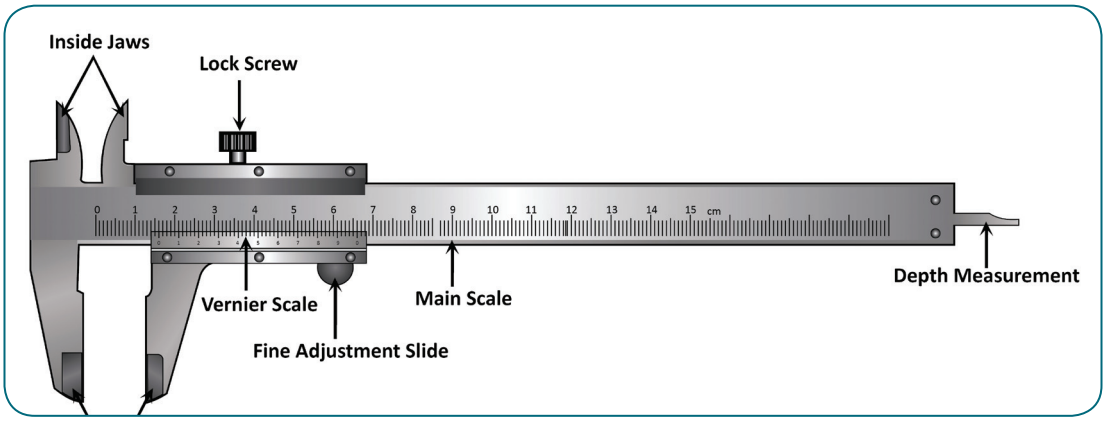
b. the least count

c. Vernier Calliper

d. main scale and L-shaped frame

e. main scale division and vernier scale division.

2.



Precision Measurement - Micrometer

At the end of this module you will be able to:

- identify the parts of the Micrometer;
- explain the working principle.

Session Plan	
1	Micrometer
2	Parts of Micrometer
3	Pitch
4	Least Count
5	Zero Error
6	Determination of Diameter of a Wire
7	Key Learning
8	Worksheet

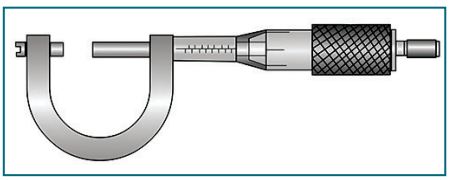
Micrometer

Mohan watches a preoccupied Sailesh holding a measuring tape in his hand. Interrupting his thoughts he asks Sailesh.

Mohan: Hello Sailesh! What are you doing with this measuring tape?

Sailesh: Hello Mohan! I was just wondering if I can measure the length of this tin sheet with a measuring tape, but what if I need to measure its thickness? Wouldn't it be a tedious task to measure its thickness?

Mohan: Hmm...There is a special instrument called Micrometer or Micrometer Screw Gauge. It is used to measure very small dimensions. You can even measure thickness of a wire or thickness of a metal sheet with the help of Micrometer. A Micrometer looks something like this:



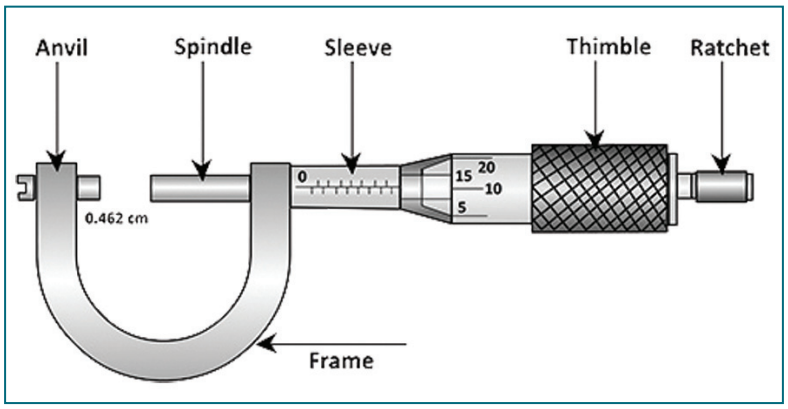
Sailesh: That's quite amazing! I can hardly see the thickness of wire leave alone measuring it.

Mohan: Yes, Sailesh you will be surprised to know that a Micrometer can measure thickness upto 0.01mm accurately.

Sailesh: Oh that's great.

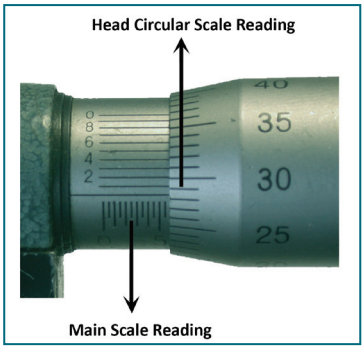
Parts of Micrometer

Mohan: Let's learn more about its various parts. Take a peek at this labeled figure given below:



Mohan: Now, let's talk about some important parts of the Micrometer in detail. A Micrometer is composed of:

- **Frame** - It is a C-shaped body that holds the anvil and barrel in constant relation to each other. It is thick as it needs to minimize flexion, expansion and contraction, which would distort the measurement.
- **Anvil** - The upper end of the frame towards which the spindle moves is called the Anvil, it is also known as Stud.
- **Spindle** - It is a metal block which can move forward and backwards. The object to be measured is held in between the spindle and Anvil.
- **Sleeve** - The stationary cylinder with the linear scale on it shows the main scale reading. A reference line showing readings in millimeters are drawn on the cylinder attached to the nut is called the main scale reading.
- **Thimble** - It is a hollow cylinder which houses the cylinder with main scale reading. The thimble also has markings and is divided into equal parts, generally 50 to 100 parts. This is the head scale or circular scale reading.
- **Ratchet** - It is the screw head provided at the end of the Micrometer. It is used to rotate the spindle.



Sailesh: That's a lot of names to remember!

Mohan: Don't worry Sailesh. When you start working with these instruments you will remember these with no extra effort.

Pitch

Mohan: Moving on... Let me tell you about the pitch of the Screw Gauge.

Sailesh: Pitch? What does a pitch have to do with a measuring instrument?

Mohan: Pitch of the screw is the distance moved by the spindle per revolution. For instance, the Screw Gauge has a pitch of 0.5 mm. Hence in this case, for one revolution of the screw the spindle moves forward or backward 0.5 mm. This movement of the spindle is shown on an engraved linear millimeter

scale on the sleeve.

Sailesh: So, the pitch can be defined as the distance advanced by the screw tip when the head is given one complete rotation.

Mohan: That's right Sailesh!

Sailesh: Mohan, but how does one calculate the pitch.

Mohan: It's quite easy all you need to is follow these steps:

Step 1: Ensure the zero of the head scale coincides with a definite division on the main scale.

Step 2: Rotate the screw five times.

Step 3: Observe the distance moved by the main scale.

For example:

Distance moved by main scale = 5mm

No. of rotation = 5

Pitch = Distance moved by main scale/ no. of division

Pitch = 5/5 = 1 mm

Sailesh: That was quite simple!

Least Count

Sailesh has had plenty of time to understand the working of a Micrometer with Mohan's help. Here is how they went about studying it.

Mohan: Does the terminology 'Least Count' ring a bell.

Sailesh: Yes!! We did study about the least count of a Vernier Calliper in our last session.

Mohan: Similarly, we can calculate the smallest distance that a Micrometer can measure accurately with this simple formula. Here take a look.

Least count = Pitch of Screw Gauge/ No. of head scale divisions

For Example:

Pitch = 1 mm

No. of head scale division = 100

Least count = 1/100= 0.01mm

Remember, the least count is the smallest distance that a Micrometer can measure accurately.

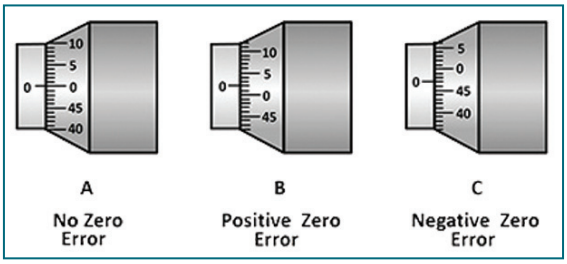
Sailesh: So Mohan, does this mean that a Screw Gauge can also have a zero error.

Mohan: Yes Sailesh!! Your assumption is just right! Just like you can measure the zero error for a Vernier Calliper you can measure the zero error for a Screw Gauge.

Zero Error

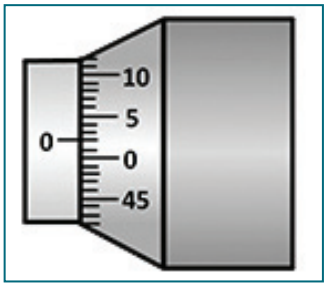
So remember to thoroughly check the Micrometer for the zero error prior to its use. If on bringing the flat end of the screw in contact with the stud, the zero mark of the circular scale does not coincide with the zero mark on base line of the main scale, the instrument is said to have zero error. This can be both positive and negative zero error.

- Positive Zero Error – when zero of the head scale is below the reference line of the main scale
- Negative Zero Error - when zero of the head scale is above the reference line of the main scale



Sailesh: How do you calculate a zero error in Micrometer?

Mohan: When a Screw Gauge has a positive zero error you should pursue the following given steps:



Positive Zero Error

Step 1: Note the division on the head scale coinciding with the zero of the main scale.

For example: The 1st division of the head scale is coinciding with the reference line of the main scale. As the zero of the head scale is below the reference line, this is a positive zero error.

Step 2: Multiply the coinciding head scale division with the least count.

Zero Error = Coinciding Head Scale x Least count

Zero Error = 1 x 0.01 = + 0.01mm

Zero Correction = - 0.01mm

Step 3: Add the zero correction with the observed reading to obtain the actual reading.

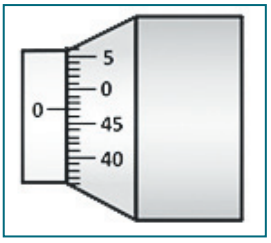
Actual reading = Observed reading + zero correction (-0.01mm)

Sailesh: I see. What about the negative zero error?

Mohan: In order to calculate the negative zero error follow the steps given below:

Step 1: Note the division on the head scale coinciding with the zero of the main scale.

For example: The 98th division of the head scale is coinciding with the reference line of the main scale. As the zero of the head scale is below the reference line, this is negative zero error.



Negative Zero Error

Step 2: Subtract the coinciding division with the total number of divisions on the head scale.

Total No. of divisions = 100

Coinciding division = 98

Subtract 98 from 100 gives us 2 as balance.

Step 3: Multiply the coinciding head scale division which is 2 with the least count.

Zero Error = Coinciding Head Scale x Least count

Zero Error = 2 x 0.01 = 0.02mm

Step 4: Assign a negative sign to the zero error as it is negative zero error.

Zero Error = 2 x 0.01 = - 0.02mm

Zero Correction = +0.02mm

Step 5: Add the zero correction with the observed reading to obtain the actual reading.

Actual reading = Observed reading + zero correction (0.02mm).

Sailesh: This is a bit confusing. I will recapitulate these steps before I use a Screw Gauge to ensure accuracy.

Determination of Diameter of a Wire

Sailesh: Mohan, wouldn't be exciting to measure the diameter of a wire using a Micrometer?

Mohan: Yes Sailesh... That's what's wonderful about a Micrometer. It helps you to measure even small dimensions like that of a wire with great accuracy. All you need to is follow these steps:

Step 1: Calculate the least count.

Least count = Pitch of Screw Gauge/ No. of head scale divisions

Let least count be 0.01mm.

Step 2: Calculate the zero error.

Zero Error = Coinciding Head Scale x Least count

Step 3: Place the given wire in between the stud and the flat end.

Step 4: Rotate the screw till the wire is firmly gripped by the both ends.

Step 5: Note the main scale reading and the head scale reading.

Step 6: Calculate the diameter of the wire with the given formula.

Actual Diameter= Observed Diameter + Zero Correction

Step 7: Repeat the above steps three to five times and take the average of the obtained outcomes. This will be the actual diameter.

You can use the following table to tabulate your findings.

Sl. No.	Main Scale Reading (MSR)	Circular Scale Reading (CSR)	Observed diameter (mm) = MSR + (CSR x Least Count)	Actual Diameter= Observed Diameter + Zero Correction
1.				
2.				
3.				
4.				

Sailesh: Well friends, today I have learned great deal about Micrometers. However, practice makes perfect! So, off I go to measure some thin wires!

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. Write a detailed note on Pitch and Least Count.

2. What is zero error? How can you calculate zero error in Micrometer?

Worksheet

1. *Determine whether Sailesh has his facts right or wrong. Write 'True' against the statements that hold true and 'False' against the ones that are wrong.*

   a. Micrometer is also called Micrometer Screw Gauge.

   b. Micrometer helps you to measure thickness of a wire or a metal sheet.

   c. Positive zero error is when zero of the head scale is above the reference line of main scale.

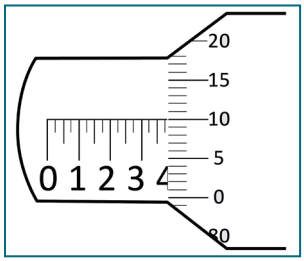
   d. Anvil is also called Stud.

   e. Spindle is a metal block which can move forward and backwards.

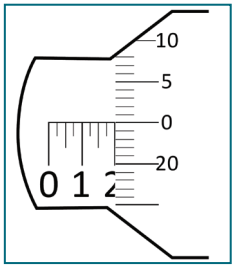
2. *Sailesh wants to calculate the pitch of a Micrometer but seems to have forgotten the formula. Help him by giving the correct formulae of pitch in Micrometer.*

3. *Sailesh has spent the last half an hour measuring various items with a Micrometer. Given that least count is 0.01 mm and Micrometer is free of zero error, help Sailesh calculate the actual diameter of the items based on the readings shown below:*

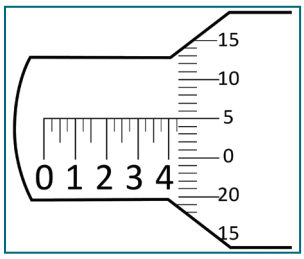
a.



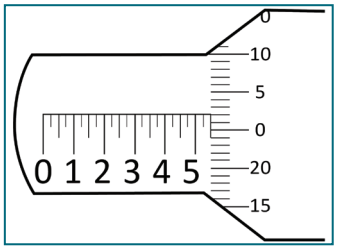
b.



c.



d.



Notes

Answers:

1.

a. True

b. True

c. False

d. True

e. True

2. Pitch = Distance moved by main scale/ No. of division.

3.

a. 3.85mm

b. 2mm

c. 4.3mm

d. 5.52mm

Precision measurement other instruments

At the end of this session, you will be able to:

- use the different measuring instruments.

Session Plan	
1	Interchangeability and Tolerance
2	Types of Gauges
3	Dial Gauge
4	Height Gauge
5	Internal Micrometer
6	Key Learning
7	Worksheet

Interchangeability and Tolerance

Sailesh has come across the term 'Interchangeability and Tolerance'. He wonders if it is linked with lathe machine or to him as an operator, he asks Mohan about it:

Sailesh: Mohan, my friend told me about a term named interchangeability and tolerance. I want to know about it in detail?

Mohan: Sure, Sailesh. Let me first tell you what is Interchangeability and tolerance.

Well interchangeability as the term explains itself is something or part that can be interchanged or replaced with a new one. And tolerance too as the word suggests is the maximum allowed limit to which a dimension can be stretched.

Did you understand anything?

Sailesh: Yes, to some extent... I would appreciate if you could elaborate with an example.

Mohan: Sure, I would like to say that there is nothing in the universe which has absolute meaning without tolerance. For example, even the atomic clock which is supposed to be the most accurate clock of the world has a tolerance. However small it may be.

Sailesh: how is it linked with lathe machine?

Mohan: Well, I will explain it. In turning operations, machine tool quality, quality of material, RPM of the machine, feed of the tool, the ambient temperature of the room all play a role to contribute towards the final dimension of a turned piece. Thus all the dimensions of all pieces turned may not always be the same.

Sailesh: why do we specify a tolerance?

Mohan: Very good question, Sailesh! It is because within a tolerable limit; variation in a dimension is acceptable because it falls within the parameter of interchangeability.

Type of Gauges

Sailesh is holding a vernier caliper in his hand and thinking about other instruments that can be used for measurements. On seeing Mohan, he asks him about the same.

Sailesh: Mohan, like vernier caliper are there some more instruments that help in taking measurements?

Mohan: Yes Sailesh, in machining, there are various measurements that can be done by standard measuring tool like micrometer, height gauge etc. For the purpose of consistency in manufactured output and ensuring the output within tolerances, certain methods of controlling the same are required.

Sailesh: What are various types of measuring instruments or gauges?

Mohan: Well, there are three types of measurement control devices which are categorized into three basic categories:

1. Movable- The movable ones are vernier calipers, internal and external micrometers, height gauge, dial gauges, angle protectors.
2. Fixed- The fixed ones include Go-no go gauges, snap gauges, slip gauges, thread gauge, tri-square gauge.
3. Contours- The contours refer to special shapes of any types primarily for non standard special items.

Sailesh: Can you show me all these instruments in our workshop?

Mohan: Yes, why not! I will show you all the measuring instruments in the evening.

Sailesh: Thank you!

Dial Gauge

After seeing the different type of gauges and other instruments kept in workshop. Sailesh wants to know how they work.

Sailesh: Mohan, what is this circle kind of instrument?

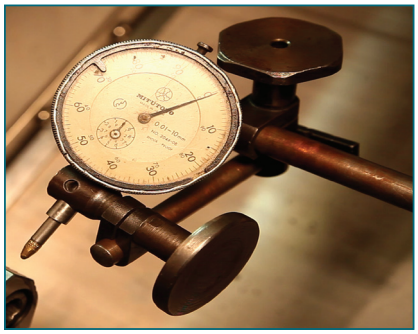
Mohan: This is a dial gauge. The Dial Gauges are very fine measuring instruments for gauging the variance. These are single point touch ball mounted on the nose of a dial gauge, which when pushed or pulled shows deflection of the needle, which can be measured.

Sailesh: So, can we see the measurement in positive deflection or both?

Mohan: The dial gauge will show both positive and negative deflection; moreover the deflection can be quantified and measured. The least count of these devices can be as low as one micron millimeter.

Dial Gauge

Dial Gauge



Puppy Dial Gauge

Sailesh: How do you take measurements with a dial gauge?

Mohan: The Dial gauge is normally held like this on a fixed plane with dial gauge ball free to touch the surface for measuring the variance.

Sailesh: Is this also a kind of dial gauge?

Mohan: Yes, this is known as Puppy Dial Gauge which is much smaller than the normal dial gauge and because of its smaller size and lighter weight it is used more often and specially in applications where there is less space to work.

Sailesh: We can even call it a concise dial gauge.

Height Gauge

Sailesh: Do we have a special instrument to measure the height of something or do we do it with the help of a dial gauge only?

Mohan: There is a special instrument called height gauge that is used either to determine the height of something, or for repetitious marking of items to be worked on.

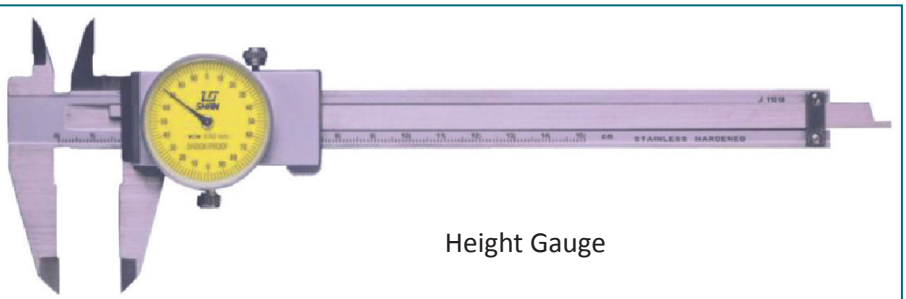
Sailesh: Where is this measuring tool used?

Mohan: This measuring tool is used in metal working or metrology to either set or measure vertical distances; the pointer is sharpened to allow it to act as a scriber and assist in marking out work pieces.

Sailesh: OK, so what are the features of a dial gauge?

Mohan: Let me tell you about its construction a little…

It has simple construction which consists of a base which remains in contact with surface plate. The main scale is inserted and fixed on the base. This main scale also serves as rail on which vernier scale assembly slides up and down. There is a locking screw with which we can lock position of vernier scale assembly with respect to main scale. Above it there is a fine adjustment screw which we can adjust zero of vernier scale while assembly is locked by lock no-1. You can adjust height of probe while second part is fixed by lock -2.



Height Gauge

Sailesh: Can you tell me how it works, as I am still confused……

Mohan: Sure, I'll tell you. The height gauge can be used to measure the height of an object by using the underside of the scriber as the datum. The datum may be permanently fixed or the height gauge may have provision to adjust the scale, this is done by sliding the scale vertically along the body of the height gauge by turning a fine feed screw at the top of the gauge; then with the scriber set to the same level as the base, the scale can be matched to it. This adjustment allows different scribers or probes to be used, as well as adjusting for any errors in a damaged or re-sharpened probe.

Sailesh: How to can we take measurements with height gauge?

Mohan: Taking measurements with height gauge is very simple. First, clean the surface plate and the probe gently with cotton. Then set zero of main scale and vernier scale so that the division of zero coincide with each other. Then move the vernier caliper toward the work piece. Let the probe come down and sit on the work piece. Then take the reading from the main scale and the vernier scale.

Close view for zero setting

Probe is placed on the work piece top

Show reading of the vernier and main scale.

Sailesh: It looks easy to work with.

Mohan: Yes, but it's not that easy, as you need to take care about the marking too. In its simplest working, clean the surface table and the probe gently with cotton. Then set zero of main scale and vernier scale so that the division of zero coincide with each other. Then move the vernier scale assembly upward to a desired height and lock the position. Then move the vernier caliper toward he work piece and move base in such a manner that the probe touches the portion where which needs to marked. The sharp edge will scratch on the work piece which is required.

Image required for H.G. probe on surface plate level.

Close view for zero setting

Probe is lifted up to desired height.

Giving mark on the work piece.

Sailesh: The process is quite lengthy and needs careful working!

Mohan: You are right Sailesh! Hope you have understood everything?

Sailesh: Yes, I got it. Thank you.

Internal Micrometer

Sailesh: Mohan I have learnt how to take measurement with dial gauge and height gauge, now I want to learn about micrometer.

Mohan: Sure, I will tell you how the internal micrometer works but let me first tell you what an internal micrometer is.

Internal micrometer as the name suggests, is used to measure holes and bores only of precise size, the micrometers are available in steps say 25- 30mm, 30-40mm and so on.

This is also an internal micrometer and can measure other geometry other than holes and bores. They can even accurately measure gaps between two planes, inside dimension of a square and a slot.

Sailesh: That's great.

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. What is Interchangeability and Tolerance?

2. What is a Dial gauge? Explain in detail.

3. What is a Height gauge? Explain in detail.

Worksheet

1. *Mohan asks Sailesh and his friends a few rapid fire questions, which need to be answered in one or two words. See if you can help them answer the questions.*

    1. For what purposes is internal micrometer used?
    2. Can you measure gap between two planes, inside dimension of a square, slot accurately with internal micrometer? Say, Yes or no.
    3. The height gauge is used to measure height of something or for repetitious marking of items to be worked on. Is this statement true or false?
    4. Name the dial gauge which is smaller in size and lighter in weight than normal a dial gauge?
    5. Name the instrument that is helpful to take measurements accurately and that has a single point touch ball mounted on the nose of gauge?

Notes

Answers:

1.

1. To measure holes and bores only of precise size.
2. Yes.
3. True
4. Puppy Dial Gauge.
5. Dial Gauge.

Precision Measurement- Gauges

At the end of this session, you will be able to:

- use the different types of fixed gauges.

Session Plan	
1	Gauges and Types of Gauges
2	Angle Gauge
3	Thread Gauge
4	Tri-Square Gauge
5	Key Learning
6	Worksheet

Gauges and Types of Gauges

Sailesh wants to learn about measuring instruments apart from the vernier calliper that can help you in getting accurate readings. He asks his co-worker Narinder. He too, does not know much about the measuring instruments and so they decide to ask Mohan:

Sailesh: I know that the vernier calliper is one of the most reliable measuring instruments; is there any other measuring instruments that can be considered reliable?

Mohan: Yes, gauges are equally reliable.

Sailesh: Gauge! What is that?

Mohan: Well, in simple words; gauge is a measuring instrument that helps you to take accurate measurements and readings.

Sailesh: Are there different types of gauges?

Mohan: Yes Sailesh, Gauges can be categorized into three basic categories:

1. Movable- The movable ones are vernier calipers, internal and external micrometers, height gauge, dial gauges, angle protectors.
2. Fixed- The fixed ones include Go-no go gauges, snap gauges, slip gauges, thread gauge, tri-square gauge and angle gauges.
3. Contours- The contours refer to special shapes of any types primarily for non standard special items.

Sailesh: Oh! That means there are so many options available to for us to take accurate measurements...

Mohan: Yes, you are right Sailesh. Now I will tell you about some important gauges in just a while. Are you ready?

Sailesh: Yes why not!

Angle Gauge

Mohan gets a book that has detailed information about Gauges. Excited Sailesh asks Mohan:

Mohan: I have brought this book written by Mr. Batra who is a scholar on lathe machine. Let me start with the angle gauge.

Sailesh: Angle gauge is a type of fixed gauges, right?

Mohan: Absolutely right! Angle Gauge is also known as Angle Protector. It is an instrument to gauge the angle, it is different from measuring the angle, as this a process of comparison or matching the angle, it is termed as Gauging.

The angle gauges have the following basic shapes / types:

1. Used normally for External Angle Gauging however depends upon the application, the use can be accordingly modified.
2. Used normally for Internal Angle Gauging however depends upon the application, its usage can be accordingly modified.

Thread Gauge

Mohan: Now, I will tell you about another kind of fixed gauge. It is called Thread Gauge. Are you ready?

Sailesh: Yes, I am ready.

Mohan: well, the threads are of various types and pitches, depending on the use. However there are some commonly used thread types with standard pitch, the thread gauges are used to gauge the same.

Sailesh: Are thread gauges of two types; external and internal thread gauges?

Mohan: That's right, Sailesh! Do you know about threads?

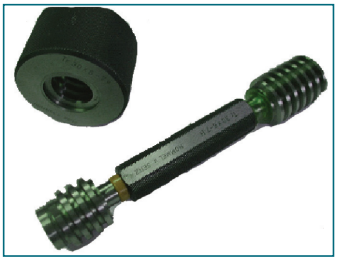
Sailesh: No, I just guessed.

Mohan: Well, let me explain about both kinds of threads one by one:

External Thread Gauge

1. **External Thread Gauges**: The External Thread Gauges are used to compare the pitch and the shape of the thread with respect to the Specified Thread Gauge.

2. **Internal Thread Gauge**: In case of Internal Thread Gauge, there are Go and No-Go thread gauges of that particular thread as required with the specific Type, Dia and Pitch for the thread.



Internal Thread Gauge

Tri-square Gauge

Mohan: Now, I will tell you in brief about the Tri-square gauge.

See this is how a Tri-square gauge looks like:

Sailesh: Oh! It looks like a L-shaped Scale.

Mohan: Yes, that's right! It looks like L- Shaped scale. It is used normally to gauge 90 degree angle both internal and external.

Sailesh: That's great! This was an interesting session. I wish to see all these types of Gauges in real life. While, I go with Mohan to see why don't you try to do the same!!

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. What is a gauge? Name the different types of gauges.

2. Explain about the Angle gauge.

3. Write about the Thread gauge.

Worksheet

1. *Sailesh is talking about gauges. Fill in the blanks to complete the statements:*
   a. _____ is an L-shaped scale.
   b. Tri-square scale can be used to gauge 90 degree _____ (both internal and external) angle.
   c. Go and No go are type of _____ gauge.
   d. The external thread gauge is used to compare the _____ and the _____ of the thread with respect to the specified thread gauge.
   e. Thread gauge are of two types _____ and _____.
2. *Name any 5 types of fixed gauges?*
3. *Name any 5 types of movable gauges?*

Notes

Answers:

1.
   a. Tri-square
   b. Both internal and external.
   c. Internal
   d. Pitch and shape
   e. External and internal.
2. *Go-No go gauges, snap gauge, slip gauge, and thread gauge, Tri-square gauge and angle gauge.*
3. *Vernier calliper, internal and external gauge, height gauge, dial gauge and angle protector.*

Understanding Materials

At the end of this session, you will be able to:

- state the basic properties of materials(hardness, ductility, toughness).

Session Plan	
1	Classification of Engineering Materials
2	Properties of Metals
3	Cast Iron
4	Wrought Iron
5	Steel
6	Alloy Steel
7	Stainless Steel
8	Non- Ferrous Metals
9	Bearing Metals
10	Key Learnings
11	Worksheet

Classification of Engineering Materials

Today, Mohan is taking a special session for all the workers to give to tell them about the classification of materials.

Mohan: It is important for operators to know about understanding materials. Being operators you must be familiar with the effects which the manufacturing processes and heat treatment have on the properties of the materials. We need to know about the basic properties of materials that we use commonly as engineering materials and their limitations in machining operation.

Classification of Engineering Materials

The engineering materials are mainly classified as:

1. Metals and their alloys such as; iron, steel, copper, aluminium, etc.
2. Non-metals such as; glass, rubber, plastic, etc.

The metals may be further classified as:

a. Ferrous metals

b. Non-ferrous metals

The ferrous metals are those which have the iron as their main constituent, such as cast iron, wrought iron and steel.

The non-ferrous metals are those which have a metal other than iron as their main constituent, such as copper, aluminum, brass, tin, zinc, etc.

Mohan: Is it making sense to you?

Sailesh: Yes, it seems easy to understand and interesting too.

Properties of Metals

Mohan: Now let us understand the properties of metals.

Sailesh: Properties of metal! What does that mean?

Mohan: Properties of metal are nothing but characteristics of metal. The properties of metal are divided in two parts:

1. Physical properties of metal
2. Mechanical properties of metal

Let's discuss them in detail now:

1. Physical properties of metal

The **physical properties of the metals** include luster, colour, size and shape, density, electric and thermal conductivity, and melting point.

Mohan: I have made a table that shows the important physical properties of some pure metals.

Physical properties of metals.				
Metal	**Metal Density In (kg/m3)**	**Melting point in (°C)**	**Thermal conductivity (W/m°C)**	**Coefficient of linear expansion at 20°C (µm/m/°C)**
Aluminium	2700	660	220	23
Brass	8450	950	130	16.7
Bronze	8730	1040	67	17.3
Cast iron	7250	1300	54.5	9
Copper	8900	1083	393.5	16.7
Lead	11400	327	33.5	29.1
Monel metal	8600	1350	25.2	14
Nickel	8900	1453	63.2	12.8
Silver	10500	960	420	18.9
Steel	7850	1510	50.2	11.1
Tin	7400	232	67	21.4
Tungsten	19300	3410	201	4.5
Zinc	7200	419	113	33
Cobalt	8850	1490	69.2	12.4
Molybde-num	10200	2650	13	4.8
Vanadium	6000	1750	—	7.75

Sailesh: Now, can you tell us about the mechanical properties of metal?

Mohan: Yes, Sailesh.

The mechanical properties of the metals are those which are associated with the ability of the material to resist mechanical forces and load. These mechanical properties of the metal include strength, stiffness, elasticity, plasticity, ductility, brittleness, malleability, toughness, resilience, creep and hardness. We shall now discuss these properties as follows:

- **Strength**: - It is the ability of a material to resist the externally applied forces without breaking or yielding. The internal resistance offered by a part to an externally applied force is called stress.
- **Stiffness**: - It is the ability of a material to resist deformation under stress. The modulus of elasticity is the measure of stiffness.
- **Elasticity**: - It is the property of a material to regain its original shape after deformation when the external forces are removed. This property is desirable for materials used in tools and may be noted that steel is more elastic than rubber permanently. This property of the material is necessary for forgings, in stamping images on coins and in ornamental work.
- **Ductility**: - It is the property of a material enabling it to be drawn into wire with the application of a tensile force. A ductile material must be both strong and plastic. The ductility is usually measured by the terms, percentage elongation and percentage reduction in area. The ductile material commonly used in engineering practice (in order of diminishing ductility) are mild steel, copper, aluminum, nickel, zinc, tin and lead. Note: The ductility of a material is commonly measured by means of percentage elongation and percentage reduction in area in a tensile test
- **Brittleness**: - It is the property of a material opposite to ductility. It is the property of breaking of a material with little permanent distortion. Brittle materials when subjected to tensile loads snap off without giving any sensible elongation. Cast iron is brittle.
- **Malleability**: - It is a special case of ductility which permits materials to be rolled or hammered into thin sheets. A malleable material should be plastic but may not be so strong. The malleable materials commonly used in engineering practice (in order of diminishing malleability) are lead, soft steel, wrought iron, copper and aluminum.
- **Toughness**: - It is the property of a material to resist fracture due to high impact loads like hammer blows. The toughness of the material decreases when it is heated. It is measured by the amount of energy that a unit volume of the material has absorbed after being stressed up to the point of fracture. This property is desirable in parts subjected to shock and impact loads.
- **Machinability**: - It is the property of a material which refers to a relative case with which a material can be cut. The machinability of a material can be measured in a number of ways such as comparing the tool life for cutting different materials or thrust required to remove the material at some given rate or the energy required to remove a unit volume of the material. It may be noted that brass can be easily machined than ordinary steel.
- **Resilience**: - It is the property of a material to absorb energy and to resist shock and impact loads. It is measured by the amount of energy absorbed per unit volume within elastic limit. This property is essential for spring materials.
- **Creep**: - When a part is subjected to a constant stress at high temperature for a long period of time, it will undergo a slow and permanent deformation called creep. This property is considered in designing internal combustion engines, boilers and turbines.
- **Fatigue**: - When a material is subjected to repeated stress; it fails at stresses below the yield point stresses. Such type of failure of a material is known as fatigue. The failure is caused by progressive crack formation which are usually fine and of microscopic size. This property is considered in designing shafts, connecting rods, springs, gears, etc.

- **Hardness**: - It is a very important property of the metals and has a wide variety of meanings. It embraces many different properties such as resistance to wear, scratching, deformation and machinability etc. It also means the ability of a metal to cut another metal. The hardness is usually expressed in numbers which are dependent on the method of making the test.

Sailesh: Is it necessary to memorize this?

Mohan: No, you don't need to but yes you should know about it thoroughly.

Cast Iron

Mohan: Now, I will teach you about cast Iron.

Sailesh: What is cast iron?

Mohan: The cast iron is obtained by re-melting pig iron with coke and limestone in a furnace known as cupola. It is primarily an alloy of iron and carbon.

Sailesh: How much carbon content is present in this alloy?

Mohan: The carbon content in cast iron varies from 1.7 percent to 4.5 percent. It also contains small amount of silicon, manganese, phosphorous and sulphur. The carbon in a cast iron is present in either of the following two forms:

1. Free carbon or graphite, and
2. Combined carbon or cementite.

Sailesh: If cast iron is hard like iron then how is it used?

Mohan: No, Sailesh the cast iron is a brittle material so it cannot be used in those parts of machines which are subjected to shocks.

Sailesh: What makes it a valuable material for engineering?

Mohan: The properties of cast iron which make it a valuable material for engineering purposes are its low cost, good casting characteristics, high compressive strength, wear resistance and excellent machinability. The compressive strength of cast iron is much greater than the tensile strength.

Sailesh: Are there some fixed values of cast iron?

Mohan: The values of the ultimate strength of cast iron are:

Tensile strength= 100 to 200 MPa

Compressive strength = 400 to 1000 MPa

Shear strength= 120 MPa

Wrought Iron

Mohan: Let me now tell you about wrought iron

Sailesh: I know that the wrought iron is considered the purest form of iron.

Mohan: You are right Sailesh, but do you know its exact composition?

Sailesh: I just know that it has very high composition of iron in it but I don't know its values.

Mohan: Well, wrought iron is purest as it contains at least 99.5% iron but may contain upto 99.9% iron. The typical composition is Carbon = 0.020%, Silicon = 0.120%, Sulphur = 0.018%, Phosphorus = 0.020%, Slag = 0.070%, and the remaining is iron.

Sailesh: How is wrought iron made?

Mohan: The wrought iron is produced from pig iron by re-melting it in the puddling furnace of reverberator type. The molten metal free from impurities is removed from the furnace as a pasty mass of iron and slag. The balls of this pasty mass, each about 45 to 65 kg are formed. These balls are then mechanically worked both to squeeze out the slag and to form it into some commercial shape.

Sailesh: Is it a soft metal? What is its strength value?

Mohan: No, Sailesh, wrought iron is not a soft metal; in fact it is a tough, malleable and ductile material. It cannot stand sudden and excessive shocks. Its ultimate tensile strength is 250 MPa to 500 MPa and the ultimate compressive strength is 300 MPa. It can be easily forged or welded. It is used for chains, crane hooks, railway couplings, and water and steam pipes.

Steel

Mohan: Now, I will tell you about steel.

Sailesh: I just know that steel is the most commonly used metal. I don't know anything about its composition.

Mohan: Steel is an alloy of iron and carbon, with carbon content up to a maximum of 1.5%. The carbon occurs in the form of iron carbide, because of its ability to increase the hardness and strength of the steel. Other elements e.g. silicon, sulphur, phosphorus and manganese are also present to greater or lesser amount to impart certain desired properties to it. Most of the steel produced now-a-days is plain carbon steel or simply carbon steel.

Sailesh: How can we define carbon steel?

Mohan: Carbon steel is defined as steel which has its properties mainly due to its carbon content and does not contain more than 0.5% of silicon and 1.5% of manganese.

Sailesh: what are various types of steel depending upon the carbon content in it?

Mohan: The plain carbon steels varying from 0.06% carbon to 1.5% carbon are divided into the following types depending upon the carbon content.

1. Dead mild steel— up to 0.15% carbon
2. Low carbon or mild steel— 0.15% to 0.45% carbon
3. Medium carbon steel— 0.45% to 0.8% carbon
4. High carbon steel— 0.8% to 1.5% carbon

The steel is specified on the basis of minimum tensile strength or yield strength.

Sailesh: Are there some impurities in steel and what effects do these impurities have on steel?

Mohan: Yes, Sailesh, steel has a lot of impurities and following are the effects of impurities like Silicon, sulphur, manganese and phosphorus on steel.

- **Silicon**: The amount of silicon in the finished steel usually ranges from 0.05 to 0.30%. Silicon is added in low carbon steels to prevent them from becoming porous. It removes the gases and oxides, prevent blow holes and thereby makes the steel tougher and harder.
- **Sulphur**: It occurs in steel either as iron sulphide or manganese sulphide. Iron sulphide because of its low melting point produces red shortness, whereas manganese sulphide does not affect so much. Therefore, manganese sulphide is less objectionable in steel than iron sulphide.
- **Manganese**: It serves as a valuable deoxidizing and purifying agent in steel. Manganese also combines with sulphur and thereby decreases the harmful effect of this element remaining in the steel. When used in ordinary low carbon steels, manganese makes the metal ductile and of good bending qualities. In high speed steels, it is used to toughen the metal and to increase its critical temperature.
- **Phosphorus**: It makes the steel brittle. It also produces cold shortness in steel. In low carbon steels, it raises the yield point and improves the resistance to atmospheric corrosion. The sum of carbon and phosphorus usually does not exceed 0.25 %.

Sailesh: I have heard about free cutting steels; how is it made?

Mohan: The free cutting steels contain sulphur and phosphorous. These steels have higher sulphur content than other carbon steels. In general, the carbon content of such steels vary from 0.1 to 0.45 per cent and sulphur from 0.08 to 0.3 per cent. These steels are used where rapid machining is the prime requirement. It may be noted that the presence of sulphur and phosphorus causes long chips in machining to be easily broken and thus prevent clogging of machines. Now a day, lead is used from 0.05 to 0.2 per cent instead of sulphur, because lead also greatly improves the machinability of steel without the loss of toughness.

Alloy Steel

Mohan: Do you know anything about alloy steel? Ok, tell me in your own words what is alloy steel?

Sailesh: Alloy steel according to me can be steel which has elements other than carbon are added in it.

Mohan: Yes Sailesh, you are right to some extent. Let me tell you clearly. alloy steel may be defined as a steel to which elements other than carbon are added in sufficient amount to produce an improvement in properties. The alloying is done for specific purposes to increase wearing resistance, corrosion resistance and to improve electrical and magnetic properties, which cannot be obtained in plain carbon steels

Sailesh: what are the alloying elements in it?

Mohan: The chief alloying elements used in steel are nickel, chromium, molybdenum, cobalt, vanadium, manganese, silicon and tungsten. Each of these elements add certain qualities upon the steel to which it is added. These elements may be used separately or in combination to produce the desired characteristic in steel.

Sailesh: What are the possible effects of alloying elements in steel?

Mohan: Following are the effects of alloying elements on steel:

- **Nickel**: It increases the strength and toughness of the steel. These steels contain 2 to 5% nickel and from 0.1 to 0.5% carbon. In this range, nickel contributes great strength and hardness with high elastic limit, good ductility and good resistance to corrosion. An alloy containing 25% nickel possesses maximum toughness and offers the greatest resistance to rusting, corrosion and burning at high temperature. It has proved to be of advantage in the manufacture of boiler tubes, valves for use with superheated steam, valves for I.C. engines and spark plugs for petrol engines.

A nickel steel alloy containing 36% of nickel is known as invar. It has nearly zero coefficient of expansion. So it is in great demand for measuring instruments and standards of lengths for everyday use.

- **Chromium**: It is used in steels as an alloying element to combine hardness with high strength and high elastic limit. It also imparts corrosion-resisting properties to steel. The most common chrome steels contains from 0.5 to 2% chromium and 0.1 to 1.5% carbon. The chrome steel is used for balls, rollers and races for bearings. A nickel chrome steel containing 3.25% nickel, 1.5% chromium and 0.25% carbon is used for armour plates. Chrome nickel steel is extensively used for motor car crankshafts, axles and gears requiring great strength and hardness.
- **Tungsten**: It prohibits grain growth, increases the depth of hardening of quenched steel and confers the property of remaining hard even when heated to red colour. It is usually used in combination with other elements. Steel containing 3 to 18% tungsten and 0.2 to 1.5% carbon is used for cutting tools. The principal uses of tungsten steels are for cutting tools, dies, valves, taps and permanent magnets.
- **Vanadium**: It aids in obtaining a fine grain structure in tool steel. The addition of a very small amount of vanadium (less than 0.2%) produces a marked increase in tensile strength and elastic limit in low and medium carbon steels without a loss of ductility. The chrome-vanadium steel containing about 0.5 to 1.5% chromium, 0.15 to 0.3% vanadium and 0.13 to 1.1% carbon have extremely good tensile strength, elastic limit, endurance limit and ductility. These steels are frequently used for parts such as springs, shaft, gears, pins and many drop forged parts.
- **Manganese**: It improves the strength of the steel in both the hot rolled and heat treated condition. The manganese alloy steels containing over1.5% manganese with a carbon range of 0.40 to 0.55% are used extensively in gears, axles, shafts and other parts where high strength combined with fair ductility is required. The principal use of manganese steel is in machinery parts subjected to severe wear. These steels are all cast and ground to finish.
- **Silicon**: The silicon steels behave like nickel steels. These steels have a high elastic limit as compared to ordinary carbon steel. Silicon steels containing from 1 to 2% silicon and 0.1 to 0.4% carbon and other alloying elements are used for electrical machinery, valves in I.C. engines, springs and corrosion resisting materials.
- **Cobalt**: It gives red hardness by retention of hard carbides at high temperatures. It tends to decarburise steel during heat-treatment. It increases hardness and strength and also residual magnetism and coercive magnetic force in steel for magnets.
- **Molybdenum**: A very small quantity (0.15 to 0.30%) of molybdenum is generally used with chromium and manganese (0.5 to 0.8%) to make molybdenum steel. These steels possess extra tensile strength and are used for air-plane fuselage and automobile parts. It can replace tungsten in high speed.

Stainless Steel

Mohan: what do you know about stainless steel?

Sailesh: Stainless Steel is among the most commonly used metal. It is commonly defined as that steel which when correctly heat treated and finished, resists oxidation and corrosive attack from most corrosive media.

Mohan: Very good, Sailesh! It's absolutely right.

Sailesh: But I don't know about the different types of steel.

Mohan: Don't worry; I am now going to tell you about its types. The various types of stainless steel are:

- **Martensitic Stainless Steel**: The chromium steels containing 12 to 14 per cent chromium and 0.12 to 0.35 per cent carbon are the first stainless steels developed. Since these steels possess martensitic structure, therefore, they are called martensitic stainless steels. These steels are magnetic and may be hardened by suitable heat treatment and the hardness obtainable depends upon the carbon content.

  These steels can be easily welded and machined. When formability, softness, etc. are required in fabrication, steel having 0.12 per cent maximum carbon is often used in soft condition. With increasing carbon, it is possible by hardening and tempering to obtain tensile strength in the range of 600 to 900 N/mm2, combined with reasonable toughness and ductility. In this condition, these steels find many useful general applications where mild corrosion resistance is required. Also, with the higher carbon range in the hardened and lightly tempered condition, tensile strength of about 1600 N/mm2 may be developed with lowered ductility.

  These steels may be used where the corrosion conditions are not too severe, such as for hydraulic, steam and oil pumps, valves and other engineering components. However, these steels are not suitable for shafts and parts working in contact with non-ferrous metals (i.e. brass, bronze or gun metal bearings) and with graphite packing's, because electrolytic corrosion is likely to occur. After hardening and light tempering, these steels develop good cutting properties. Therefore, they are used for cutlery, springs, surgical and dental instruments.

  Note: The presence of chromium provides good resistance to scaling upto a temperature of about 750°C, but it is not suitable where mechanical strength in the temperature range of 600 to 750°C is required. In fact, creep resistance of these steels at this temperature is not superior to that of mild steel. But at temperature below 600°C, the strength of these steels is better than that of carbon steels and upto 480°C is even better than that of austenitic steels.

- **Ferritic Stainless Steel** The steels containing greater amount of chromium (from 16 to 18 per cent) and about 0.12 per cent carbon are called ferritic stainless steels. These steels have better corrosion resistant property than martensitic stainless steels. But, such steels have little capacity for hardening by heat treatment. However, in the softened condition, they possess good ductility and are mainly used as sheet or strip for cold forming and pressing operations for purposes where moderate corrosion resistance is required. They may be cold worked or hot worked. They are ferro-magnetic, and usually undergo excessive grain growth during prolonged exposure to elevated temperatures. They may develop brittleness after electric arc resistance or gas welding. These steels have lower strength at elevated temperatures than martensitic steels. However, resistance to scaling and corrosion at elevated temperatures are usually better. The machinability is good and they show no tendency to intercrystalline corrosion.

  Note: When nickel from 1.5 to 2.5 per cent is added to 16 to 18 per cent chromium steel, it not only becomes more resistant to corrosion than martensitic steel but also makes it hardenable by heat treatment. Such steel has good resistance to electrolytic corrosion when in contact with non-ferrous metals and graphite packing's. Thus it is widely used for pump shafts, spindles and valves as well as for many other fittings where a good combination of mechanical and corrosion properties are required.

- **Austenitic Stainless Steel** The steel containing high content of both chromium and nickel are called austenitic stainless steels. There are many variations in chemical composition of these steels, but the most widely used steel contain 18 per cent chromium and 8 per cent nickel with carbon content as low as possible. Such steel is commonly known as 18/8 steel. These steels cannot be hardened by quenching; in fact they are softened by rapid cooling from about 1000°C. They are non- magnetic and possess greatest resistance to corrosion and good mechanical properties at elevated temperature.

These steels are very tough and can be forged and rolled but offer great difficulty in machining.

They can be easily welded, but after welding, it is susceptible to corrosive attack in an area adjacent to the weld. This susceptibility to corrosion (called inter crystalline corrosion or weld decay) may be removed by softening after welding by heating to about 1100°C and cooling rapidly. These steels are used in the manufacture of pump shafts, rail road car frames and sheathing, screws, nuts and bolts and small springs. Since 18/8 steel provide excellent resistance to attack by many chemicals, therefore, it is extensively used in chemical, food, paper making and dyeing industries.

Note: When increased corrosion resistance properties are required, for some purposes, then molybdenum from 2 to 3 per cent may be added.

Non-ferrous Metals

Mohan: Now I am going to tell you about Non-ferrous Metals.

Sailesh: I guess non-ferrous metals are those which have some other metal as its chief constituent. Am I right Mohan?

Mohan: Yes, Sailesh you are right. But your information has something missing in it…

Sailesh: And what is that?

Mohan: The non-ferrous metals are those which contain a metal other than iron as their chief constituent. The non-ferrous metals are usually employed in industry due to the following characteristics:

1. Ease of fabrication (casting, rolling, forging, welding and machining),
2. Resistance to corrosion,
3. Electrical and thermal conductivity and
4. Weight

Sailesh: I have heard that there are various other non-ferrous metals that are used in engineering practice. I can name some of them like aluminium, copper, lead, tin, zinc, nickel, etc. and their alloys.

Mohan: You are right Sailesh! Now let me tell you about them in detail:

- Aluminium: It is white metal produced by electrical processes from its oxide (alumina), which is prepared from a clayey mineral called bauxite. It is a light metal having specific gravity 2.7 and melting point 658°C. The tensile strength of the metal varies from 90 MPa to 150 MPa. In its pure state, the metal is weak and soft for most purposes, but when mixed with small amounts of other alloys, it becomes hard and rigid. So, it may be blanked, formed, drawn, turned, cast, forged and die cast. It is a good conductor of electricity and is widely used for overhead cables. The high resistance to corrosion and its non-toxicity makes it a useful metal for cooking utensils under ordinary condition and thin foils are used for wrapping food items. It is extensively used in aircraft and automobile components where it saves weight.

Aluminium Alloys

The aluminium may be alloyed with one or more other elements like copper, magnesium, manganese, silicon and nickel. The addition of small quantities of alloying elements converts the soft and weak metal into hard and strong metal, while still retaining its light weight. The main aluminium alloys are discussed below:

1. **Duralumin**: It is an important and interesting wrought alloy. It is composed of: Copper = 3.5 – 4.5%; Manganese = 0.4 – 0.7%; Magnesium = 0.4 – 0.7%, and the remainder is aluminium. This alloy possesses maximum tensile strength (upto 400 MPa) after heat treatment and age hardening. After working, if the metal is allowed to age for 3 or 4 days, it will be hardened. This phenomenon is known as age hardening.

It is widely used in wrought conditions for forging, stamping, bars, sheets, tubes and rivets. It can be worked in hot condition at a temperature of 500°C. However, after forging and annealing, it can also be cold worked. Due to its high strength and light weight, this alloy may be used in automobile and aircraft components. It is also used in manufacturing connecting rods, bars, rivets, pulleys, etc.

2. **Y-alloy:** It is also called copper-aluminium alloy. The addition of copper to pure aluminium increases its strength and machinability. It is composed of: Copper = 3.5 – 4.5%; Manganese = 1.2 – 1.7%; Nickel = 1.8 – 2.3%; Silicon, Magnesium, Iron = 0.6% each; and the remainder is aluminium.

   This alloy is heat treated and age hardened like duralumin. The ageing process is carried out at room temperature for about five days.

   It is mainly used for cast purposes, but it can also be used for forged components like duralumin. Since Y-alloy has better strength (than duralumin) at high temperature, therefore, it is much used in aircraft engines for cylinder heads and pistons.

3. **Magnalium**: It is made by melting the aluminium with 2 to 10% magnesium in a vacuum and then cooling it in a vacuum or under a pressure of 100 to 200 atmospheres. It also contains about 1.75% copper. Due to its light weight and good mechanical properties, it is mainly used for aircraft and automobile components.

4. **Hindalium**: It is an alloy of aluminium and magnesium with a small quantity of chromium. It is the trade name of aluminium alloy produced by Hindustan Aluminium Corporation Ltd, Renukoot (U.P.). It is produced as a rolled product in 16 gauge, mainly for anodized utensil manufacture.

- **Copper**: It is one of the most widely used non-ferrous metals in the industry. It is a soft, malleable and ductile material with a reddish-brown appearance. Its specific gravity is 8.9 and melting point is 1083°C. The tensile strength varies from 150 MPa to 400 MPa under different conditions. It is a good conductor of electricity. It is largely used in making electric cables and wires for electric machinery and appliances, in electrotyping and electroplating, in making coins and household utensils.

It may be cast, forged, rolled and drawn into wires. It is non-corrosive under ordinary conditions and resists weather very effectively. Copper in the form of tubes is used widely in mechanical engineering. It is also used for making ammunitions. It is used for making useful alloys with tin, zinc, nickel and aluminium.

Copper Alloys

The copper alloys are broadly classified into the following two groups:

Copper-zinc alloys (Brass): The most widely used copper-zinc alloy is brass. There are various types of brasses, depending upon the proportions of copper and zinc. This is fundamentally a binary alloy of copper with zinc each 50%. By adding small quantities of other elements, the properties of brass may be greatly changed. For example, the addition of lead (1 to 2%) improves the machining quality of brass. It has a greater strength than that of copper, but has a lower thermal and electrical conductivity. Brasses are very resistant to atmospheric corrosion and can be easily soldered. They can be easily fabricated by processes like spinning and can also be electroplated with metals like nickel and chromium.

Copper-tin alloys (Bronze): The alloys of copper and tin are usually termed as bronzes. The useful range of composition is 75 to 95% copper and 5 to 25% tin. The metal is comparatively hard, resists surface wear and can be shaped or rolled into wires, rods and sheets very easily. In corrosion resistant properties, bronzes are superior to brasses. Some of the common types of bronzes are as follows:

Phosphor bronze: When bronze contains phosphorus, it is called phosphor bronze. Phosphorus increases the strength, ductility and soundness of castings. The tensile strength of this alloy when cast varies from 215 MPa to 280 MPa but increases upto 2300 MPa when rolled or drawn. This alloy possesses good wearing qualities and high elasticity. The metal is resistant to salt water corrosion. The composition of the metal varies according to whether it is to be forged, wrought or made into castings. A common type of phosphor bronze has the following composition according to Indian standards:

Copper = 87-90%, Tin = 9-10%, and Phosphorus = 0.1-3%. It is used for bearings, worm wheels, gears, nuts for machine lead screws, pump parts, linings and for many other purposes. It is also suitable for making springs.

Silicon bronze: It contains 96% copper, 3% silicon and 1% manganese or zinc. It has good general corrosion resistance of copper combined with higher strength. It can be cast, rolled, stamped, forged and pressed either hot or cold and it can be welded by all the usual methods. It is widely used for boilers, tanks, stoves or where high strength and good corrosion resistance is required.

Beryllium bronze: It is a copper base alloy containing about 97.75% copper and 2.25% beryllium. It has high yield point, high fatigue limit and excellent cold and hot corrosion resistance. It is particularly suitable material for springs, heavy duty electrical switches, cams and bushings. Since the wear resistance of beryllium copper is five times that of phosphor bronze, therefore, it may be used as a bearing metal in place of phosphor has a film forming and a soft lubricating property, which makes it more suitable as bearing metal.

Manganese bronze: It is an alloy of copper, zinc and little percentage of manganese. The Usual composition of this bronze is as follows:

Copper = 60%, Zinc = 35%, and Manganese = 5%

This metal is highly resistant to corrosion. It is harder and stronger than phosphor bronze. It is generally used for bushes, plungers, feed pumps, rods etc. Worm gears are frequently made from this bronze.

Aluminium bronze: It is an alloy of copper and aluminium. The aluminium bronze with 6-8% aluminium has valuable cold working properties. The maximum tensile strength of this alloy is 450 MPa with 11% of aluminium. They are most suitable for making components exposed to severe corrosion conditions. When iron is added to these bronzes, the mechanical properties are improved by refining the grain size and improving Aluminium bronzes are widely used for making gears, propellers, condenser bolts, pump components, tubes, air pumps, slide valves and bushings, etc. Cams and rollers are also made from this alloy. The 6% aluminium alloy has a fine gold colour which is used for imitation jewellery and decorative purposes.

Gun Metal: It is an alloy of copper, tin and zinc. It usually contains 88% copper, 10% tin and 2% zinc. This metal is also known as Admiralty gun metal. The zinc is added to clean the metal and to increase its fluidity.

It is not suitable for being worked in the cold state but may be forged when at about 600°C. The metal is very strong and resistant to corrosion by water and atmosphere. Originally, it was made for casting guns. It is extensively used for casting boiler fittings, bushes, bearings, glands, etc.

Lead: It is a bluish grey metal having specific gravity 11.36 and melting point 326°C. It is so soft that it can be cut with a knife. It has no tenacity. It is extensively used for making solders, as a lining for acid tanks, cisterns, water pipes, and as coating for electrical cables.

The lead base alloys are employed where a cheap and corrosion resistant material is required. An alloy containing 83% lead, 15% antimony, 1.5% tin and 0.5% copper is used for large bearings subjected to light service.

Tin: It is brightly shining white metal. It is soft, malleable and ductile. It can be rolled into very thin sheets. It is used for making important alloys, fine solder, as a protective coating for iron and steel sheets and for making tin foil used as moisture proof packing.

A tin base alloy containing 88% tin, 8% antimony and 4% copper is called babbit metal. It is a soft material with a low coefficient of friction and has little strength. It is the most common bearing metal used with cast iron boxes where the bearings are subjected to high pressure and load.

Note: Those alloys in which lead and tin are predominating are designated as white metal bearing alloys. This alloy is used for lining bearings subjected to high speeds like the bearings of aero-engines.

Bearing Metals

Mohan: Now, I am going to tell you about Bearing Metals. The most commonly used bearing metals are:

1. Copper-base alloys,
2. Lead-base alloys,
3. Zinc base alloys, and
4. Nickel-base alloys.

Now, let us learn about them in detail:

1. **Copper-base alloys**: The copper base alloys are the most important bearing alloys. These alloys are harder and stronger than the white metals (lead base and tin base alloys) and are used for bearings subjected to heavy pressures. These include brasses and bronzes.
2. **Cadmium- base alloys**: These alloys contain 95% cadmium and 5% silver. It is used for medium loaded bearings subjected to high temperature. The selection of a particular type of bearing metal depends upon the conditions under which it is to be used. It involves factors relating to bearing pressures, rubbing speeds, temperatures, lubrication, etc. A bearing material should have the following properties:
    a. It should have low coefficient of friction.
    b. It should have good wearing qualities.
    c. It should have ability to withstand bearing pressures.
    d. It should have ability to operate satisfactorily with suitable lubrication means at the maximum rubbing speeds.
    e. It should have a sufficient melting point.
    f. It should have high thermal conductivity.
    g. It should have good casting qualities.
    h. It should have minimum shrinkage after casting.
    i. It should have non-corrosive properties.
    j. It should be economical in cost.
3. **Zinc Base Alloys**: Most of the die castings are produced from zinc base alloys. These alloys can be with a good finish at fairly low temperatures. They have also considerable strength and are low in cost. The usual alloying elements for zinc are aluminium, copper and magnesium and they are all held in close limits. The composition of two standard die casting zinc alloys are as follows:
    1. Aluminium 4.1%, copper 0.1%, magnesium 0.04% and the remainder is zinc.
    1. Aluminium 4.1%, copper 1%, magnesium 0.04% and the remainder is zinc.

Aluminium improves the mechanical properties and also reduces the tendency of zinc to dissolve iron. Copper increases the tensile strength, hardness and ductility. Magnesium has the beneficial effect of making the castings permanently stable. These alloys are widely used in the automotive industry and for other high production markets such as washing machines, oil burners, radios, photographs, television, business machines, etc.

4. **Nickel Base Alloys**: The nickel base alloys are widely used in engineering industry on account of their high mechanical strength properties, corrosion resistance, etc. The most important nickel base alloys are discussed below:

    1. **Monel Metal**: it is an important alloy of nickel and copper. It contains 68% nickel, 29% copper and 3% other constituents like iron, manganese, silicon and carbon. Its specific gravity is 8.87 and melting point 1360°C. It has a tensile strength from 390 MPa to 460 MPa. It resembles nickel in appearance and is strong, ductile and tough. It is superior to brass and bronze in corrosion resisting properties. It is used for making propellers, pump fittings, condenser tubes, steam turbine blades, sea water exposed parts, tanks and chemical and food handling plants.

    2. **Inconel**: It consists of 80% nickel, 14% chromium, and 6% iron. Its specific gravity is 8.55 and melting point 1395°C. This alloy has excellent mechanical properties at ordinary and elevated temperatures. It can be cast, rolled and cold drawn. It is used for making springs which have to withstand high temperatures and are exposed to corrosive action. It is also used for exhaust manifolds of aircraft engines.

    3. **Nichrome**: It consists of 65% nickel, 15% chromium and 20% iron. It has high heat and oxidation resistance. It is used in making electrical resistance wire for electric furnaces and heating elements.

    4. **Nimonic**: It consists of 80% nickel and 20% chromium. It has high strength and ability to operate under intermittent heating and cooling conditions. It is widely used in gas turbine engines.

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. Write about physical and mechanical properties of metals.

2. What is cast iron and wrought iron?

3. Write about the effects of impurities like Silicon, sulphur, manganese and phosphorus on steel.

Worksheet

1. *Sailesh is asking his co-workers some questions to test their understanding about the session. Help him by answering the questions:*

   a. Name any two types of Metals?

   b. Physical properties of metals include _____, _____, _____, _____, electrical and thermal conductivity and melting point.

c. Strength and stiffness are counted among the mechanical properties of metal. Yes or No.

d. _____ is the property of metal that enables metal to be drawn into wire.

e. _____ is the opposite property to ductility.

2. *State whether they are true or false:*

a. The carbon content in cast iron varies from 1.7% to 4.5 %.

b. Wrought Iron is considered the purest form of iron.

c. Wrought iron cannot be forget or welded.

d. Steel is an alloy of carbon and iron.

e. Non- ferrous metals are those which contain a metal other than iron as their chief constituent.

Notes

Answers:

1.

a. Ferrous metals and Non- ferrous metals.

b. Luster, color, size and shape, density.

c. Yes.

d. Ductility.

e. Brittleness.

2.

a. True.

b. True.

c. False.

d. True.

e. True.

Machine Optimization

At the end of this module you will be able to:

- state the cutting speed for different materials;
- identify the coolant as per processed material.

Session Plan	
1	Machine Optimization
2	What is Cycle Time
3	Factors Fostering Machine Optimization
4	Key Learnings
5	Worksheet

Machine Optimization

Sailesh reads in a newspaper that people do many things to to increase their profit and decrease their expenditure. He wonders how that can be implemented in case of lathe machines; he decides to asks Mohan.

Sailesh: I have read an article about various ways of increasing profits while reducing the expenditure. How can we do this while working on the lathe machine?

Mohan: It is done through machine optimization.

Sailesh: Machine optimization! What is that?

Mohan: Machine optimization is a process through which you can get maximum output or production in minimum cost, time and machine wear and tear. Let me explain:

Machine Optimization ⟶ Machine Utilization

Machine Optimization ⟶ No Underutilization.

You will see that through machine optimization you can improve both machine utilization and check machine underutilization.

Sailesh: This looks helpful....

Mohan: Yes, Sailesh. It is true that the sole purpose of any workshop is to earn profit in terms of money and the enemy of profits is ideal time for setting, machine breakdown, tool breakage and job rejection.

Sailesh: This simply means that it is very important to take care of machine optimization to earn good profits and do away with the enemies of profits. I will always remember it.

What is Cycle Time

Mohan has just detected another break in the cutting tool. This is the second tool that has been damaged in the past week. He sternly tells the worker to be careful as frequent breakage in the tools is bound to increase the cycle time. Sailesh is a little puzzled as to what Mohan's means by cycle time. He decides to ask Mohan…

Sailesh: Mohan, what is cycle time?

Mohan: Cycle time is the time it takes to complete a production run divided by the number of good work pieces produced. You see Sailesh; during production runs there may be activities occurring that are not very productive. Anything that adds to the length of time it takes to complete a production run may increase your cycle time.

Sailesh: But Mohan, how can we avoid these delays?

Mohan: There are many ways to increase machine optimization. This is called cycle time reduction. Let me show you some techniques that will help you use the machine in such a way that we get maximum output/ production with minimum cost, time and machine wear and tear.

Factors Fostering Machine Optimization

Sailesh and Mohan go on to discuss the factors that ensure the proper utilization of machines. Mohan has made a list. Here take a look!

Mohan: Let's begin by presenting some factors, including basic premises that will help you understand how to improve machine utilization.

1. Always use coolant of proper grade for a particular operation for a particular material.
2. Always use feed, speed and depth of cut ratios as recommended by the manufacturer.

Remember friends, increasing any of these parameters no doubt will lead to more production per hour, but will result in very fast wear and tear of the machine. Thus, to earn a few hundred rupees we will lose the accuracy of the machine which costs lakhs.

3. Use a proper tool for a particular type of operation. Avoid using substitutes of cutting tools.
4. Use proper grade of tools like carbide bit tools for operations; it can take heavier cut and feed. Thus, produce more number of components per hour without affecting wear and tear of machine.
5. Choose correct operation sequence for operation / procedure to manufacturing of any component so that job is done in a single setting and loading. Maximum number of side changes should be not more than two. It will reduce your component loading time.
6. We should always mind holding allowance for any job. It will ensure proper clamping of job. Holding length of job should never be less than minimum required length as recommended by the machine manufacturer. Always hold the work piece from center to center if the job is of long size. Otherwise the job bend and may also lead to an accident.
7. If there is a provision of increasing / decreasing force of tail stock then decrease the force; if the machine is thin otherwise the job will bend because of the force of tail stock.
8. Tools should never be over hung because over hanged tool will lead to breakage of tool which may further lead to job rejection and accidents.
9. The work piece should be steady and supported by tail stock.

10. Always load the job within the range of the machine capacity (weight wise). All the machines are designed to support the job load for a particular extent (weight of job). If we load a heavy job on an incapable machine we will never get the required accuracy and finish. The graph of the wear and tear of the machine will be vertical.

For example, if the manufacturer makes any machine to machine aluminum components up to 100 mm diameter and 1000 mm length, and we load a forging of 80 mm diameter and 700 mm length, then dimensional parameters are o.k. but weight of the job of forged material will be many times higher than the weight for which machine is meant.

11. Perform roughing operation on a conventional machine and then finish operation on CNC machine.

12. Ensure consistency in the diameter size of the component. This comes from conventional machining, because it will ensure consistency in the depth of cut. If diameter is plus it will lead to more depth of cut thus also damage tool.

13. Always ensure availability of setting tools while doing setting on the machine and measuring tools. It will decrease machine ideal time thus increase production time.

14. Always use proper size of raw material for any job. For any job which has 20mm final size then take 22 mm diameter raw material. If 22 mm diameter suits then do not go for 25mm, 28 mm or higher size. It will increase machining timing and material cost.

15. The finish of pre final and final cuts is important. Do not waste time by low feed and depth of cut for intermediate operation. The intermediate cuts should not be given too much priority.

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. What do you understand by machine optimization?

2. List the important factors that should be kept in mind for maximum machine optimization.

Worksheet

1. Help Sailesh to answer these questions:

1. Machine optimization ensures _____ with minimum cost, time and machine wear & tear.

   a. maximum time saving
   b. minimum use of resources
   c. maximum output
   d. minimum use of resources

2. Holding length of job should never be less than minimum required _____as recommended by the machine manufacturer.

   a. length
   b. height
   c. weight
   d. breadth

4. Use feed, _____ and depth of cut ratios as recommended by manufacturer.

   a. accuracy
   b. speed
   c. length
   d. size

Notes

Answers:

1.

1. c
2. a
3. b

PPE

At the end of this module you will be able to:

- wear the Personal Protective Equipment

Session Plan	
1	What is Safety
2	Personal Protective Equipment
3	Key Learnings
4	Worksheet

What is Safety

Reading about the slew of industrial accidents has put Sailesh in a glum mood. Mohan tries to comfort Sailesh, asserting that safety can prevent such accidents. They then go on to analyze the definition of safety and accidents.

Definition of Safety: Safety is an action which organizes and controls all our acts in such a manner that one should not get involved, exposed ourselves or others in an accident.

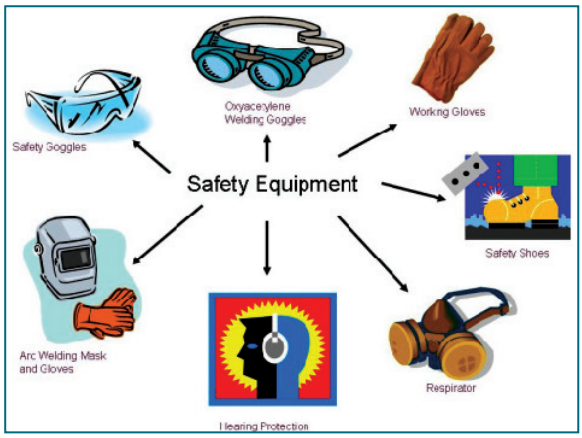
Definition of Accident: It is an unplanned and uncontrolled event in which the action or reaction of an object, substance, person or radiation results in personal injury or probability thereof.

Mohan: Simply said, safety refers to freedom from danger, injury and damage. Always remember Sailesh, a worker's life is most valuable than "safety equipments". Take care of your workers by providing best safety devices. After all "safe work is self-protection and a key to production".

Come Sailesh; let me take you around and show you the personal protective equipments we use in our workshop.

Personal Protective Equipments

Sailesh: Hey friends! I have just taken a round of the workshop and I already feel much safer. The Personal Protective Equipments (PPE) in our workshop includes laboratory/workshop coats, safety glasses or goggles, masks and respirators, hard hats, safety gloves and shoes. I have taken some snapshots for you. Here, take a look.



Sailesh was almost done for the day, when Mohan offers him the chance to try out the PPE. As expected Sailesh jumps at the chance. While Sailesh tries on his gear, Mohan stresses on careful handling of the PPE.

Mohan: Remember Sailesh, laboratory coats or gowns must always be properly worn and fastened. They must be kept in good condition and replaced if holed or torn. Of course, it goes without saying that they should be cleaned and washed regularly.

You must look after other PPE and keep it clean. Avoid leaving it on a workshop bench as it may get contaminated. Report defective or ill-fitting equipment to your manager or supervisor and have it replaced.

Non-disposable facemasks and respirators require regular cleaning and maintenance. To ensure effectiveness, they must first be face-fit tested at the Occupational Health. All PPE should be taken off before leaving the workshop area and entering other areas, such as staff rooms or offices.

Once Mohan is assured that Sailesh can properly use and care for the PPE needed to do his job, he sends him packing for the day.

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. List the types of personal protective equipment you must use when doing your work/ tasks

2. What should you look for to determine that your PPE is in good working condition?

Worksheet

1. *Help Sailesh answer the questions listed below by picking out the correct option.*

   a. _____ is a type of PPE.

      i. Gloves
      ii. Iron Rod
      iii. Vernier Calliper

   b. When should PPE be worn?

      i. When everyone else wears it
      ii. When your Manager is looking
      iii. When working in the workshop

   c. b.What should you do if you find damaged PPE?

      i. Nothing, hope someone else will sort it out
      ii. Tell a supervisor immediately so he can replace it
      iii. Try to repair it yourself

   d. What should you do with PPE after you have used it?

      i. Put it away in your locker or assigned place
      ii. Leave it on top of the workshop bench so, it can be easily found next time
      iii. Throw it in the corner, hoping someone will put it back in its place

   e. Why do we use PPE?

      i. Because other employers do
      ii. Because it looks nice
      iii. To protect employees against injuries

2. *Sailesh's friend Ajay has worn his PPE incorrectly. Help him out by matching the PPE from the column A with the body part it protects from column B.*

Column A		Column B	
a.	Helmet	i.	Eyes
b.	Safety Goggles	ii.	Hands
c.	Gloves	iii.	Head
d.	Respirators	iv.	Ears
e.	Hearing Protection	v.	Nose/ Lungs

3. *Sailesh has sent you a host of pictures of the various personal protective equipments. Identify the image and give it the right name.*

a. Safety shoes	
b. Helmet	
c. Safety goggles	
d. Hand gloves	
e. Hearing protection	

Notes

Answers:

1.

a. Gloves
b. When working in the workshop
c. Tell a supervisor immediately so he can replace it
d. Put it away in your locker or assigned place
e. To protect employees against injuries

2.

a. c b. a c. b d. e
e. d

3.

a. Safety shoes
b. Helmet
c. Safety goggles
d. Hand gloves
e. Hearing protection.

Generic Safety

At the end of this module you will be able to:

- observe the safety measures while working.

Session Plan	
1	What is an Accident
2	Steps to Prevent Accidents
3	Preventive Measures
4	Key Learnings
5	Worksheet

What is an Accident

Sailesh: Welcome back friends. After yesterday's session I was a little worried about all the accidents that can take place in the workshops. Now that I have spoken to Mohan I am a little more relaxed. In our little chat this morning, Mohan told me a great deal about workplace accidents. Let me start by recapitulating the definition of an accident.

It is an unplanned and uncontrolled event in which the action or reaction of an object, substance, person or radiation results in personal injury or probability thereof.

The definition may sound like a string of big words joined together. Don't worry, read it again and SLOWLY this time around.

Simply said, an accident is an incident that happens unexpectedly and unintentionally, typically resulting in damage or injury.

Turning back to main question at hand- why do accidents occur? Thankfully for you I had jotted down some of the unsafe conditions and acts that came up in my discussion with Mohan. Here take a look.

a. Unsafe Conditions:

- Lack of proper safeguard on machinery
- Hazardous arrangement, procedure etc.
- Improper ventilation
- Improper illumination
- Unsafe dress

b. Unsafe Acts:

- Using unsafe equipments or equipments unsafely
- Operating or working at unsafe speed
- Unsafe loading, placing and combining
- Taking unsafe position or posture
- Making safety devices inoperative

In addition to this, it is very important to realize the cost that we have to pay for an accident. Sometimes this cost is irreparable. Besides the loss of life there are other concerns too.

- Cost of lost time of injured employee
- Cost of time lost by supervisors and other executives
- Cost due to damage to the machine, tools, material and property
- Cost due to loss of production
- Cost of compensation and medical aid
- Overhead cost (expense of light, rent and idle machines etc)

So you see friends...accidents results in sudden losses and often cost both the victim and the company. We must take adequate safety measures to prevent accidents. Now, while you process this information, I am going learn about some steps that can prevent accidents. After all, precaution is better than cure.

Steps to Prevent Accidents

Sailesh: Hello Mohan! This morning you were about to tell me how to prevent accidents, before you were called away for an important task.

Mohan: Oh yes Sailesh, I remember. I fished out this safety manual for you. It lists out the measures one can take to prevent accidents. Why don't you go through these while I get on with this job.

Sailesh starts reading eagerly.... Here is a gist of what he read.

1. Dress and Behaviour

- Close fitting uniform should be worn in the workshop.
- Do not wear rings, watches, scarves and tie, because they can get caught up in the moving parts of the machinery.
- When operating machine tools, long shirt sleeves should be rolled up.
- Long hair should be tied and covered with a safety cap.
- Safety goggles must be worn while chipping, turning, milling, grinding, drilling, etc.
- Shoes having oil resistant thick soles should be worn.
- Do not talk to, or otherwise distract anyone who is working with tools or machinery.
- Fooling about in the workshop, especially near machine tools can cause serious injury.
- Keep clear of moving parts of machine and loads being hoisted.
- Never stand under a load supported by lifting device.
- Do not leave a machine unattended while it is running. Do not leave equipments in dangerous positions.
- Use correct gang ways and watch where you walk.

2. Safety with Hand Tools:

- Files must never be used as levers. They should always have a proper handle fitted to them. Chisels with mushroom head are dangerous. When chipping always make sure that chips do not hit somebody standing nearby.
- Always hold the work firmly in a vice or other holding device. If the work shifts during operation a nasty injury may result.
- Always use correct size of spanner and avoid the use of packing or extension handles.

- Always use a screw driver which fits correctly in the screw head. Do not hold the work in hand while tightening or loosening a screw. The screw driver may slip and cause injury in hand.

3. Safety with the Machine:

- Do not try to operate any machine before you fully understand its mechanism.
- Be sure how to stop a machine before you start it. Never operate a machine unless all safety guards are in position.
- Always keep the shop floor free from oil, grease, tools and other cuttings.
- Job must be clamped properly.
- Cutter must be held properly.
- Only recommended speeds and feeds should be used.
- Use cleaning brush to clear off sward from the machine.
- Do not attempt to measure the job when it is in moving position.
- Do not try to change gears when machine is in running position.
- Do not try to stop rotating tool job by hand.
- Only recommended lubricants and coolants should be used.
- Grinding wheel should be checked for crack before fitting it.
- Switch off power while mounting or dismounting a chuck and removing job from the machine.
- Chuck key must be removed before starting the machine.
- A wooden plank should be used on bed ways of lathe while mounting and dismounting the chuck.
- Get first aid immediately for any injury.

Sailesh is now fully aware of the preventive measures that will keep him safe.

Preventive Measures

Sailesh: Before we close this session; let's recapitulate some of the important preventive measures we must take while working on a lathe machine:

1. Safety Guards. By providing proper safeguards to the machines, accidents can be prevented. Some guards are built into a permanent casing, while some are attached afterwards.
2. Fencing: Machines or their parts should be fenced when it is not possible to provide safeguards.
3. Hoists, cranes and lifts etc. must be of sound constructions. They must be tested periodically.
4. Physical Conditions: Sufficient illumination and ventilation should be provided. Floor should be free from oiliness. Floor should be kept clean.
5. Safety measures include special clothing for the protection of body, such as gloves, apron, goggles etc. Lose clothing may be a source of danger.
6. Repair work on machines should not be done when it is running.
7. All the tools should be kept at their proper places.
8. Chips should not be removed by hand.
9. Wear PPE.

Sailesh: That's all for today. See you on the other side of this page till then be safe!

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. What is an accident? Give various causes of accidents.

2. List some steps to prevent accidents.

Notes

Basic Parts and Working Principle

At the end of this module you will be able to:

- identify the parts of the conventional lathe machine;
- explain the working principle of a conventional lathe machine.

Session Plan	
1	Introduction to Lathe Machine
2	Function of Lathe Machine
3	Parts of a Lathe Machine
4	Functions of Different Parts of Lathe
5	Key Learnings
6	Worksheet

Introduction to Lathe Machine

Today Sailesh will be introduced to Lathe Machine. He is excited and wants to try his hands on lathe machine. Mohan tells him that it is important to know about Lathe Machine before he really starts working on it.

Mohan: Shailesh! Let me introduce the lathe machine to you before you actually start working on it. Don't hesitate to ask questions in case you have any doubts.

Lathe machine is one of the oldest machines used to cut wood or metal. The traditional lathe machine uses a rope that is wound round the work. One end of the machine is attached to a flexible branch of a tree while the other end is pulled by a man, causing the job to rotate intermittently.

Sailesh: It sounds interesting!

Mohan: Now, I will tell you about the history of lathe machine.... don't worry it's not boring....

Earlier, hand tools were used. Later a strip of wood called "lath" was used to support the rope and that is how the machine came to be known as "lathe". This device continued to develop through centuries and in the year 1797 Henry Maudslay, an Englishman, designed the first screw cutting lathe machine. This lathe machine is the forerunner of the present day high speed, heavy duty production lathe, a machine tool which has practically given shape to our present day civilization by building machines and industries.

Function of the Lathe Machine

Mohan: Sailesh, today I will tell you about the function of a lathe machine.

Sailesh: Please tell me.

Mohan: Ok, the function of the Lathe machine is to cut a metal or wooden block into a desired shape and size. The block, also called the work piece is held securely and rigidly on the machine, rotated against the cutting tool. The cutting tool removes the extra metal uniformly from the clock and gives the block a desired shape. To cut the material properly the tool should be harder than the material of the work piece, should be rigidly held on the machine and should be fed or progressed in a definite way related to the work.

Is it clear?

Sailesh: Yes, please continue.

Mohan: Let me tell you the Specification of Lathe:

1. Height of Centre: The height of the centers measured over the lath bed.
2. Swing over Bed: Swing or max diameter that can be rotated over the bed ways.
3. Swing or Diameter over Carriage. This is the largest diameter over carriage. This is the largest diameter over carriage of work that will revolve over the lathe saddle.
4. Distance between Centres: Max job length in mm that may be held between the centers (head stock & tail stock)
5. Bed Length: This indicates the approx. floor space occupied by the lathe.

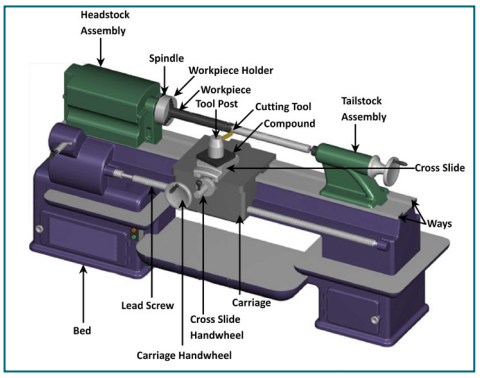
Parts of a Lathe Machine

Now, it's time for Sailesh to get familiar with the parts of a lathe machine.

Mohan: Sailesh let me show you a lathe machine diagram. I have marked all the parts of machine in the diagram.

Sailesh: Thank you, Mohan. I will keep this image with me always for reference.

Lathe Machine



Functions of Different Parts of Lathe

Mohan: By now, you can identify all parts of the lathe machine. I think this is the perfect time to tell you the functions of different parts of Lathe.

A lathe machine has different parts and all parts have different function:

1. **Bed**: The lathe bed forms the base of the machine. The headstock and the tailstock are located at either end of the bed and the carriage rests over the lathe bed and slides on it.

The two functions of a lathe bed are:

- To locate the fixed units in accurate relationship to each other.
- To provide slides upon which the operating units can be moved.

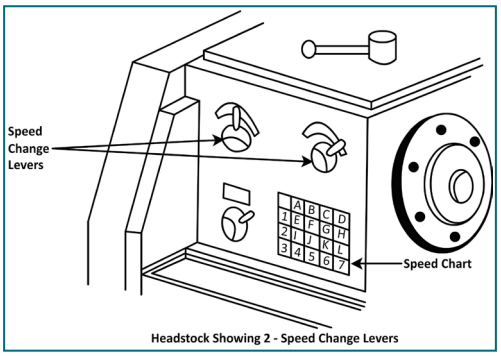
2. **Headstock**: The headstock is positioned on the left hand end of the lathe bed. The headstock provides mechanical means of rotating the work piece at multiple speeds. It comprises of:

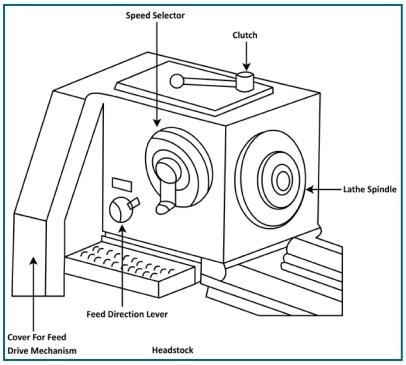
- Hollow spindle
- Mechanism for driving and altering the spindle speed

All the parts are housed within the headstock casting.

Its main functions are to:

- Provide a means to assemble work-holding devices.
- Transmit the drive from the main motor to the work to make it revolve.
- Accommodate shafts with fixed and sliding gears for providing a wide range of work speeds.
- Have shift leavers to slide gears to bring in mesh for different speeds.
- Have a means for lubricating the gears, shafts and bearings.

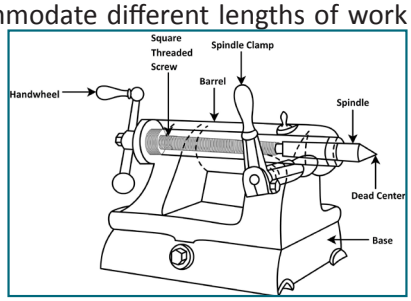




3. **Tailstock**: The tailstock is located on the inner ways at the right hand end of the bed.

Functions of tailstock:

- It supports the other end of the work when it is being machined between centers.
- It holds a tool for performing operations such as drilling, reaming, tapping, etc.
- To turn the external taper by offsetting the body of the tailstock with respect to the base.
- To perform external operation of the shaft held between centers.



The tailstock has two points, namely “base” and “body”. To accommodate different lengths of work piece, the body of the tailstock can be adjusted along the ways chiefly by sliding it to the desired position where it can be clamped by bolts and plates. The upper casting of the body can be moved towards or away from the operator by means of the adjusting screws. The body is bored to act as a barrel by means of a screw when the tailstock spindle that moves in and out of the barrel by means of a screw when the tailstock hand wheel is turned. The front of the spindle has a taper hole into which the dead center or other tool fit. The screw thread is left handed, so that clockwise rotation of the hand-wheel causes the spindle to advance, while anticlockwise rotation causes the spindle to be drawn inward and ultimately the end of the screw strikes the back of the dead center or any tool that is fitted into the hole. To remove tools from the spindle, it is therefore, only necessary to back up on the hand-wheel until the spindle end is nearly inside the casting. After the adjustment is made, the spindle is clamped in position by tightening the locking bolt on split lug.

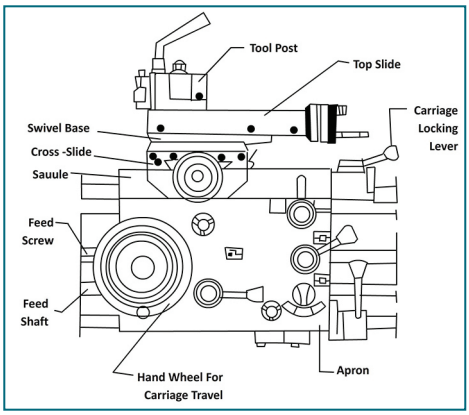
4. Carriage

The carriage is the part of the lathe which slides over the bed-ways between the headstock and the tailstock. It provides various movements for the cutting tool manually as well as by power feed. The carriage can be locked on the bed at any desired position by tightening the carriage lock-screw. The carriage consists of the following parts: saddle, cross slide, compound side or compound rest, tool post, and apron.

a. **Saddle**: It is a casting that fits over the bed and slides along the ways. It carries the cross slides and tool post. There is a provision for locking the saddle to prevent any movement when surfacing operations are carried out.

b. **Cross slide**: The cross slide comprises a casting, machined on the underside for attachment to the saddle and carries the tool post or compound rest. In order to move the cross slide, the feed screw is turned by rotating the hand-wheel. Transverse movement is obtained when the nut mounted on the feed screw is engaged with the binder screw of the cross slide.

c. **Compound Rest**: The compound rest or compound slide is mounted on the top of the cross slide and has a circular base graduated in degrees. It is used for obtaining angular cuts and short tapers as well as convenient positioning of the tool to the work.



d. **Tool Post**: This is located on the top of the compound rest to hold the tool and to enable it to be adjusted to a convenient working position. The type and mounting of the tool post depends upon the

class of work for which it is to be used. The rigidity of the tool holder and effective method of securing are the essential factors in designing a tool post.

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. What is a lathe machine? Give its functions in brief.

2. Write about functions of any two parts of Lathe

3. Describe tailstock.

Worksheet

1. *Fill in the blanks:*

   a. _____ is a casting that fits over the bed and slides along the ways.

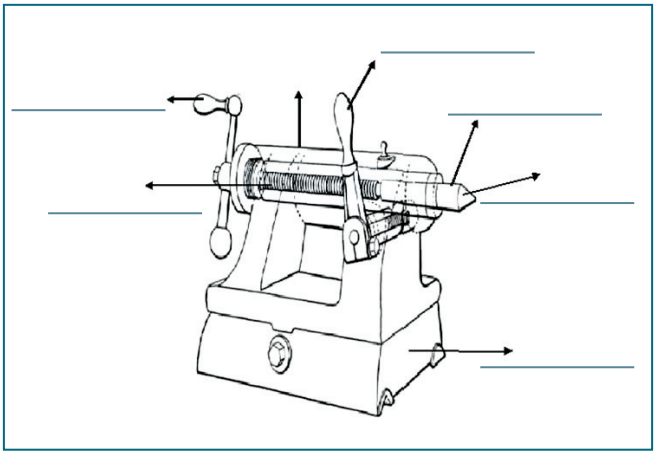
   b. _____ is used for obtaining angular cuts and short tapers.

   c. Carriage slides over the bed ways between the _____ and the _____.

   d. _____ help in holding a tool for performing operations such as drilling, reaming, tapping etc.

   e. _____ is positioned on the left hand end of the lathe bed.

2. *Label the different parts of lathe machine in the diagram given below*



Notes

Answers:

1.

a. Saddle

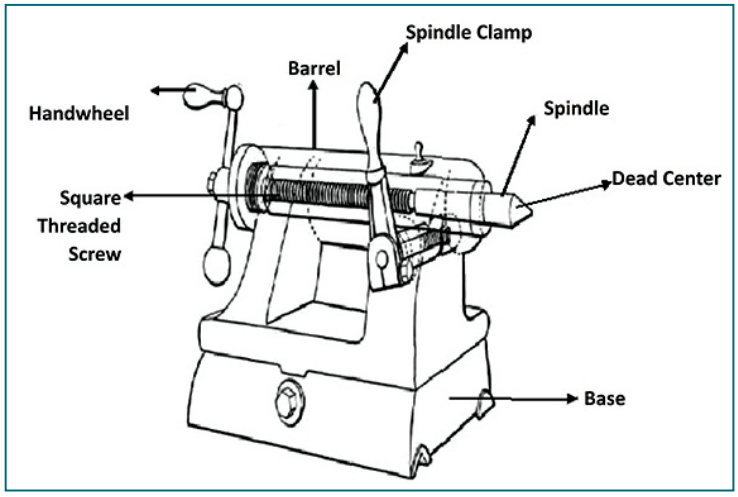
b. compound rest or compound slide.

c. headstock and tailstock.

d. Tailstock

e. headstock.

2.



Cutting Tool Angles – Single Point Cutting

At the end of this module you will be able to:

- explain about the types of cutting tools;
- explain the angles of various lathe cutting tool;
- list the cutting tools.

Session Plan	
1	Cutting Tools
2	Properties of Cutting Tools Material
3	Types of Single Point Cutting Tools
4	Cutting Tool Material
5	Tool Angles
6	Selection of Angles
7	Key Learning
8	Worksheet

Cutting Tools

Sailesh is looking at the chart showing some machinery cutting tools. Getting interested in the cutting part of the lathe machine, he calls Mohan to get information about cutters.

Sailesh: Mohan can you tell me how we do the cutting part with lathe machine?

Mohan: Cutting is done with the help of cutting tools or cutters.

Sailesh: Cutting tools, what is that?

Mohan: In lathe machines, a cutting tool or cutter is a tool that is used to remove material from the work piece by means of shear deformation (removal of extra metal).

Sailesh: All types of cutting can be done in one way only???

Mohan: No Sailesh, cutting is done by either single point or multi point tools. Single point tools are used when we need to undertake some turning, shaping, and remove material by means of one cutting edge. You need multipoint tools in case of milling and drilling. The grinding tools are multipoint tools.

Sailesh: Oh ok, it sounds interesting..... I will like to see them too.

Mohan: Sure, come I will show you cutting tools.

Properties of Cutting Tool Material

Sailesh has seen the cutting tools. While holding a cutting tool in his hand, he asks Mohan about various properties of cutting tools material.

Sailesh: Mohan, can you tell me what are the properties of cutting tools material that I should look for?

Mohan: I was about to tell you, anyway, the properties of cutting tools materials are:

- **Hardness**: Only when the tool is harder than the material it has to cut , it can cut / shear the other material
- **Hot Hardness**: The cutting tool material should retain its original hardness at higher temperatures.
- **Toughness**: If the tool is not tough, it would break when the cutting loads and jerks acts on it.
- **Wear Resistance**: If the material is not wear resistant, it would wear out fast without giving much life to the tool cutting edge.
- **Low Coefficient of Friction**: The coefficient of friction at the chip - tool interface should be as low as possible, which is achieved by the way of right tool angles, proper cutting speed and use of proper coolant. This also gives good cutting finish.
- **Cost**: The cost of cutting tool material should be within reasonable limits.
- **Ease of Manufacture**: The raw material for the cutting tool should be easily available and it should be easy to manufacture the tools.

Sailesh: Oh, thank you Mohan. This information will help me a lot in machine work.

Mohan: My pleasure.

Types of Single Point Cutting Tools

Sailesh is getting curious to know about cutting tools in detail. Filled with enthusiasm he asks Mohan about various types of single point cutting tools.

Sailesh: I am keenly interested to know in detail about the cutting tools of lathe machine. I remember that you told me two types of cutting tools; single point and multipoint cutting tools.

Mohan: Yes, Sailesh. I will tell you about single point cutting tools as it of utmost importance.

Well, the types of single point cutting tools are as following:

- **Solid Tools**: These tools are generally made of high speed steel, Tungston Carbide, Titanium Nitride Coated Tungston Carbide Bits and Ceramic Tool Bits have square or rectangular cross – section, Cutting Tool Bits with cutting edges, clearance angles ground as per requirement.
- **HSS Tools**: These are squares, rectangular (5 to 20 mm square/ rectangular), 75 to 150 mm long pieces used with a tool holder.
- **Tip Tools**: A tip tool is normally used for turning hard materials/ or high speed machining. Tips made out of Tungston Carbide, Titanium Nitride Coated Tungston Carbide and Ceramic Bits are fixed to the tool shanks by means of screws or brazed in case of tungsten carbide on the shank made of carbon steel.
- **Throw-away Tools**: These tools are made of carbide bits of rectangular, square or triangular shape and are clamped in special tool holders. They are used as throw-away inserts in mass production. Throwable bits are available in Carbide, Titanium Nitride Coated Tungston Carbide and Ceramic Bits.

Cutting Tool Material

Sailesh: Mohan, if we need cutting tools to be good in all the properties, then what are cutting tools made up of?

Mohan: Very good question Sailesh; if we want to have cutting tools of best quality then we need to know the material of the cutting tools.

Well. cutting tools are made up of the following:

- **High Carbon Steel**: It contains carbon from about 0.75 to 1.25%. It is useful for marking light finishing cuts and for machining soft materials.
- **High Speed Steel**: Also known as HSS, it contains very little carbon. It contains alloying elements like chromium, manganese, molybdenum, tungsten and cobalt. It is tough enough to withstand shocks during cutting and retains hardness at higher speeds than carbon steel.
- **Cemented Carbide Tip**: Carbide is the hardest cutting material available. They are very brittle too. Majorly used as throw away type tool bit.

Sailesh: Oh! That means the cutting tools are really hard and that's what make them highly reliable.

Tool Angles

Mohan is addressing all his workers that accurate cutting is possible only if the tool is used in a proper angle. Confused about what are tool angles, Sailesh asks Mohan:

Sailesh: What is a tool angle and how does an angle effect the operation?

Mohan: Whenever we need to cut a certain piece, it is preferred to use sharp objects.

Sailesh: Yes, sharp objects like blade?

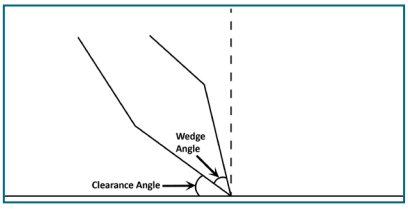
Mohan: Yes, it is worth remembering that in case of sharp objects the force is applied over a thin area. Hence, whenever we need to cut metal parts, it becomes essential to use proper angle of cutting.

Sailesh: What is the role of angles when the cutting object are made up of such hard materials like high speed steel or even materials like diamond, ceramics or tungsten carbide?

Mohan: You are somewhat right Sailesh, but the thing to remember is that HSS material is selected in the form of square rods or round rods and ground to provide necessary cutting angles to aid the process of machining.

Now, I tell you the 3 basic angles used on a cutting tool:

1. **Clearance angle (α) –**



- Angle between machined surface on the work piece and the trailing edge of work piece.
- This is provided for avoiding friction (rubbing) arising due to contact between work piece and cutting tool.
- Clearance angle is selected based on the work piece material.

The types of clearance angles:

- **Front Clearance Angle**: The front clearance angle prevents the front flank of the tool from rubbing against the work. A minimum clearance angle is given to provide maximum support to the tool cutting edges by increasing the lip angle. The front clearance angle should be increased for large diameter work.
- **Side Clearance Angle**: The side clearance angle prevents the side of the tool from rubbing against the work when longitudinal feed is applied. The side clearance angle depends upon the amount of feed given. Larger feed will require greater side clearance angle.

2. Wedge angle (β)

- Is the angle between rake surface and clearance surface.
- Wedge angle imparts strength to the cutting tool.

3. Rake angle (□)

- Angle between plane perpendicular to machined surface on the work piece and the surface on which chip flows.
- Rake angle is provided for easy penetration of cutting tool into the work piece and remove material in the form of chip.

The back rake angle affects the ability of the tool to shear the work material and form the chip.

It can be positive or negative. Positive rake angles reduce the cutting forces resulting in smaller deflections of the work piece, tool holder, and machine. If the back rake angle is too large, the strength of the tool is reduced as well as its capacity to conduct heat. In machining hard work materials, the back rake angle must be small, even negative for carbide and diamond tools. The higher the hardness; the smaller the back rake angle. For high-speed steels, back rake angle is normally chosen in the positive range.

- **Front rake angle**: The front rake influences the machining condition when the tool removes metal from its cutting edge.
- **Side rake angle**: The side rake influences the machining condition when it removes metal on its side cutting edge only. A knife edge turning tool may not have any front rake but must have a definite amount of side rake. Side rake also allows the chips to flow by the side of the tool without getting entangled with the tool post.

The other angles provided on a tool are:

Nose Radius: the nose of a tool is slightly rounded in all turning tools.

The functions of nose radius are as follows:

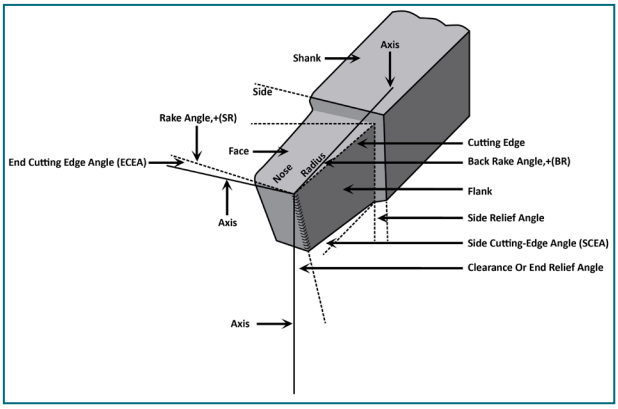
1. Greater nose radius clears up the feed marks caused by the previous shearing action and provides better surface finish. All finish turning tools have greater nose radius than rough turning tools.
2. It increases the strength of the cutting edge, tends to minimize wear taking place in sharp pointed tool with consequent increase in tool life.
3. Accumulation of heat is less than that in a pointed tool which permits higher cutting speeds.
4. Slight reduction in cutting force may be obtained.

Very large nose radius may cause chatter. For rough turning, nose radius is usually 0.4mm and for finish turning it varies from 0.8 to 1.6mm.

Side Cutting Edge Angle: The side cutting edge angle of turning tools vary from 00 to 900. The following are the advantages of increasing side cutting edge angle.

1. It increases tool life as, for the same depth of cut the cutting force is distributed on a wider surface.
2. It diminishes chip thickness for the same amount of feed and permits greater cutting speed.
3. It dissipates heat quickly for having wider cutting edge.
4. It improves surface finish.

End Cutting Edge Angle: The main function of the end cutting edge angle is to prevent the trailing front cutting edge of the tool from rubbing against the work. The end cutting edge angle ranges from 8 to 15 degree. A large end cutting edge unnecessarily weakens the tool.



Always use the formula $\alpha + \beta + \Upsilon = 90$ degrees

Sailesh: Ok, now I understood what type of cutting tool angle you were talking about!

Mohan: Yes, always keep in mind that the action taken in right angle helps you achieve best results.

Selection of Angles

Sailesh: I am getting a little puzzled about which tool angle to select? And how will we decide which tool angle is best?

Mohan: Relax Sailesh! In case of cutting tools, geometry depends mainly upon the properties of the tool material and the work material. I have given the standard terminology in the following figure. For the single point tools, the most important angles are the rake angles and the end and side relief angles.

Keep in mind that the tool angles vary under different machining conditions. The following table gives the recommended angles as per the material used.

Material	Front rake deg.	Front clearance deg.	Side rake deg.	Side clearance deg.
Mild Steel	10 – 12	6 - 8	10 - 12	6 - 8
Stainless Steel	5 - 7	6 - 8	8 - 10	7 - 9
Aluminium	30 - 35	8- 10	14 - 16	12 - 14
Brass	0 - 6	8 - 10	1 - 5	10 - 12
Cast iron	3 - 5	6 - 8	10 - 12	6 - 9
Copper	14 - 16	12-14	18 - 20	12 - 14

- Harder materials are provided with smaller clearance angle and smaller rake angle. In some cases rake angle is provided in the negative direction while using highly brittle and hard tool materials like tungsten carbide.
- The cutting force and power requirement changes due to change in cutting tool geometry.
- Application of coolant aids in reducing the cutting force required for machining while using a cutting tool with specific geometry.

Sailesh: Oh! That clarifies all my doubts. I have attained a lot of information about the cutting tool angles; it's my turn to check these angles in practice.

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. List all properties of Cutting Tools Material.

2. Give the 3 basic angles used on a cutting tool.

3. Give various types of Single Point Cutting Tools.

Worksheet

1. *Sailesh has made a list of some points to remember. Help him by telling which ones are correct. Put a (Y) for 'YES' on the correct statements and (N) for 'No' on the incorrect statements:*

   a. Clearance angles, wedge angle and Rake angles are the three tools angles _____

   b. Front clearance and side clearance are two types of clearance angles _____

   c. Wedge angles give strength to the cutting tools _____

   d. The end cutting edge angle ranges from 15 to 30 degrees _____

   e. Side wedge and slide wedge angles are two types of wedge angles _____

2. *Choose the correct options:*

a. Cutting tool material is made from:

i. High speed steel ☐
ii. High speed carbon ☐
iii. High speed cement ☐

b. Select the type of single point cutting tool from the following:

i. Throw- away Tools ☐
ii. Liquid based Tools ☐
iii. Perpendicular Tools ☐

c. Choose the option which is NOT among the properties of cutting tool material:

i. Hardness ☐
ii. Water Resistance ☐
iii. Ductility ☐

d. Select the option which is NOT a type of clearance angle:

i. Front clearance angles ☐
ii. Slide clearance angles ☐
iii. Side clearance angles ☐

e. Select the operation which is NOT done with single point tools:

i. Turning ☐
ii. Shaping ☐
iii. Milling ☐

Notes

Answers:

1.

1.	Y	2.	Y	3.	Y	4.	N	5.	N

2.

a.	i	b.	i	c.	iii	d.	ii	e.	iii

Outer Diameter (OD) Turning

At the end of this module you will be able to:

- identify the inputs needed with reference to the outer diameter turning;
- perform the operation.

Session Plan	
1	Turning
2	Work Setting
3	Tool Setting
4	Steps for OD Turning
5	Key Learnings
6	Worksheet

Turning

After being introduced to the Lathe Machine; today Sailesh would be introduced to the Turning operations. Sailesh is excited about the things that are to follow in the day. He is preparing himself by reading a book on Turning. He reads what is Turning in the book which describes it as “Turning is a machining process in which a cutting tool, typically non-rotary tool bit, describes a helical tool path by moving more or less linearly while the work piece rotates. “

He goes on reading.

The tool's axes of movement may be literally a straight line, or they may be along some set of curves or angles, but they are essentially linear (in the non-mathematical sense). Usually the term "turning" is reserved for the generation of external surfaces by this cutting action, whereas this same essential cutting action when applied to internal surfaces (that is, holes, of one kind or another) is called "boring". Thus the phrase "turning and boring" categorizes the larger family of (essentially similar) processes. The cutting of faces on the work piece (that is, surfaces perpendicular to its rotating axis); whether with a turning or boring tool, is called "facing", and may be lumped into either category as a subset.

Here in this book; he read about facing which he finds hard to understand. Finding Mohan bit busy, he plans to ask him about it later on.

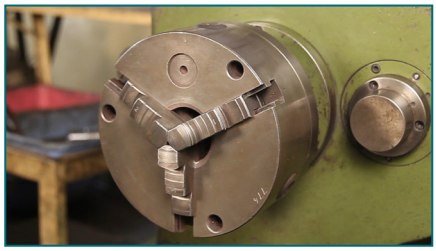
Work Setting

Sailesh has read about Turning, he now wants to actually work on the machine. He remembers that Mohan had told him that before working practically on the machine, it is important to have proper knowledge of work setting.

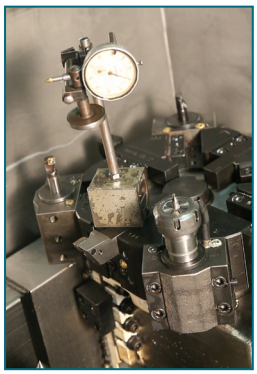
It reminds him that Mohan told him that in case of a turning job, it is essential to do the truing of the job in the independent chuck. If turning is not done the following will be the results:

- Uneven load on the cutting tool
- For the same depth more metal will be removed from the out of centre portion
- Surface turned may not be cylindrical.

Now, Sailesh is thinking how to actually undertake the job of truing. He refers to the book again while he is telling his friends the process to be adopted for turning:



- Keep the main spindle in a neutral position.
- Measure the job diameter with an outside calliper.
- Position the jaws of the independent chuck, equidistant from the centre.
- The distance between the inner face of the opposite jaws is equal to the diameter of the work.
- Open the adjacent jaws sufficiently enough to insert the work.
- Place the work inside the chuck, keeping sufficient portion outside the chuck for turning, and tighten the two adjacent jaws enough to grip the work.



- Place the dial gauge on the bed-ways close to the chuck.
- Adjust the pointer to make its tip move close to the top or side portion of the work with a minimum gap.
- Rotate the chuck by hand and observe if there is any deviation from the dial gauge pointer observe for the reason may be the piece is not properly fitted into the chuck or the job itself is not straight. If there is margin with respect to the diameter to be turned then continue else replace the piece.
- Engage the spindle levers at about 250 rpm and run the machine.
- Touch the pointer of dial gauge and see the fluctuation once again.
- If the fluctuation is not much then it indicates that the work is trued.
- Finally, tighten the jaws with more pressure.
- Check once again for the true running of the work.

Tool Setting

After getting an understanding of the importance of work settings, now Sailesh is learning about the tool setting as he wants to convey the information to all his friends.

He thinks that the tool settings are similar to work settings. He calls Mohan to clarify.

Sailesh: Are tool setting and work setting similar?

Mohan: No not all. Let me tell you what tool setting is.

A tool must be properly set with reference to the line of the centres. If the tool nose is positioned slightly above the line of centres, the rake angle is increased and the clearance angle is decreased. This also decreases the cutting angle. It has a somewhat favourable effect on the cutting conditions. The chips flow over the tool face with less friction. The chip is coiled into a spiral, hence less deformed and thus the cutting force is reduced.

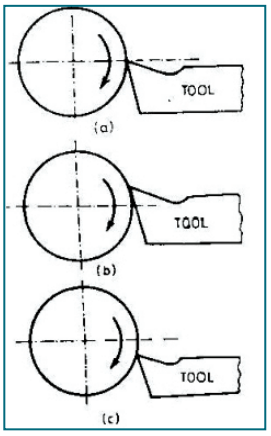
But on the other hand, the friction between the flank and the work is increased. The cutting edge has a tendency to dig into the work, which may result in chattering, reduced accuracy. A poorer surface finish and in some cases, even breakage of the tool.

In order to clarify Sailesh's thinking Mohan asks him to jot down the important points that one should keep in mind while doing tool setting:

- For turning operations set the nose of the tool slightly above the centre of the work.
- Normally the cutting edge should be set to a maximum of 0.5mm above the centre. If it is set too high, the work will rub against the tool and the cutting action will be inefficient. If the tool is set too much below the centre of the work, it will lead to breakage of the tool and very poor surface of the job.
- The cutting tool should be set in such a manner that it does not extend too far out of the tool holder as this will lead to chattering finish on the job.
- Too much overhang will cause vibrations and may break the tool.

To give better understanding of the tool settings Mohan makes some diagrams with steps:

- Setting on centre line: The correct height of the tool is always required. It is obtained by placing metal sheet packing underneath. The packing must be straight and clean.
- Setting above the Centre Line: [clearance angle] the rake angle becomes smaller, clearance angle reduces thus more friction is created between clearance face of the tool and cut face of the work piece. Chips separate easily and thicker chips can be removed. For roughing operation the tool is occasionally set above the centre line (up to 2% of the work piece diameter).
- Setting below the Centre Line: the rake angle becomes larger and less friction is created between clearance face and cut face becomes smaller and chips are difficult to separate.



Now, Mohan asks Sailesh to make a list of various steps to clamp the tool on the tool post:

Step 1: Clean the tool.

Step 2: Loosen the mounting screw on the tool post using suitable spanner or wrench.

Step 3: Remove old tools if any present.

Step 4: Clean the tool seating surface.

Step 5: Place the tool in the tool post.

Step 6: Clamp the tool by tightening the mounting screws

Step 7: Adjust the centre height of the tool.

This can be done by adjusting the screw in the quick change tool post or by placing shims (small pieces of steel sheets) between the cutting tool and seating surface.

Step 8: Check for approach angle to avoid rubbing of machine surface by the tool.

Work Setting

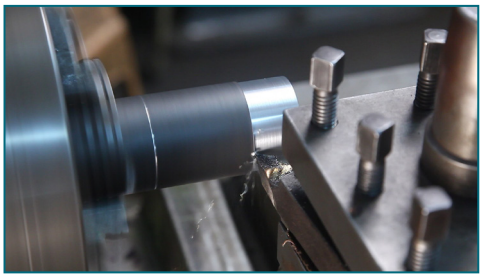
Feeling excited about the turning process; Sailesh requests Mohan to tell him about the OD turning in detail.

Sailesh: I read that the turning processes are typically carried on a lathe, considered to be the oldest machine tools, and can be of various types such as straight turning, taper turning, profiling, facing, drilling, boring, thread cutting (both internal and external), grooving (both internal and external) etc. All these types of turning processes can produce various shapes of materials like straight, conical, curved or grooved work piece. In general, turning uses simple single-point cutting tools. Each group of work piece materials has an optimum set of tools angles which have been developed through the years.

Mohan: You are right Sailesh! I would like to add more things to your learning. Plain turning is done in two stages.

- Rough turning
- Finish turning

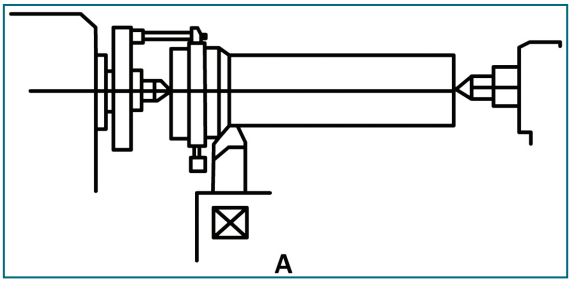
Rough turning:



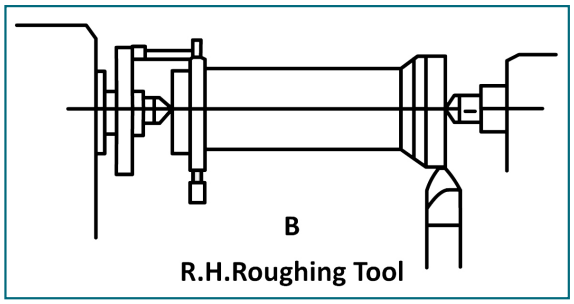
By rough turning we bring the size of the work close to the required size, leaving sufficient metal for finishing. The tool is selected, depending upon the amount of material to be removed.

The normal direction of feeding the tool is from the tailstock end to the headstock end. By rough turning we can remove maximum metal in the shortest time. Surface finish and accuracy are not of much importance during rough turning. (Figs a and b)

The tool cutting edge should be sharp and ground to the recommended rake and clearance angles. Depending upon the material to be turned and the tool material, the recommended cutting speed is to be chosen from the chart available, to determine the spindle speed for plain turning. The selection of the recommended feed rate is also important which greatly depends upon the depth of cut and the cutting speed chosen.

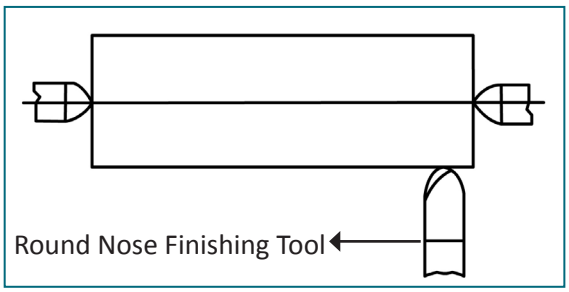


A



B

R.H.Roughing Tool



Finish turning:

It is done after rough turning, and it aims to bring the size of the work within the specified accuracy and with better surface finish. The tool to be used in this case will be a finish turning tool which may be a round nose finishing tool with a larger nose radius than normal.

Finish turning is also carried out from the tailstock end towards the headstock end.

Finishing cut as far as possible is to be limited to one or two depending upon the amount material available for removal. The cutting speed chosen should be higher than that chosen for roughing by 1 1/2 to 2 times, whereas the feed rate may, depending upon the surface finish needed, be as low as possible.

Steps to do OD turning are as follows:

1. Clamp the work piece on the chuck.
2. Turn on the main switch to rotate the spindle and check for trueness.
3. Mount the cutting tool in the tool post and check for the centre height.
4. Select the required spindle speed by changing the belt position in the pulley or by changing the gear shifter lever.
5. Move the slide closer to the starting position of the work piece.
6. Touch the tool on the work piece outside diameter and note the reading on the dial mounted on the hand wheel of cross slide.
7. Pull out the tool away from work piece by moving the cross slide.
8. Move the longitudinal slide to right side to a convenient position for start turning.
9. Adjust the cross slide by adding needed depth of cut.
10. Switch ON the spindle.
11. Move the longitudinal slide to the left side for turning the work piece by rotating hand wheel, up to the required length.
12. After reaching the required length pull out the tool away from the work piece by moving the cross slide by rotating the hand wheel.
13. Move the apron to right side to a convenient position by rotating the hand wheel.
14. Stop the spindle and measure the machined outside diameter using micrometer or vernier calliper.
15. Compare present machined outside diameter and the drawing dimension for deciding further steps of cut.
16. Repeat step 11 to 15 till correct dimensions are achieved.

Sailesh: It is not that easy as I thought it to be! I feel we need to read it once more before I practically work on OD turning. While I go to clarify all my doubts with Mohan, why don't you give this session a quick reading!!

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. Give the process of truing properties before turning.

2. Give various steps to clamp the top on the tool post.

3. Write the steps of OD turning in brief.

Worksheet

1. *Help Sailesh in completing his lines by filling the blank spaces with correct answers:*

   a. In the process of truing it is important to keep the main spindle in a _____ position.

   b. Place the surface on the _____ close to the chuck.

   c. You need to give _____ pressure on the top of the _____ to make the tip on touch the work and feel.

   d. If the tool nose is positioned above the line of centres, the rake angle is _____ and the clearance angle is _____.

   e. For turning operations you need to set the nose of tool slightly above the _____ of the work.

2. *Sailesh is telling about plain turning. Help him by telling him the two types of Plain turning.*

3. *Sailesh has forgotten two results that are expected if truing is not done. He just remembers one result i.e surface turned may not be cylindrical. Help him by telling the remaining two results.*

Notes

Answers:

1.

a. Neutral

b. Bed-ways

c. Slight, Pointer

d. Increased, Decreased

e. Centre

2.

Rough turning and Finish turning.

3.

Uneven load on the cutting tool and For the same depth more metal will be removed from the out of centre portion

Facing

At the end of this module you will be able to:

- identify the tools with reference to the facing operation;
- perform the operation.

Session Plan	
1	Facing
2	Work Setting
3	Tool Setting
4	Steps for Facing
5	Defects Found on Faced Work
6	Key Learnings
7	Worksheet

Facing

Sailesh has learnt how a lathe machine works in the previous session. Now, he is curious to work on it. He thinks that it is easy to create a flat surface with lathe; and wants to do the facing operation.

He requests Mohan to tell him in detail about Facing.

Sailesh: Mohan, I know that for creating a flat surface, we need to perform the facing operation. Please tell in detail about it.

Mohan: Sure Sailesh! The facing operations are undertaken with the help of facing tool. Keep in mind that most of the time the work piece is cylindrical and you need to use a chuck to give face rectangular or odd- shaped work to form cubes and non-cylindrical shapes. When a lathe cutting tool removes metal it applies considerable tangential (i.e. lateral or sideways) force to the work piece. To safely perform a facing operation the end of the work piece must be positioned close to the jaws of the chuck. The work piece should not extend more than 2-3 times its diameter from the chuck jaws unless a steady rest is used to support the free end.

Sailesh: I have understood facing and the facing tool. I am a little confused about the reasons of facing.

Mohan: Relax Sailesh! Let me tell you the purpose of facing:

- To make the surface flat and perpendicular with the axis of the work piece.
- To maintain the length of the work piece.
- To remove the rough surface on the faces of the work.

Work Setting

In the previous session Sailesh learnt about the work settings of OD turning. He doubts that the work settings are same in facing too. Suspiciously, he asks Mohan:

Sailesh: I read about the work settings in OD turning and now you are telling me that facing also has work setting. Are the work setting same in every lathe work?

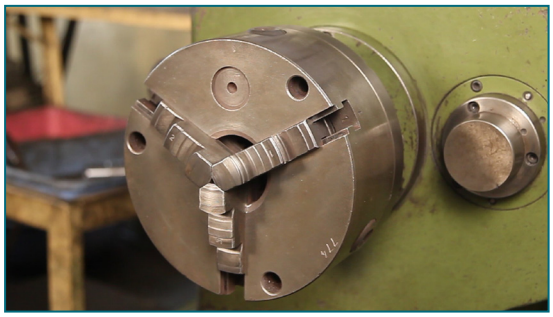
Mohan: Not really! They both are same to some extent. You must listen carefully otherwise you may get confused.....

Just like in OD diameter you need to do the truing job similarly in facing too you have to do the same. It's important to do truing in the independent chuck before doing any operation. In case truing is not done, it will lead to following results:

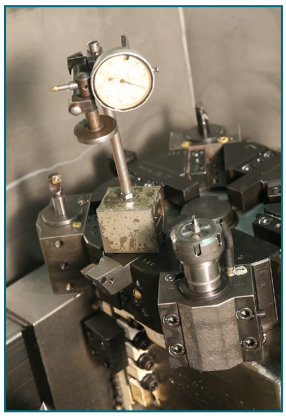
- Uneven load on the cutting tool.
- For the same depth more metal will be removed from the out of centre portion.
- Surface turned may not be cylindrical.

The process to be adopted for truing:

- Keep the main spindle in a neutral position.
- Measure the job diameter with an outside caliper.
- Position the jaws of the independent chuck, equidistant from the centre.
- The distance between the inner face of the opposite jaws is equal to the diameter of the work.
- Open the adjacent jaws sufficiently enough to insert the work.
- Place the work inside the chuck, keeping sufficient portion outside the chuck for turning, and tighten the two adjacent jaws enough to grip the work.



3 Jaw Chuck



Dial Gauge with Stand

- Place the dial gauge on the bed-ways close to the chuck.

Dial Gauge with stand

- Adjust the pointer to make its tip move close to the top or side portion of the work with a minimum gap.
- Rotate the chuck by hand and observe if there is any deviation from the dial gauge pointer if any observe; for the reason may be the piece is not properly fitted into the chuck or the job itself is not straight. If there is margin with respect to the diameter to be turned then continue or replace the piece.

- Engage the spindle levers at about 250 rpm and run the machine.
- Touch the pointer of dial gauge and see the fluctuation once again.
- If the fluctuation is not much then it indicates that the work is trued.
- Finally, tighten the jaws with some more amount pressure.
- Check once again for the true running of the work.

Tool Setting

Sailesh thinks that Tool setting would be same in facing like it was in Outer Diameter. He asks Mohan about it:

Sailesh: Mohan are the tool settings same as we read in Outer diameter yesterday?

Mohan: Not at all, Sailesh. They are different. In case of facing, you need to choose a cutting tool with a slightly rounded tip. A sharp pointed tool may cut little gloves across the face of the work and prevent you from getting a nice smooth surface. You must clamp the cutting tool in the tool post and turn the tool post so that the tip of the cutting tool will meet the end of the work piece at a slight angle during facing. The angle will vary in between 30º to 60º. It is important that the tip of the cutting tool be right at the centerline of the lathe; if it is too high or too low you will be left with a little bump at the centre of the face.

Sailesh: You must note down the steps to clamp the top on the tool post:

Step 1: Clean the tool.

Step 2: Loosen the mounting screw on the tool post using suitable spanner or wrench.

Step 3: Remove old tools if any present.

Step 4: Clean the tool seating surface.

Step 5: Place the tool in the tool post.

Step 6: Clamp the tool by tightening the mounting screws.

Step 7: Adjust the centre height of the tool.

This can be done by adjusting the screw in the quick change tool post or by placing shims (small pieces of steel sheets) between the cutting tool and seating surface.

Step 8: Check for approach angle to avoid rubbing of machine surface by the tool.

Steps for Facing

Sailesh excited as he thinks that he may be able to do facing himself.

Mohan: Sailesh now, I will tell you the steps that you need to perform in facing operations:

Step 1: Clamp the work piece on the chuck.

Step 2: Turn on the main switch to rotate the spindle and check for trueness

Step 3: Mount the cutting tool in the tool post and check for the center height.

Step 4: Select the required spindle speed by changing the belt position in the pulley or by changing the gear shifter lever.

Step 5: Move the slide closer to the starting position of the work piece.

Step 6: Touch the tool on the work piece face and note the reading on the dial mounted on the hand wheel of longitudinal slide.

Step 7: Pull out the tool away from work piece by moving the longitudinal slide.

Step 8: Move the cross slide to operator's side to a convenient position for start facing.

Step 9: Adjust the longitudinal slide by adding needed depth of cut.

Step 10: Switch ON the spindle.

Step 11: Move the cross slide towards the center for face turning the work piece by rotating hand wheel of cross slide, up to the center till the disappearance of any pip.

Step 12: Pull out the tool away from the work piece by moving the longitudinal slide by rotating the hand wheel.

Step 13: Move the apron to right side to a convenient position by rotating the hand wheel.

Step 14: Stop the spindle and measure the machined length/ thickness using micrometer or vernier calliper.

Step 15: Compare present machined length/ thickness and the drawing dimension for deciding further steps of cut.

Step 16: Repeat step 11 to 15 till correct dimensions are achieved.

Mohan: Sailesh will you be able to demonstrate these steps to your co-workers?

Sailesh: Yes, I have understood the procedure and I will surely be able to assist anyone who needs my help!

Defects Found on Faced Work

Mohan tells Sailesh that the steps for facing can help him in doing facing work but sometimes faced work has some defects. It is important to find defects on faced work.

Mohan: Sailesh, it's time to teach you how to find defects on faced work.

- **Concave face:**

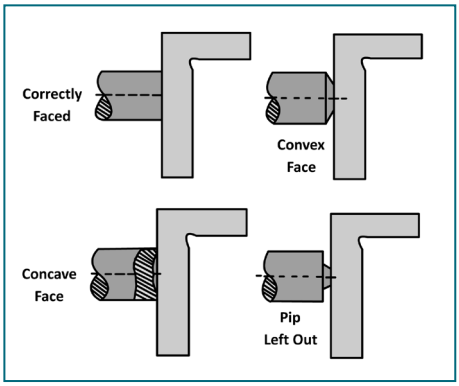
The cause for this defect is the tool digging into the work during the feeding because the tool is not clamped rigidly. By clamping the tool rigidly with a minimum overhang, this defect may be avoided.

- **A Convex Face:**

The cause for this defect is the tool's cutting edge being blunt or getting blunt during its travel and the carriage not being locked. To avoid this defect you must re-sharpen the tool and use it. Also, lock the carriage to the bed of the lathev.

- **A pip in the center:**

This is due to the tool not being set to the centre height, and this defect can be avoided by placing the tool to the centre height.



Tri- Square

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. What is facing and give the steps for facing.

2. How will you know the defects of facing work?

3. Write the steps for tool settings.

Worksheet

1. *Help Sailesh differentiate between the true and false statements. Write "True" against the true statements and "false" against the false ones.*

   a. The Tool settings are same in both outer diameter and facing. _____

   b. Like OD diameter you need to do the truing job similarly in facing too. _____

   c. The cause for concave face is the tool digging into the work during the feeding because the tool is not clamped rigidly. _____

   d. A pip in the center is commonly seen because the tool is not set to the centre height, and this defect can be avoided by placing the tool to the centre height. _____

   e. In case of facing, you need to choose a cutting tool that does not have- a rounded tip. _____

2. *Sailesh is giving instructions to his juniors but he has missed a few words so help him by filling the correct word in the blank:*

a. The facing operations are undertaken with the help of _____.

b. The work piece should not extend more than _____ its diameter from the chuck jaws unless a steady rest is used to support the free end.

c. Facing helps you to maintain the _____ of the work piece.

d. Facing helps you to remove the _____ and on the faces of the work.

e. In order to avoid _____ defect you must re-sharpen the tool and use it.

Notes

Answers:

1.

a.	true.	b.	true.	c.	true.
d.	true.	e.	false.		

2.

a.	Facing tool.	b.	2-3 times.	c.	Length
d.	Rough surface	e.	Convex face.		

Turning Between Centers

At the end of this module you will be able to:

- perform turning between centers.

Session Plan	
1	Turning Between Centre in Lathe Machine
2	Types of Lathe Center
3	Lathe Dog and Drive Plate
4	Procedure of Turning Between Centers
5	Key Learnings
6	Worksheet

Turning Between Centre in Lathe Machine

Sailesh is curious to learn about how to turn between centers in lathe machine. He also wants to know how a work piece is to be transferred between machining operations without any loss of accuracy. He asks Mohan:

Sailesh: What is a lathe center?

Mohan: It's very simple! A lathe center, often shortened to center, is a tool that has been ground to a point to accurately position a work piece on an axis. The primary use of a center is to ensure concentric work is produced; this allows the work piece to be transferred between machining operations without any loss of accuracy.

Sailesh: Okay, I got that! But what is turned between the centres?

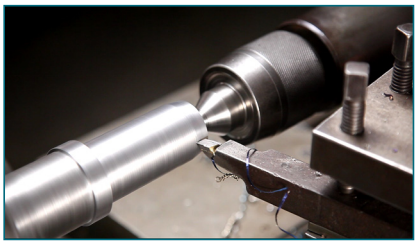
Mohan: See, the shaft type work pieces which require geometrical accuracy on longer lengths are turned holding them between centers fixed in headstock and tailstock. Rotating the work piece positively is done through a driving plate fixed in the headstock and a removable carrier fixed on the work piece connecting the work piece and the spindle.

Sailesh: Oh I see... But what is this center used for?

Mohan: A center is used to support longer work piece where the cutting forces would deflect the work excessively, reducing the finish and accuracy of the work piece or creating a hazardous situation. A center has applications anywhere that a centered work piece may be used; this is not limited to lathe usage but may include setups in dividing heads, cylindrical grinders, tool and cutter grinder or other related equipment. The term between centers refers to any machining operation where the job needs to be performed using centers.

Sailesh: Alright! Now I am clear about it. But, how is a center inserted into a matching hole?

Mohan: A center is inserted into a matching hole on the work piece drilled by a center drill.



Types of Lathe Center

Sailesh: Now, I have understood the procedure of turning between centers in a lathe machine and I am going to learn about the different types of lathe centers.

Mohan: Okay, it's really simple.

First is Dead Center.

A dead center (one that does not turn freely) may be used to support the work piece at either the fixed or rotating end of the machine. When used in the fixed position, a dead center produces friction between the work piece and center, due to the rotation of the work piece. Lubrication is therefore between the center and work piece to prevent friction. Additionally the tip of the centre may have an inset of cemented carbide which will reduce the friction slightly and allow for faster speeds.

Sailesh: Okay, but why are these centers used in a lathe machine?

Mohan: Well, these centers are typically fully hardened to prevent damage to the important mating surface of the taper and to preserve the 60 degree angle of the nose. As tungsten carbide is much harder than steel carbide tipped center; it has greater wear resistance than a solid steel center.

Second is a ball tipped center.

This ordinary center is the type of center used for most general work. In the ball tipped center, the ball point consists of a hard alloy tip brazed into an ordinary steel shank.

Sailesh: How are these centers different from dead centers?

Mohan: The ball tipped center is more expensive types of center, one which will give excellent service against wear and strain. Another important trait of these centers is that they are used to minimize wear and strain on the ordinary center except that little less than half of the center has been ground away.

Mohan: Shailesh , let me now tell you about the live or revolving center.

A live center or revolving center is constructed so that the 60° center runs in its own bearings and is used at the non-driven or tailstock end of a machine. It allows higher turning speeds without the need for separate lubrication, and also greater clamping pressures.

Sailesh: Okay, but where is this live center or revolving center, used?

Mohan: CNC lathes use this type of center almost exclusively and they may be used for general machining operations as well. Spring-loaded live centers are designed to compensate for center variations, without damage to the work piece or center tip. This assures the operator of uniform constant tension while machining. Some live centers also have interchangeable shafts. This is valuable when situations require a design other than a 60° male tip.

I Hope you are clear about all the lathe centers I have explained so far. Now let me tell you about a pipe center.

A pipe center, also known as a bull nose center is a type of live center which has a large diameter conical nose rather than a sharp point. This allows the center to be used in the bore of pipe or other work piece with a large interior diameter.

Sailesh: Is there any important advantage of using the pipe centre?

Mohan: The main advantage of a pipe center is that it supports the work piece securely, and can be used for parts whose larger inner diameter prevents the use of a normal pointed center.

Thin-walled material such as pipes easily collapses if excessive force is used at the chuck end.

Mohan: Now, let us look at the cup center, which is mainly of two types.

The woodworking variety is a variation of the traditional live center. This type of cup center has a central point like a normal live center and also has a ring surrounding it. The ring supports the softer material around the center point and prevents the wood from splitting under pressure from the central point.

A different variety of cup center is used for metalworking. The metalworking variety of cup center has a tapered hole rather than a conical point. It supports the part by making contact with the outside diameter of the end of the part, rather than using a center hole.

Mohan: Next is a drive center which is used in the driving end of the machine (headstock). It consists of a dead center surrounded by hardened teeth. These teeth bite into the softer work piece allowing the work piece to be driven directly by the center.

This allows the full diameter of the work piece to be machined in a single operation, this contrasts with the usual requirement where a carrier is attached to the work piece at the driven end.

Sailesh: I have a query.

Mohan: Please feel free to ask.

Sailesh: Where are these centers used?

Mohan: They are often used in woodworking or where softer materials are machined. Drive centers are also known as grip centers in some industrial circles.

I Hope you are now clear about the drive center and we can proceed towards the spring center.

A spring center is a metalworking lathe center for maintaining a cutting tool like a reamer or a tap, in axial alignment with a hole being worked on. It consists of a point backed by a spring to push the cutting tool into the work piece.

Lathe Dog and Drive Plate

Mohan: Now we will talk about the lathe dog and drive plate.

Sailesh: Lathe dog and drive plate, really?

Mohan: Yes!

Sailesh: That sounds interesting.

Mohan: A lathe dog is a device that clamps around the work piece and allows the rotary motion of the machine's spindle to be transmitted to the work piece. It is also known as a lathe carrier and is most often used when turning between centers on a lathe but it may be used on dividing heads or any similar situation.

Sailesh: What is it used for?

Mohan: It is used in conjunction with a drive plate and drive pins. The plate is mounted directly on the machine spindle (with a chuck) and the drive pin is attached to the plate, in use the carrier and work piece are inserted between enters and the leg of the carrier rests against the drive pin.

Mohan: You know these carriages may be of the straight leg or bent leg type. The straight leg requires the drive pin; the bent leg fits into a slot machined into the drive plate. The bent leg type is considered safer as there are fewer protruding parts to cause accidents.

Spindle speeds are reduced when working with carriers, due to the unbalanced nature of the setup. Care must also be taken by the operator when using carriers, as it is easy to get snagged on one.

The drive plate mounts on the spindle nose and is used in conjunction with the driving dog to drive workpiece between centers.

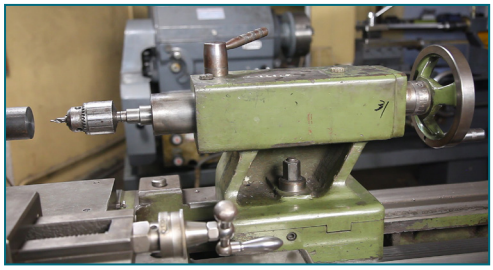
It is a circular disc made out of cast iron having a slot at its face. The slot is there to accommodate a pin which engages with the tail of lathe dog or carrier when a job is held in between center. When a bent tail dog is used the bent portion of the tail is inserted into the hole which serves the same purpose. If a straight tail lathe carrier is used then a pin has to be mounted over the drive plate.

Procedure of Turning Between Centers

Mohan: Sailesh, now we will learn the complete procedure of turning between the centers.

Sailesh: Okay.

Mohan: The step one is to munt the driving plate and center on the headstock.



1. Mount the center on the tailstock.

Next, step two (II) is to mount the carrier on to the shaft (outside the machine)

In step three (III), place the assembly locating the carrier at the correct position in the driving plate and work piece center hole resting on the headstock center; hold this assembly.

Then, rotate the tailstock hand wheel for positioning the center on the center hole of the work piece.

One you have performed the first three steps, you can perform OD turning and facing operation as per the procedure to drawing dimensions.

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. What is turning between centers?

2. Describe any two types of lathe centers.

3. What is the function of lathe dog and drive plate?

Worksheet

1. *Read the following questions and choose the correct answer.*

1. Which of the following is called the bull nose center?
   a. Ball tipped center
   b. Pipe center
   c. Spring center
   d. None of the above

2. Which of the following device clamps around the work piece and allows the rotary motion of the machine's spindle to be transmitted to the work piece?
   a. Lathe carrier
   b. Dead center
   c. Both a and b
   d. Neither a nor b

3. Which of the following lathe centre rotates with the work piece?
   a. Dead center
   b. Pipe center
   c. Live center
   d. Cup center

4. Which type of lathe centre provides excellent service against wear and strain?

   a. Ball tipped Center

   b. drive center

   c. Dead center

   d. Live or revolving center

5. A device that clamps around the work piece and allows the rotary motion of the machine's spindle to be transmitted to the work piece.

   a. Lathe center

   b. Lathe dog

   c. Drive plate

   d. Turning point

Notes

Answers:

1.

1. b
2. a
3. c
4. a
5. b

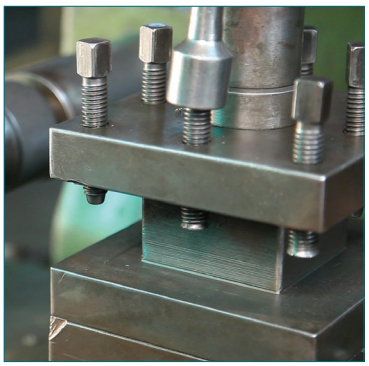
Taper Turning

At the end of this module you will be able to:

- perform the operation of taper turning.

Session Plan	
1	Introduction to Taper Turning
2	Types of Tapers
3	Taper Turning Methods
4	Taper Turning by Swiveling Compound Slide
5	Taper Turning by Form Tool
6	Taper Turning by Tailstock Off-set Method
7	Taper Turning Using Taper Turning Attachment
8	Key Learnings
9	Worksheet

Introduction to Taper Turning



Sailesh is curious to learn about a taper and how it works. He asks Manish.

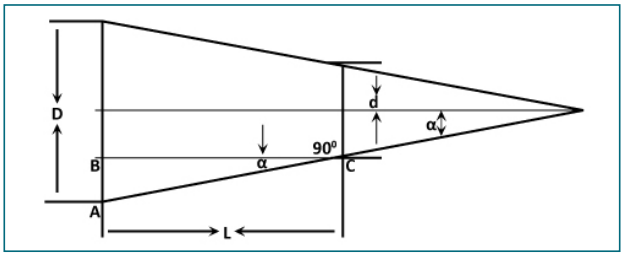
Sailesh: What is a taper?

Mohan: A uniform increase or decrease in dimension of the work piece along its length from one end to another is called taper.

Sailesh: Okay! Tell me more about it.

Mohan: The taper elements are generally designated by a number of symbols Including:

Taper Element:



D = large diameter of taper in mm

d = small diameter of taper in mm

l = length of tapered part in mm

2α = Full taper angle

α = angle of taper or half taper angle

Sailesh: But, how is the amount of taper in a work piece usually specified?

Mohan: The amount of taper in a work piece is usually specified by the ratio of the difference in diameters of the taper to its length. This is termed as the conicity and it's designated by the letter K.

K = D – d/ L

Sailesh: Okay. Can you give me an example?

Mohan: Yes.

Example1: In the above figure

Let D = 100 mm, d = 90 mm, and l = 100 mm, find the value of K.

K = D – d/ L

K = (100 – 90)/ 100 = 10/00 = 1/10

This 1/10 means that the amount of the taper is reduced by 1 mm. The amount of taper

1/10 may also be expressed as a decimal i.e. 0.1. The equation may be rearranged in various ways to calculate any one of the unknown quantities.

L = D – d/ K

If the large diameter, the small diameter and the conicity are known the length of the taper can be calculated.

Types of Tapers

Mohan: I hope you have understood the working of tapers. Let me tell you about the different types of tapers.

Sailesh: Yes, I would certainly like to know more about the different types of tapers.

Mohan: Standard taper is the first and most basic type of taper.. The machine parts and tools having inside or outside taper are standardized to facilitate interchangeability of parts. Tapered surfaces which follow standard dimensions are called standard tapers.

Sailesh: But why are these tapers called standard tapers?

Mohan: Good question Sailesh. Standard tapers adapted by the bureau of Indian standard for various tools and machine parts like drills, reamers, milling cutter shanks, arbors, lathe centers, etc. are Morse tapers. Morse tapers are available in seven sizes numbered: 0, 1, 2,3,4,5 and 6.

The amount of taper varies from number to number. The no.0 (zero) Morse taper is the smallest while No. 6 is the largest in size.

Mohan: But there is a disadvantage associated with standard tapers because of the non-uniformity of the angle of the taper for different Morse taper sizes.

Sailesh: Okay, And what are the other types of tapers.

Mohan: Okay! You know what? Metric tapers are sometimes used as standard tapers. Metric tapers are made is seven sizes and designated by the number 4.6, 80, 100, 120, 160 and 200. The taper number standards for the large diameter of the taper in mm.

Sailesh: Okay!

Mohan: Do you know what is the main advantage of the metric tapers?

Sailesh: No.

Mohan: All metric tapers have the same angle of taper.

The amount of taper and taper angle for standard tapers is as follows:

Standard Tapers	Amount of Taper or Conicity	Half Taper Angle	Full Taper Angle
0	1:19.212	1029'27''	2058'54"
1	1:20.047	1025'43''	2051'26"
2	1:20.020	1025'50''	2051'41"
3	1:19.922	1026'16''	2052'32"
4	1:19.254	1029'15''	2058'31"
5	1:19.002	1030'26''	3000'53"
6	1:19.180	1029'36''	2059'12"
Metric tapers: No.s 4,6,80,100,120,160,200	1:20	1025'26"	2051'51"

Mohan: You know Sailesh, there is something really interesting about the standard tapers. In the British system, in addition to Morse standard tapers there are two other important standard tapers:

1. Brown and Sharpe standard taper
2. Jarno standard taper

Sailesh: Is there any difference between the usage of these tapers?

Mohan: Yes there is. The Brown and Sharpe standard taper is used mostly on milling machines. There are 18 sizes in the series, numbered from 1 to 18. In this standard, the taper is 0.500 in per ft in all of its numbers except the no. 10 which has a taper of 0.5161 in per ft.

The Jarno system of taper is the most sensible system. This was originally designed for use in lathes. There are 20 sizes in the series, each one being identified by a number ranging from 1 to 20 and the taper is 0.60 in per ft in each size.

Taper Turning Methods

Mohan: Now, let us understand the working of the taper turning method.

Sailesh: Okay, But what is this taper turning?

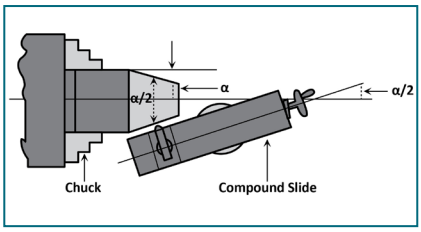
Mohan: A taper turning is done on a lathe by different methods. There are various types of methods to make a part tapered.

- Taper turning by swiveling compound slide
- Taper turning by form tool
- Taper turning by tailstock off set method
- Taper turning by using taper turning attachment
- Taper turning by combination feed movement

Taper Turning by Swiveling Compound Slide

Mohan: Let's begin with the taper turning by swiveling compound slide.

This method employs the principle of turning taper by rotating the workpiece on the lathe axis and feeding the tool at an angle to the axis of rotation of the workpiece. The tool mounted on the compound rest which is a attachment to a circular base, graduated in degree, which may be swiveled and clamped at any desired angle.



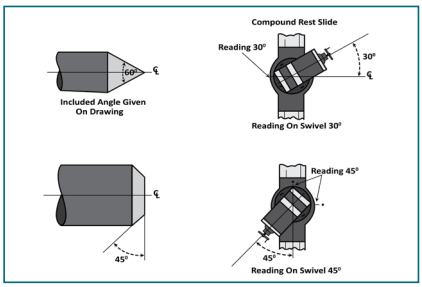
Once the compound rest is set at the desired half taper angle, rotation of the compound slide screw will cause the tool to be fed at that angle and generate a corresponding taper. This method is limited to turn a short taper owing to the limited movement of the compound rest may be swiveled at 450 on either side of the lathe axis enabling it to turn a steep taper. The movement of the tool in this method is purely controlled by hand, this gives a low production capacity and poorer surface finish.

The setting of the compound rest is done by swirling the rest at the half taper angle, if this is already known. If the diameter of the small and large end and length of taper are known, the half angle can be calculated from the following equation:

$$\text{Tan } (\alpha/2) = (D-d)/2L$$

Sailesh: How will I learn to cut taper using the compound site?

Mohan: The procedure to cut taper using compound slide is very easy and simple. It involves the following steps:

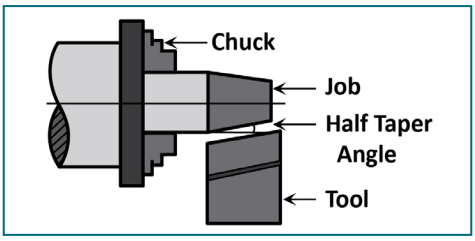


1. Refer to the drawing for amount of taper required in degrees.
2. Loosen compound rest lock screws.
3. Swivel compound rest as follows.
4. Tighten compound rest lock screws.
5. Set cutting tool to center with tool holder at right angles to taper to be cut.
6. Tighten tool post securely.
7. Back off top slide of compound rest so there will be enough travel to machine length of taper.
8. Move carriage to position cutting tool near start of taper and then lock carriage.
9. Rough turn taper by feeding cutting tool using compound rest feed handle.
10. Check taper for accuracy and readjust compound rest setting if necessary.
11. Finish-turn and check taper for size and fit.

Taper Turning by Form Tool

Mohan: Next, I will tell you about the taper turning form tool.

You know this form tool has a cutting edge formed at the desired angle. The form tool is a replica of the shape (Taper angle) to be produced on the work piece. The form tool is fed against a rotating work piece; the shape of the tool is reproduced on the work piece.



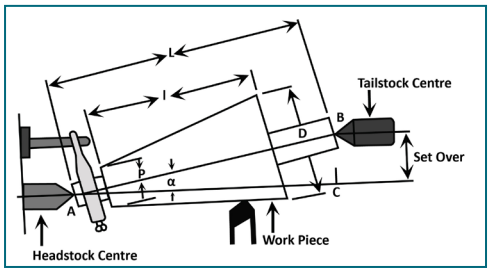
Sailesh: Can the form tool method be used anywhere?

Mohan: No, it is restricted only for short length tapers. In this case we can plunge or give feed to the tool at fixed position where taper is required as shown in the figure.

Taper Turning by Tailstock Off-set Method

Mohan: We will now learn about taper turning by tailstock off-set method.

Sailesh: Okay, please explain me about it.



Sailesh: In a lathe the live center and dead center both lies on the same axis. The work is held between the live and dead center. The dead center (tailstock) axis is shifted to the required length on the lathe bed to get taper on the work piece. In tailstock off-set method small taper angle is obtained for longer length workpiece. The workpiece is held between the centers and the carriage is moved on the bed. The tool traces the lathe axis but the workpiece is offset by an angle.

Hence, the taper is generated on the workpiece. In this method we can cut taper upto 140.

Tailstock Offset, X = [(D-d)L]/2l

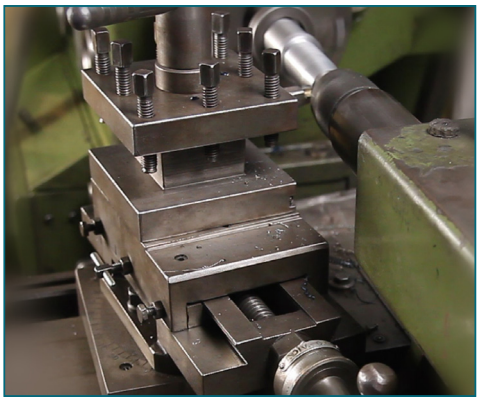
X = Setover, D = Large diameter, d = Small diameter, L = Length of work piece, l = Length of Taper

Procedure for offsetting the tailstock:

1. Adjust tailstock spindle distance; it will be used in matching setup and lock tailstock spindle clamp.
2. Mount a dial indicator in tool post with plunger in horizontal position and on center.
3. Using cross feed handle, move indicator so register 0.020 in on work and set indicator and cross feed graduated collar to zero.
4. Loosen tailstock clamp nut.
5. With tailstock adjusting setscrews, move tailstock until required offset is shown on dial indicator.
6. Tighten tailstock setscrew that was loosened, making sure indicator reading does not change.
7. Tighten tailstock clamp nut.

Note: Tailstock may also be offset by using feeler gauge between tool post and tailstock.

Taper Turning Using Taper Turning Attachment



Mohan: Next, let us look at the taper turning using taper turning attachment. You know a taper turning attachment is fitted behind the lathe bed. There is a guide bar which guides the tool based on the taper required on the work piece. The guide bar can be swiveled according to taper angle. The cross slide is made from the lead screw by removing the binder screw. In this we can make taper maximum upto 200.

The complete procedure to cut taper using taper attachment:

1. Clean and oil guide bar.
2. Loosen lock screws and offset end of guide bar the required amount or set bar to required taper in degrees.
3. Tighten lock screws.
4. With compound rest set at 900, setup cutting tool on the center.
5. Set work piece in lathe and mark length of the taper.
6. Tighten connecting screw on sliding block.
7. Move the carriage until the center of attachment opposite length be tapered.
8. Lock anchor bracket to lathe bed.
9. Take cut 15 mins long, stop lathe, check end of taper size.
10. Set depth of roughing cut to 0.5 to 0.6mm oversize and machine taper.
11. Readjust taper attachment, if necessary, take light cut and recheck taper fit.
12. Finish turn and fit taper to gauge.

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. What is taper turning?

2. Explain the procedure of taper turning using swiveled compound slide.

3. Explain the procedure of taper turning using tailstock offset method.

4. Explain the procedure of taper turning using taper attachment.

Worksheet

1. Read the following questions and choose the correct answer.

1. Which of the following is not a type of taper?

   a. Morse taper ☐

   b. Jarno standard taper ☐

   c. Brown standard taper ☐

   d. None of the above ☐

2. Calculate the tailstock offset value based on the following data:-

   Large diameter = 100 mm, Small diameter = 90 mm, Length of work piece = 100 mm, Length of taper = 50 mm

   a. 50 ☐

   b. 100 ☐

   c. 10 ☐

   d. 5 ☐

3. Calculate the conicity based on the following data:-

Large diameter = 200 mm, Small diameter = 180 mm, Length of taper = 100 mm,

a. 1/10 ☐

b. 1/5 ☐

c. 2/3 ☐

d. ¼ ☐

Notes

Answers:

1.

1. d.
2. c.
3. b.

Drilling

At the end of this module you will be able to:

- perform drilling operation.

Session Plan	
1	Drilling
2	Center Drilling
3	Centre Drilling in Lathe
4	Drill Bit
5	Using the Tailstock for Drilling
6	Key Learnings
7	Worksheet

Drilling

Today, Sailesh is curious to learn about drilling process and its use. He asks Mohan:

Sailesh: Hi, Can you please tell me something about drilling?

Mohan: Yes. It is very simple. Drilling is the process to produce a hole in a solid work piece using a cutting tool called drill. Drill has two lips having ground geometries like cutting angles in a single point cutting tool. Work piece held on the chuck is rotated and the drill mounted on the tailstock is moved by rotating the tailstock handle to produce the hole in the center of the work piece.

Sailesh: What are these drills made of?

Mohan: The drills are made up of HSS tools. The holding portion is straight or tapered. Straight drills are called parallel shank drills. The Tapered shanks are provided with Morse Taper with varying diameters.

Sailesh: That's interesting. Please tell me more about these drills.

Mohan: Parallel Shank Drills are clamped using a drill chuck with key and inserted into the taper socket provided in the tailstock. Taper shank drills are either inserted directly into the tail stock or using intermediary Morse Taper bushes.

Speeds and feeds are selected based on the work piece material, tool material and length of the drill hole on the work piece.

Center Drilling

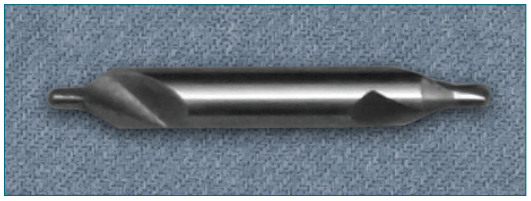
Mohan: Sailesh, now that you know what t drilling is, let me elaborate on an operation called centre drilling.

Sailesh: That's going to be really interesting. Please go ahead.

Mohan: Centre drilling is an operation through which small drill holes are produced to locate the centre of the hole on the solid flat surface by means of a cutting tool known as combination drill or center drill.

Sailesh: Okay. But, when is this operation performed?

Mohan: When the hole is longer in the work piece; center drilling is done before drilling. Centre drilling creates small conical centre which aids in locating the drill in subsequent drilling operation and avoid wobbling of drill during entry. Work piece held on the chuck is rotated and the drill mounted on the tailstock is moved by rotating the tailstock handle to produce the hole in the center of the work piece.



Sailesh: But, What facilitates these holes?

Mohan: That's a really good question. I will tell you about it. The alignment between the headstock and tailstock of the lathe enables you to drill holes that are precisely centered in a cylindrical piece of stock.

Before you drill into the end of a work piece you should first face the end as described in the facing operations section. The next step is to start the drill hole using a center drill – a stiff, stubby drill with a short tip. If you try to drill a hole without center drilling first, the drill will almost certainly wander off the center, producing a hole that is oversized and misaligned.

Centre Drill:

Sailesh: How is this hole made?

Mohan: Centre drill is the tool which is used to make a small hole to locate the centre for the proceeding drill holes. The material of centre drill used in the workshop is HSS. Centre and spotting drills are traditionally used to make a conical hole for a lathe centre in order to make a starting point for a standard twist drill, designed to prevent the twist drill from walking.

Sailesh: Mohan, I have another question for you.

Mohan: Please ask.

Sailesh: What kind of material is used to make drills?

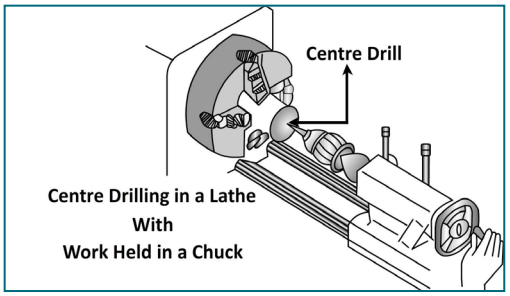
Mohan: Drill materials are HSS, HSS – Co, TiALN coating and carbide. Centre Drills are available with: Combined drills, reinforced bulge, radius form, etc.

Centre Drilling in Lathe

Mohan: Sailesh, I hope now you are clear about the concept of centre drilling, so let me tell you how centre drilling is done in lathe machine.

But, before that let me tell you that round work pieces can be quickly and accurately centre- drilled without the necessity of centre punch marks.

Sailesh: Please tell me more about it.



Mohan: To perform the operation of centre drilling the steps are:

Step 1: Hold the work in a four jaw chuck about 50 mm outside and true.

Step 2: Finish faces the work with a facing tool.

Step 3: Ensure no 'pip' is left out in the centre and the face is at right angles to the axis.

Step 4: Mount the drill chuck in the tailstock spindle.

Step 5: Remove dirt on the taper shank of the chuck and the tailstock spindle taper bore.

Step 6: Mount a suitable centre drill securely in the drill chuck. Set the lathe to about 1000 r.p.m.

Step 7: Slide the tailstock over the bed until the centre drill is close to the work face.

Step 8: Lock the tailstock in this position.

Step 9: Start the machine and slowly feed the centre drill into the work by rotating the tailstock hand wheel.

Step 10: Withdraw the drill frequently from the work piece to clean the chips and to apply the cutting fluid.

Step 11: Continue drilling until about three fourths of the tapered portion of the centre drill has entered the work.

Step 12: Ensure that uniform continuous pressure is applied during feeding and no extra force is given.

Drill Bit

Mohan: Now we will move on to the next thing that is drill bit.

Sailesh: It is a tool or a process?

Mohan: Drill bits are cutting tools used to create cylindrical holes, almost always of circular cross – section. Bits are held in a tool called drill, which rotates hem and provides torque and axial force to create the hole.

And interestingly, the shank is the part of the drill bit grasped by the chuck of the drill. The cutting edges of the drill bit are at one end, and the shank is at other.

Sailesh: Please tell me more about drill bits.

Mohan: You know what? Drill bit geometry has several aspects.

Sailesh: Like what?

Mohan: The Spiral (or rate of twist) in the drill bit controls the rate of chip removal. A fast spiral drill bit is used in high feed rate applications under low spindle speeds, where removal of a large volume of swarf is required. Low spiral drill bits are used in cutting applications where high cutting speeds are traditionally used, and where the material has a tendency to gall on the bit or otherwise clog the hole, such as aluminum or copper.

Next is Point Angle, or the angle formed at the tip of the bit, is determined by the material the bit will be operating in. Harder materials require a larger point angle, and softer materials require a sharper angle. The correct point angle for the hardness of the material controls wandering, chatter, hole shape, wear rate, and other characteristics.

The Lip Angle determines the amount of support provided to the cutting edge. A greater lip angle will cause the bit to cut more aggressively under the same amount of point pressure as a bit with a smaller lip angle. Both conditions can cause binding, wear and eventual catastrophic failure of the tool.

Sailesh: Then, how is this lip angle determined?

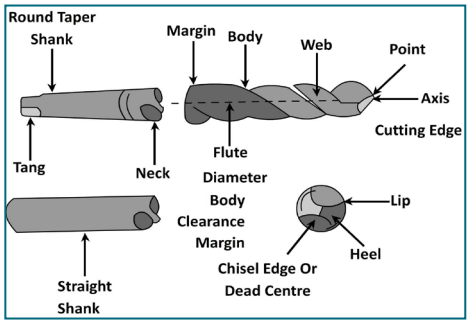
Mohan: The proper amount of lip clearance is determined by the point angle. A very acute point angle has more web surface area presented to the work at any one time, requiring an aggressive lip angle, where a flat bit is extremely sensitive to small changes in lip angle due to the small surface area supporting the cutting edges.

The Length of a bit determines how long a hole can be drilled, and also determines the stiffness of the bit and accuracy of the resultant hole. Twist drill bits are available in standard lengths, referred to as Stub-length or Screw-Machine-length (short), the extremely common Jobber-length (medium), and Taper-length or Long-Series (long).

Most drill bits for consumer use have straight shanks. For heavy duty drilling in industry, bits with tapered shanks are sometimes used.

The diameter-to-length ratio of the drill bit is usually between 1:1 and 1:10. Much higher ratios are possible (e.g., "aircraft-length" twist bits, pressured-oil gun drill bits, etc.), but the higher the ratio, the greater the technical challenge of producing good work.

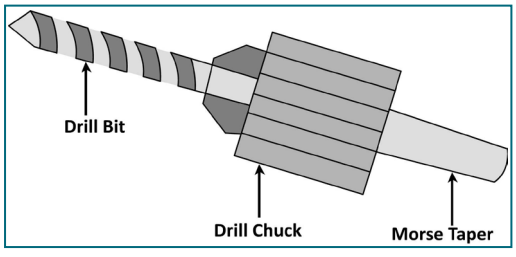
Drills bits come in standard sizes. Speeds and feeds are selected based on the work piece material, tool material and length of the drill hole on the work piece.



Cutting Speed for Drilling in m per Minutes

	HSS Drills
Soft cast iron	30 – 45
Medium cast iron	21 – 30
Malleable iron	24 – 27
Mild steel	24 – 45
Stainless steel	18 – 21
Aluminium and alloys	60 – 90
Brass and bronze	60 – 90
Copper	18 - 30

Using the Tailstock for Drilling



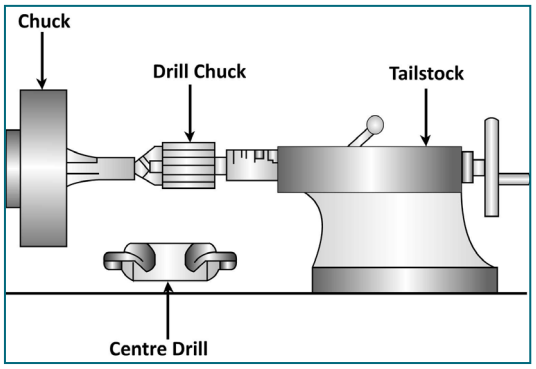
Mohan: Now, I will tell you how the tailstock is used for drilling.

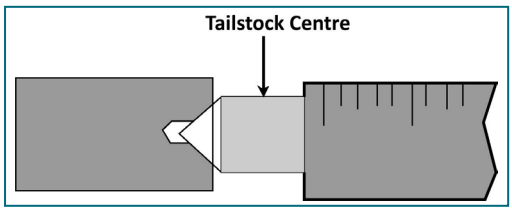
Sailesh: Okay.

Mohan: Work piece held on the chuck is rotated and the drill mounted on the tailstock is moved by rotating the tailstock handle to produce the hole in the center of the work piece.

The usual starting point for drilling with a centre lathe is to use a countersink bit. This is used to drill slightly into the material and creates a starting point for other drills that are going to be used.

Attempting to drill with a traditional drill bit without countersinking first will lead to the drill bit slipping straight away. It is not possible to drill a hole successfully or safely without using a centre drill first.

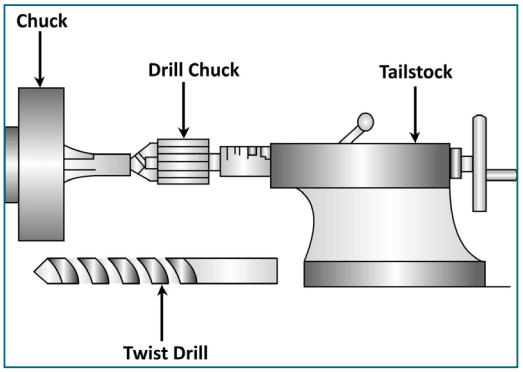




If a long piece of material has to be turned on a lathe then a centre drill is used to produce the hole at one end. This allows the drilled end to be supported by the tailstock centre.

Once a hole has been produced by a centre drill, machine twist drills can be used to enlarge the hole and if necessary to drill all the way through. If a large diameter hole is needed then a small hole is drilled first (eg. 4mm dia). Then the hole is enlarged approximately 2mm at a time.

Trying to drill a large diameter hole in one go will inevitably lead to the drill bit over heating and then jamming in the material. This is potentially dangerous.



Sailesh: Then, how can we overcome the problem of over-heating?

Mohan: When drilling, it is very important to use soluble oil as a coolant. This should be constantly fed onto the drill bit to keep it cool. This will help prevent jamming and overheating. Over heating will blunt the drill bit quickly.

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. What is drilling?

2. State the importance of centre drilling

3. Explain the geometry of Tool bit.

4. Explain the pocedure of drilling a metal block.

Worksheet

1. Read the following questions and choose the correct answer.

1. While drilling in lathe, the drill is held in the:

   a. Headstock ☐
   b. Tailstock ☐
   c. Bed ☐
   d. Tool post ☐

2. The process of producing a small drill hole on the face of the metal to locate the centre of the hole, before drilling is called _____.

   a. Boring ☐
   b. Bit Drilling ☐
   c. Center Drilling ☐
   d. None of the above ☐

3. Taper end of the drill bits are called _____.

   a. Bit taper ☐
   b. Pin taper ☐
   c. Sharpe taper ☐
   d. Morse taper ☐

4. The angle formed at the tip of the drill bit is called?

   a. Point angle ☐
   b. Spiral drill bits ☐
   c. Hole ☐
   d. Lip angle ☐

5. What are straight drills called?

   a. Tapered shanks ☐
   b. Centre drill ☐
   c. Bit drills ☐
   d. Parallel shank drills ☐

Notes

Answers:

1.

1. b.
2. c.
3. d.
4. a.
5. d.

Boring

At the end of this module you will be able to:

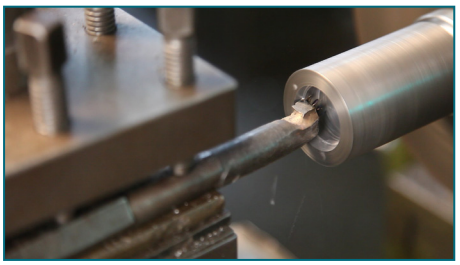
- perform the boring operation.

Session Plan	
1	Boring
2	Mounting Workpiece for Boring
3	Boring Operation
4	Key Learnings
5	Worksheet

Boring

Mohan: Today I will explain the process of boring in machining.

Sailesh: That's great.



Mohan: In machining, boring is the process of enlarging a hole that has already been drilled (or cast), by means of a single-point cutting tool (or of a boring head containing several such tools).

For example, boring a cannon barrel.

Sailesh: Why is boring done?

Mohan: Boring is used to achieve greater accuracy of the diameter of a hole, and can be used to cut a tapered hole. Boring can be viewed as the internal-diameter counterpart to turning, which cuts external diameters.

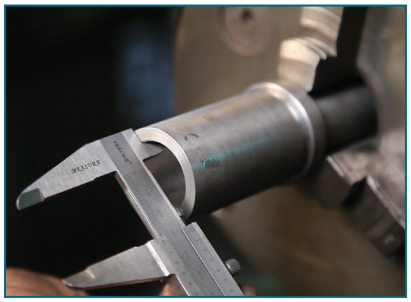
Sailesh: Are there different types of borings?

Mohan: Yes, there are various types of boring. The boring bar may be supported on both ends (which only works if the existing hole is a through hole), or it may be supported at one end. Line implies the former. Back boring is the process of reaching through an existing hole and then boring on the "back" side of the work piece (relative to the machine headstock).

Sailesh: What is the main purpose of boring?

Mohan: Boring is necessary in many cases to produce accurate holes. Drilled holes are seldom straight due to imperfections in the material which cause drills to move out of alignment. Therefore, where accuracy is important, drilled holes are usually made undersize and then bored or reamed to the proper dimensions.

Sailesh did you know that, boring is also useful in truing large holes in flat material. In this case, the hole is cut undersize using a band saw or trepanning tool and is trued to proper dimensions by boring.



The boring tool bar should be clamped as close to the holder and tool post as possible considering the depth of boring to be done. The bar will have a tendency to spring away from the work piece if the bars over hangs the tools post too far. If deep boring is to be performed, it will be necessary that the bar is as thick as possible to counteract this springing tendency.

Boring Cutter Bit Setup

Sailesh: But how exactly is boring is done? I mean what are the tools used?

Mohan: The cutter bit used for boring is similar to that used for external turning on the lathe. The bit is usually held in a soft or semisoft bar called a boring tool bar. The boring tool bar is supported by a cutting holder which fits into the lathe tool post. Boring tool bars are supplied in several types and sizes for holding different cutter bits.

Sailesh: Okay!

Mohan: Do you understand angles?

Sailesh: Yes.

Mohan: So, this bit is supported in the boring tool bar at a 90°, 30°, or 45° angle, depending upon the nature of the work piece being bored. Most general boring is accomplished with a 90 degree cutter bit. The bit is mounted at a 30° or 45° angle to the axis of the boring tool bar when it is necessary to cut up to the bottom of a hole or finish the side of an internal shoulder.

It is desirable that the boring tool bar be as large as possible without interfering with the walls of the hole. The cutter bit should not extend far beyond the boring tool bar and the bit securely in the bar; yet not have the shank end protrude far from the bar.

The cutter bits that are used for boring are shaped like left-hand turning and facing cutter bits. Greater attention must be given to the end clearance angle and the back rake angle because of the curvature of the hole.

Mounting Workpiece for Boring

Mohan: Now, I will tell you about mounting workpiece for boring

Sailesh: Is it a difficult task? How is it done?

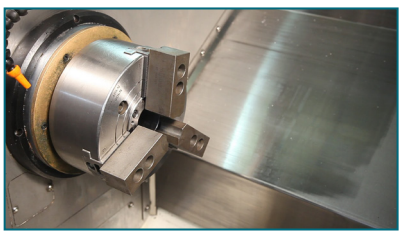
Mohan: Boring is the enlarging and truing of a hole by removing material from internal surfaces with a single point cutter bit. On the lathe, boring is accomplished by either of these two methods:

Mounting the holder and boring tool bar with cutter bit on the tool post and revolving the work piece.

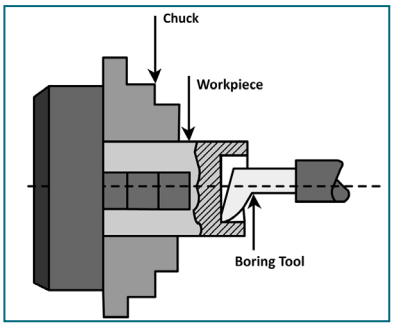
Mounting the work piece in a fixed position to the carriage and revolving the boring tool bar and cutter bit in a chuck attached to the headstock spindle. (This is a special process and not used in most machine shops).

The work piece may be supported in a chuck or fastened to a faceplate for boring operations depending upon the material to be machined.

When boring is to be performed on the ends of long stock, the work piece is mounted in a chuck and a steady rest is used to support the right end near the cutter bit. Some boring operations require the use of special chuck-mounted mandrels to hold workplaces that cannot be successfully mounted otherwise.



Boring Operation



Mohan: I will now explain how to perform a boring operation.

Sailesh: Okay

Mohan: The following steps are to be followed for the boring operation:

1. Clamp the work piece on the chuck.
2. Turn on the main switch to rotate the spindle and check for trueness.
3. Mount the boring tool in the tool post and check for the center height.
4. Select the required spindle speed by changing the belt position in the pulley or by changing the gear shifter lever.
5. Move the slide closer to the starting position of the work piece.
6. Touch the tool on the work piece inside diameter and note the reading on the dial mounted on the hand wheel of cross slide.
7. Pull out the tool, away from the work piece by moving the cross slide.
8. Move the longitudinal slide to the right side for a convenient position for start turning.
9. Adjust the cross slide by adding the needed depth of cut.
10. Switch ON the spindle.
11. Move the longitudinal slide to the left side for boring on the work piece by rotating the hand wheel, up to required length.
12. After reaching the required length pull out the tool, away from the work piece by moving the cross slide by rotating the handwheel. Unlike in OD turning the boring tool has to be pulled out from the contact surface for a smaller distance since the boring tool will collide on the wall of the machined bore.
13. Move the apron to the right side to a convenient position by rotating the handwheel.
14. Stop the spindle and measure the machined bored size using the micrometer or vernier calliper.
15. Compare present machined bore size and the drawing dimension for deciding further steps of cut.
16. Repeat step 11 to 15 till correct dimensions are achieved.

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. What is boring?

2. What are the tool bit settings for boring?

3. Write the steps of the boring operation.

Worksheet

1. Read the following questions and choose the correct answer.

1. A tool, which is used to enlarge a previously drilled hole is known as

   a. Foam tool ☐

   b. Drilling tool ☐

   c. Boring tool ☐

   d. Facing tool ☐

2. The bit is supported in the boring tool bar at a _____ , _____ & _____ angle, depending upon the nature of the work piece being bored.

   a. 90°, 30°, or 45° ☐
   b. 60°, 30°, or 45° ☐
   c. 60°, 30°, or 90° ☐
   d. None of the above ☐

3. The process of reaching through an existing hole and then boring on the "back" side of the work piece is called:

   a. Back boring ☐
   b. Side boring ☐
   c. Drilling ☐
   d. Centre drill ☐

4. The boring tool bar is supported by a _____ which fits into the lathe tool post.

   a. Boring cutter ☐
   b. Cutting holder ☐
   c. Drill ☐
   d. Centre drill ☐

5. _____ is the process of reaching through an existing hole and then boring on the "back" side of the work piece.

   a. Drilling ☐
   b. Boring ☐
   c. Back boring ☐
   d. Cutting ☐

Notes

Answers:

1.

1. c.
2. a.
3. a.
4. b.
5. c.

External Grooving

At the end of this module you will be able to:

- identify the tools with reference to the External grooving operation;
- perform the operation.

Session Plan	
1	External Grooving and its Features
2	Process of External Grooving
3	Key Learning
4	Worksheet

External Grooving and its Features

It's time for Sailesh to learn Grooving. Mohan wants to start external grooving today. He asks Sailesh about the topic to test his knowledge:

Mohan: Today, I am going to tell you about external grooving. Before I start, I want to know if you know what is grooving or what do you understand by grooving?

Sailesh: Grooving according to me is a process to create grooves on a work piece.

Mohan: That's right! Now, can you tell me anything about external grooving?

Sailesh: Hmm, external grooving can be grooving on the outer side of the work piece….

Mohan: It's right to some extent. External grooving is the process of turning in which a grove is created on the outer diameter, the details of the groove needs to be clarified in the component drawing.

Sailesh: Oh! That's means component drawing plays an important role in grooving.

Mohan: Yes, the details of grooving needs to be clarified in the component drawing. The groove thus cut on the OD can have following features:

Width of the Groove.

Depth of the Groove.

Shape of the Groove.

Sailesh: So, how do we select the grooving tool?

Mohan: The tool for turning the groove is selected on the basis of its features that I told you.

Sailesh: Hmm…This is quite easy to remember! I like this topic.

Mohan: That's good, Sailesh! I am very happy to see your interest in grooving.

Process of External Grooving

Mohan: Sailesh now that you know what external grooving is, let me tell you the process of external grooving.

Sailesh: Ok.

Mohan: The steps to be followed for external grooving are:

a. Select the External Groove Cutting tool carefully.

b. Fix the selected External groove tool in the tool holder, ensure minimum overhanging of the tool, and set the cutting tip height of the tool with respect to the turning axis of the machine.

c. Start the machine after selecting the RPM of the machine and ensure correct direction of rotation.

d. Take a skin touch at the point of external groove and measure the placement of the groove position.

e. If the same is OK, lock the carriage for transverse movement and give only the plunge cuts to achieve the inner diameter or the width of the groove as required by the component drawing.

Sailesh: The process is really interesting and easy too!

Mohan: Look at this image for clear understanding. In this image, you will see external grooving cutting tools.

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. What is external grooving and give its features.

2. Write the process of external grooving on Outer Diameter.

Worksheet

1. *Sailesh is asking his co- workers to revise what is being taught in the session. He asks them to give the name of the two types of Grooving?*

2. *Sailesh is asking his co-workers if the written statements are 'True' or 'False':*

   a. External grooving is a process of turning in which the grooving means creating a groove on the Outer Diameter. _____

   b. Component drawing plays an important role in external grooving. _____

   c. Width of groove is not a feature of external grooving. _____

   d. It is important to take a skin touch at the point of external groove and measure the placement of the groove position. _____

Notes

Answers:

1. *Internal Grooving and External Grooving.*
2.
   a. True.
   b. True.
   c. False.
   d. True.

Internal Grooving

At the end of this module you will be able to:

- identify the tools with reference to the Internal grooving operation;
- perform the operation.

Session Plan	
1	Internal Grooving and its Features
2	Process of Internal Grooving
3	Key Learnings
4	Worksheet

Internal Grooving and its Features

It's time for Sailesh to learn Grooving. Mohan wants to start internal grooving today. He asks Sailesh about it t to test his knowledge about the topic:

Sailesh: Mohan, I have learned how to make a groove outside the work piece what if I have to groove inside the piece?

Mohan: Don't worry, Sailesh! I will teach you how to do grooving inside the piece on the inner diameter. Do you know that it is called Internal Grooving?

Sailesh: Yes, I could guess from its operation.

Let me guess the definition of internal grooving... internal grooving means creating a groove inside the piece on the Inner Diameter, the grooves needs to be clarified in the component drawing.

Mohan: You are right! But you didn't mention that it is a process of turning.

Sailesh: Is component drawing important in internal grooving?

Mohan: Yes, component drawings are very important as the grove needs to be clarified in the component drawing. The features of the groove cut on the OD are:

- Width of the Groove
- Depth of the Groove
- Shape of the Groove.

Sailesh: Now, I can tell that the tool of turning has to be selected according to the features of the groove.

Mohan: Correct!

Process of Internal Grooving

Sailesh is teaching his friends the process of internal grooving as it is similar to external grooving that he has learnt previously.

Mohan: Sailesh, The process of internal and external grooving are quite similar. So, can you know describe the process of internal grooving to all present here.

Sailesh: Yes, sure. Mohan: Please start.

To do internal grooving, we have to:

a. Select the Internal Groove Cutting tool carefully.

b. Fix the selected Internal groove tool in the tool holder, ensure minimum overhanging of the tool, and set the cutting tip height of the tool with respect to the turning axis of the machine.

c. Start the machine after selecting the RPM of the machine and ensure correct direction of rotation.

d. Take a skin touch at the point of internal groove and measure the placement of the groove position. It can be controlled by the Drum movement of the cross slide.

e. If the same is OK, lock the carriage for transverse movement and give only the plunge cuts to achieve the inner diameter or the width of the groove as required by the component drawing.

f. **Mohan**: That is brilliant Sailesh! Ok guys now you can see this image which I have got for you. This image shows the Internal Groove cutting tools

Sailesh: Thank you, Mohan.

Internal Groove Cutting Tools

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. What is internal grooving? Write about the features of internal grooving.

2. Write the process of internal grooving on Inner Diameter?

Worksheet

1. *Sailesh wants to test the knowledge of his co-workers. So, he asks them to tell if the component drawing is important in internal grooving?*

2. *Sailesh is asking his co-workers to choose the correct word from the bracket and complete the statement:*

   a. Internal grooving is a process of turning in which the grooving means creating a groove in the _____ (outer/ inner) Diameter.

   b. Tool of turning has to be selected according to the _____ (features/ parts) of the groove.

   c. Start the machine after selecting the _____ (RPM/ setting) of the machine and ensure correct direction of rotation.

   d. You are required to make the selection of _____ (internal groove cutting tool/ external groove cutting tool) carefully before starting the actual process.

Notes

Answers:

1. Yes, component drawing is very important.
2.
   a. Inner.
   b. Features.
   c. RPM.
   d. Internal groove cutting tool.

Fundamentals of Thread Cutting In Lathe

At the end of this module you will be able to:

- state the elements of thread cutting.

Session Plan	
1	What is a Thread
2	Thread Terminology
3	Forms of Thread
4	Key Learnings
5	Worksheet

What is a Thread

As the days passed by Sailesh grew more and more comfortable with the Lathe machine operations. He tries to cut a thread. Needless to say it didn't work out well. By the time he managed to stop the lathe, the workpiece was mangled beyond recognition. Mohan comes to his rescue.

Mohan: Hello Sailesh, I heard about your failed attempt at cutting a thread. Don't worry you'll get a hang of it. But first you need to understand the fundamentals of thread cutting.

A screw thread, often shortened to thread, is a helical ridge formed on inside or outside of cylinder or cone. The threads can be made (or cut) both on external surface or internal surface to obtain respectively the external threads or internal threads.

Did you know Sailesh more screw threads are produced each year than any other machine element?

Sailesh: Really?

Mohan: Yes Sailesh, that's because threads have several applications:

- Fasten devices such as screws, bolts, studs, and nuts
- Provide accurate measurement, as in micrometer
- Transmit motion
- Increase force

Thread Terminology

Mohan: let us go through the various terms associated with threads.

Straight Thread: Thread formed on a cylinder

Taper thread: Thread formed on a cone.

External Thread: An external thread is a thread on the external surface of a cylinder screw or cone.

Internal Thread: An internal thread is a thread on the internal surface of a cylinder screw or cone.

Major Diameter: Commonly known as the outside diameter. On a screw thread, the major diameter is the largest diameter of the thread on the screw or nut. We also use the term full diameter when we talk about the thread of a nut.

Minor Diameter: Commonly also called the root or core diameter. It is the smallest diameter of the thread of the screw or nut.

Pitch Diameter: On a straight thread, the diameter of an imaginary cylinder that passes through thread at a point through the threads at such points where the width of the thread and groove are equal.

Number of Threads: The number of threads per inch.

Pitch: The distance from a point on a screw thread to a corresponding point on the next thread measured parallel to the axis. This distance can be measured in millimeters or by the tpi (threads per inch) which is the reciprocal of the pitch. Simply said:

The pitch in inches is = 1 / the number of threads per inch.

Lead: The distance a screw thread advances in one revolution. On a single threaded screw the lead and pitch are identical; on a double threaded screw the lead is two times the pitch; on a triple headed screw the lead is three times the pitch, etc.

Crest: Top surface joining two sides of thread. (External thread on major diameter and internal thread on minor diameter)

Root: The bottom surface joining sides of two adjacent threads. (External thread on minor diameter and internal thread on major diameter)

Side or Flank: The surface of the thread which connects the crest with the root.

Axis of a Screw: The longitudinal central line through the screw.

Half Angle of Thread: The angle included between a side of the thread and the normal to the axis, measured in an axial plane.

Angle of Thread: The angle included between the sides of the thread measured in an axial plane.

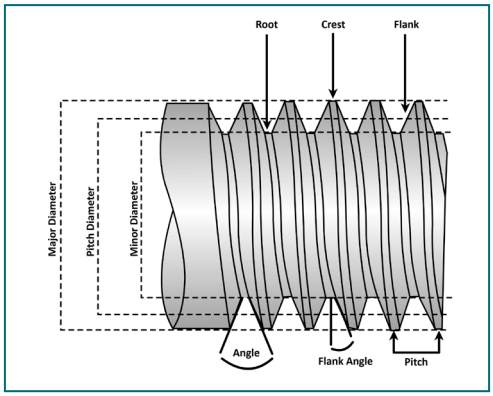
Helix Angle: Angle that the thread makes with a plane perpendicular to thread axis

Base of Thread: The bottom section of the thread; the greatest section between the two adjacent roots.

Depth of Thread: Distance between crest and root measured perpendicular to axis

Right-hand Thread: When a nut is rotated in clockwise direction looking along axis from the nut and the nut advances on the thread, the thread is right hand.

Left-hand Thread: When a nut is rotated in anti-clockwise direction looking along axis from the nut and nut advances on the thread, the thread is left hand.

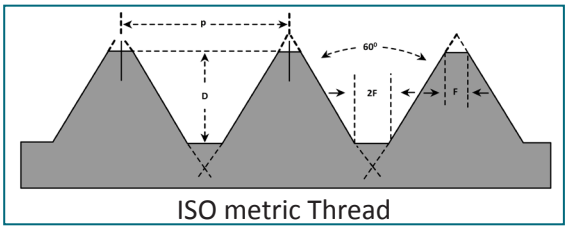


Forms of Thread

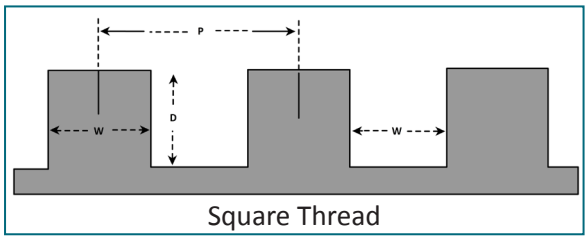
Mohan: Remember Sailesh, Threads are of various types depending on its end use. Some of the commonly used threads are: ISO metric Thread

1. Acme Thread
2. Square Thread

ISO metric Thread: The basic profile of ISO metric screw thread is shown in the diagram given below. BIS (Bureau of Indian Standard) has adopted the same thread form. In this, the thread angle is 60 degrees. As you can see the face of the root will be twice that of the crest. This is to ensure smooth travel of nut over the screw.



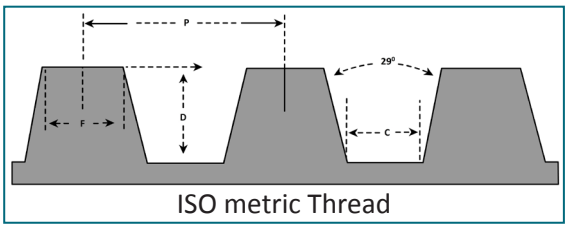
ISO metric Thread



Square Thread

Square Thread: Square thread is an ideal thread form for power transmission. In this, the thread flank is at right angle to the axis. This enables the nut to transmit very high pressures, as in the case of a screw jack and other similar applications.

Acme Thread: It is a modified form of square thread. It is much stronger than square thread because of the wider base and it is easy to cut. The thread angle is 29 degrees. The inclined sides of the thread facilitate quick and easy engagement and disengagement as for example, the split nut with the lead screw of a lathe.



ISO metric Thread

The diameter-pitch combination depends on the shape of the thread profile.

An isometric thread is the most common thread to be cut, and once you can cut it, no other thread (internal, external, acme, square, etc.) is beyond reach.

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

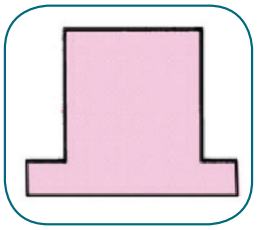
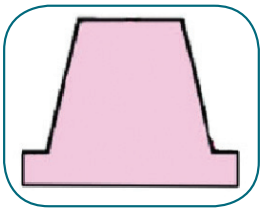
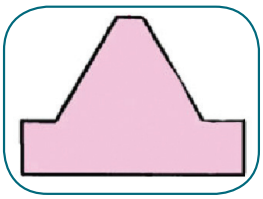
1. Write a short note on the following terms:
   a. Angle of Thread
   b. Pitch
   c. Crest
   d. Minor Diameter
   e. Depth of Thread

2. Write a detailed note on the thread forms discussed in this session.

3. Hold a discussion on some other forms of thread that Sailesh and Mohan have not discussed in this session. Write down their characteristics and applications.

Worksheet

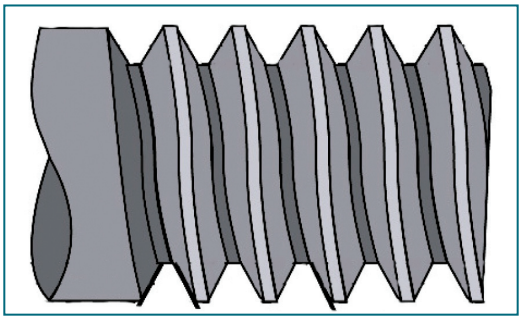
1. *Help Sailesh identify the form of thread shown below:*

1		
2		
3		

2. *Sailesh is confused between the various parts of a thread. Help him by pointing out the terms listed below:*

a. Root b. Crest c. Flanks

d. Angle of Thread e. Major Diameter



3. *Fill in the blanks with the correct answer.*

a. In the _____ thread, the thread flank is at right angle to the axis.

b. A _____ nut is rotated in anti-clockwise direction looking along axis from the nut.

c. In a common ISO metric thread, the thread angle is at _____ degrees.

d. The thread cut on an external surface is called _____threading.

e. _____ is the distance a screw thread advances in one revolution.

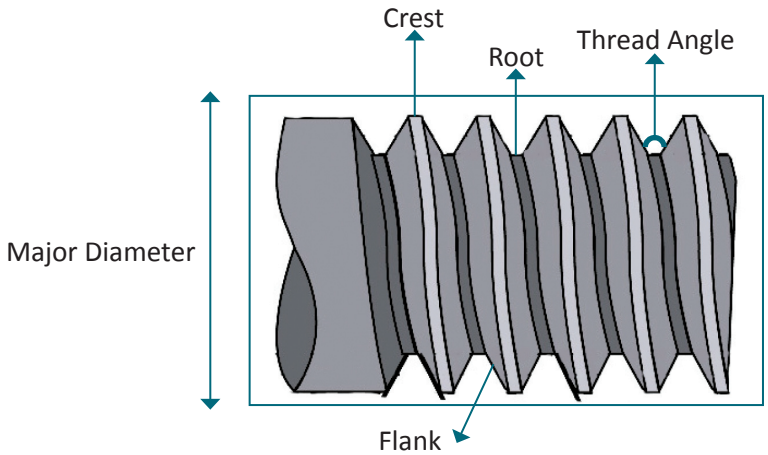
Notes

Answers:

1.

a. Square Thread b. Acme Thread c. Isometric Thread

2.



3.

a. Square Thread b. Left handed c. 60
d. External e. Lead

External Threading

At the end of this module you will be able to:

- identify the tools with reference to the External Threading operation;
- perform the External Threading operation.

Session Plan	
1	External Threading
2	Steps for External Threading
3	Preparation of Thread Cutting
4	Key Learnings
5	Worksheet

External Threading

Sailesh is looking at a fine cut screw that has external thread cut on it. While looking at the screw he is thinking about the technique of threading. He asks Mohan about it:

Sailesh: I want to know about this cut like thing on a screw.

Mohan: This is not cut like thing it is a thread or external thread.

Sailesh: what is thread or threading?

Mohan: A thread is a uniform helical groove cut inside of a cylindrical work piece, or on the outside of a tube or shaft. In case of the screw that you are holding in your hand, it has external thread.

Sailesh: What is external threading?

Mohan: External threading is the process of threading done externally on a work piece like screw etc.

Steps for External Threading

Sailesh: Mohan, I am keen to know about the steps of external threading.

Mohan: Sure, let me tell you the steps of external threading.

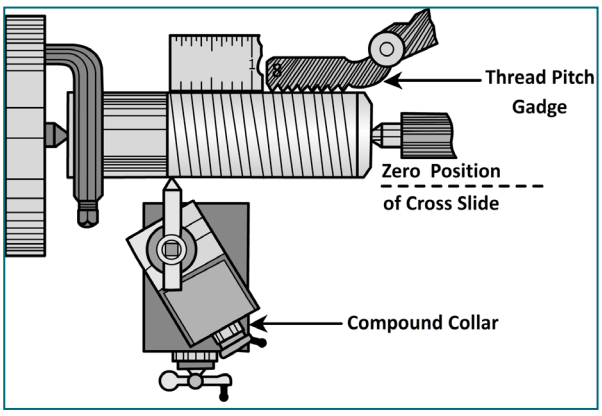
Steps to cut external right-hand thread on lathe:-

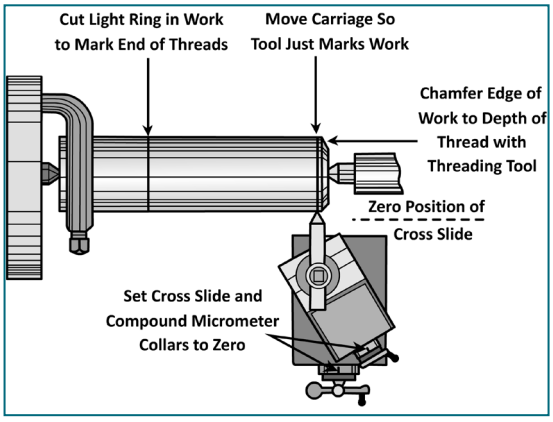
1. Set the spindle speed at one-quarter the normal turning speed.
2. Set the gearbox for the desired transverse feed or pitch.
3. Set the lead screw to turn in the forward direction. When engaged, the carriage will move from right to left.
4. Set the tool tip height to center of the lathe axis and the compound slide angle to 30º. (Because we are going to cut a 60º angle metric thread).

5. Mount the work (between centers, in a chuck or in a chuck plus a center, in a collet or in a collet plus a center) and ensure it is secure and there is no end-to-end play or job is running concentric.
6. Set the tool at right angles to the work using the thread center gauge.
7. Measure and control the diameter of the work. Good practice is to have the a. work diameter 0.002 inch (0.05 mm) under the thread major diameter.
8. Start the lathe and chamfer the right end of the work to just under the thread final minor diameter using the left edge of the cutting tool.
9. Measure and mark the work where the threads will end by cutting a light ring with the threading tool bit. Crank the tool bit away from the work so the carriage can be moved. The spindle will still be turning.
10. Manually position the carriage toward the right end of the work piece and, using the compound rest, move the tool bit towards the work so it just lightly marks it. Set both the cross slide and compound micrometer collars to zero.
11. Back the tool bit away from the work using the cross slide and position the carriage so the tool bit tip is ⅛ inch / 15 mm or more to the right end of the work, the starting point.
12. Using the compound slide, move the tool bit into the work between 0.001 and 0.003 inches (0.05 mm approximately) with your left hand on the compound crank. Engage the half-nut lever with your right hand when the thread dial reaches the right graduation. This will make the first, or scratch/ skin pass.
13. When the tool reaches the left end of the cut, quickly crank the tool away from the work piece using the cross slide crank, and disengage the half- nuts.
14. Move the carriage manually back to the starting point and set the cross slide back to the zero position.
15. Stop the spindle, and using either a thread pitch gage or a ruler, check that the cut is the desired pitch. Correct the gearbox setting and try again if the thread count is wrong.
16. Restart the spindle.

Sailesh: Although the steps are lengthy, the process seems easy.

Mohan: No Sailesh, this is not the complete procedure. Now, I must tell you how to prepare for threading. Look I have made a diagram to help you.





The above image will help you to understand how to Check thread pitch after initial scratch / skin touch.

Now, let me tell you the next steps in external threading,

17. Manually move the carriage back .Then, using the collar calibrations, move the cross slide to the zero position, with the compound slide, advance the tool bit in preparation for the next thread cut, and repeat the cycle by increasing the depth of cut with each pass. The idea is to use cross slide motion to pull the tool away from the work piece at the end of the thread and move the tool back to the zero position for the next pass. Once the compound collar has been set to zero, the compound is incremented for each pass and is never used to pull the tool away from the work.

18. Engage the half-nut lever with your right hand when the threading dial reaches the correct graduation and the tool will move from Position 3, through Position 4, and to the end of the threads at Position 5. When the tool reaches Position 5, use the cross slide to quickly withdraw the tool to Position 6. Then manually move the carriage back to Position 1. You are now ready to begin the next cutting pass.

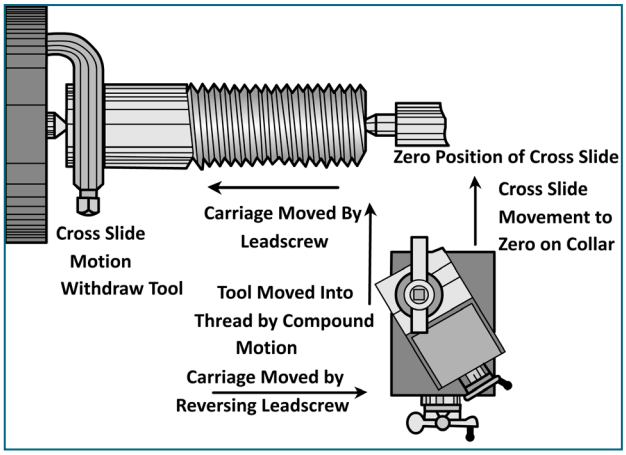
Sailesh: Is the threading job complete now?

Mohan: No, Sailesh. We are still left with the process of completing external threading job.

Sailesh: There is a separate process of completing the threading!

Mohan: Let me tell you, the completion process of threading. Again I have made a diagram for you to understand better.

Completing the threading process

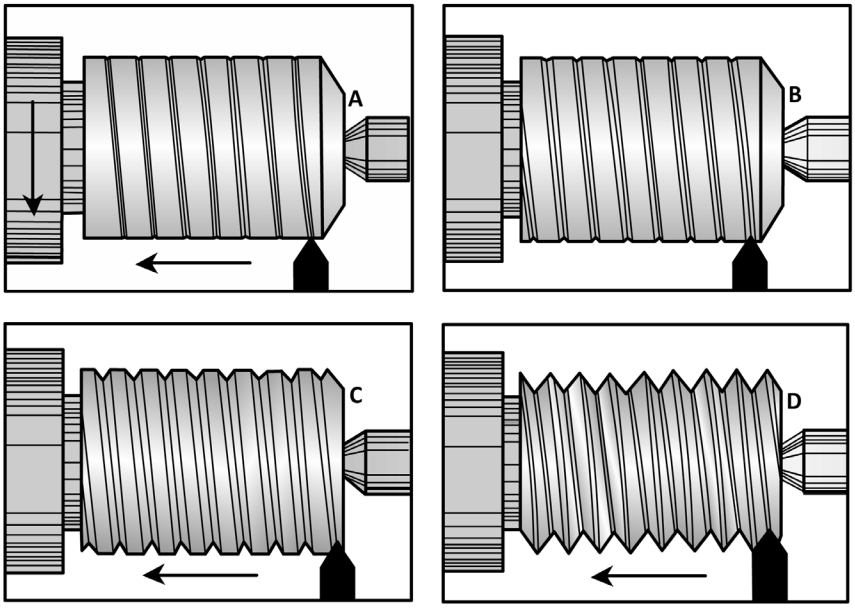


19. Tables in Machinery's Handbook show the final thread depth for all common thread forms. This depth can be read off the compound collar and can be checked using a threading micrometer. Also, thread center gauges have the thread depth for various threads engraved on them.

20. Not having this information at hand, the operator can watch the thread develop until the width at the thread bottom (root) just about equals the width of the thread top (crest). Then take additional small cuts until the mating part or master nut fits properly.

21. Use a diamond file to remove burrs on the top thread edge.

Cutting 60º external threads on a lathe equipped with a thread-chasing dial is the most common and least complicated threading operation.



Mohan: Now, the external threading process is complete.

Sailesh: It is really not an easy task!

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. What is external threading?

2. What are the steps required to complete the external threading?

Worksheet

1. *Sailesh is testing the knowledge of his co-workers. He has given some statements, complete the statements by filling up the spaces below:*

   a. External threading is done externally on a work piece like _____.

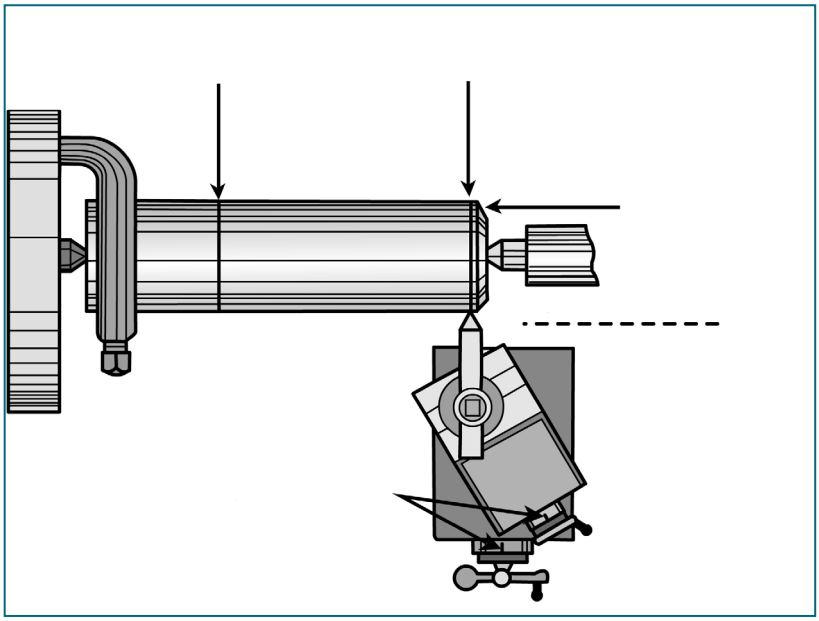
   b. A thread is a uniform _____ cut outside of tube or shaft.

   c. While doing the external threading you need to set the _____ speed at _____the normal turning speed.

   d. You need to _____ and control the_____ of the work.

   e. When the tool reaches the left end of the cut, quickly _____ the tool away from the work piece using the _____ crank, and disengage the half-nuts.

2. *Sailesh has made the following image for preparation of thread and label various parts marked:*



Notes

Answers:

1.

a. Screw.

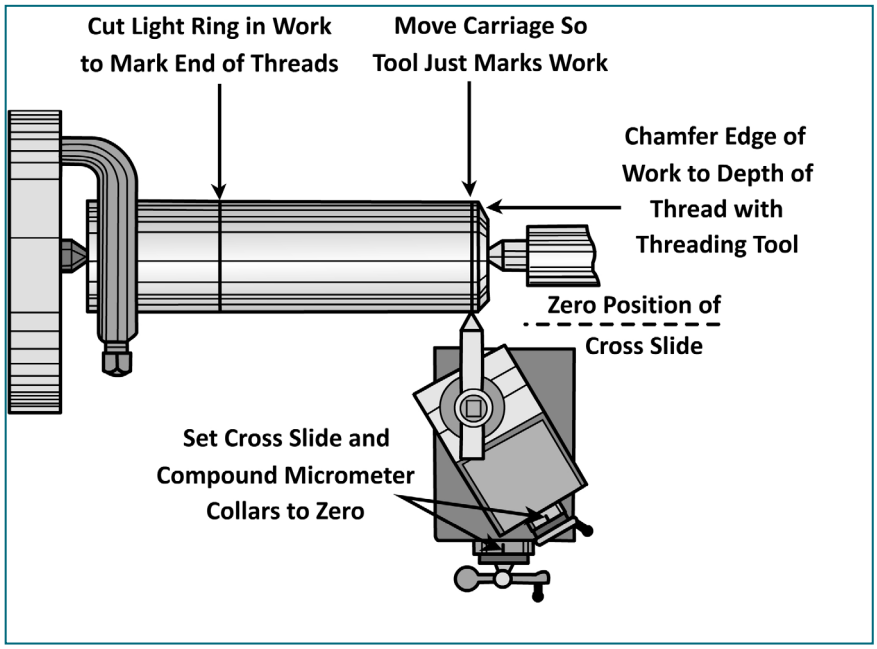
b. Helical Groove.

c. Spindle, One- quarter.

d. Measure, Diameter.

e. Crank , Cross-slide.

2.



Internal Thread Cutting

At the end of this module you will be able to:

- identify the tools with reference to the internal threading operation;
- perform the internal threading operation.

Session Plan	
1	Tools to Cut Internal Threads
2	Ways to Check Alignment of a Tool Bit
3	Setup for Cutting Internal Threads Starting from Outside the Work
4	Setup for Cutting Internal Threads Starting from Inside the Work
5	Key Learnings
6	Worksheet

Tools to Cut Internal Threads

Today, Sailesh is going to learn about the tools to cut internal threads. He asks Mohan:

Sailesh: What are the tools used to cut internal threads?

Mohan: There are mainly two types of tools to cut internal threads.

The most commonly used tools are commercial boring bars with HSS, alloy steel or tungsten carbide tool bits.

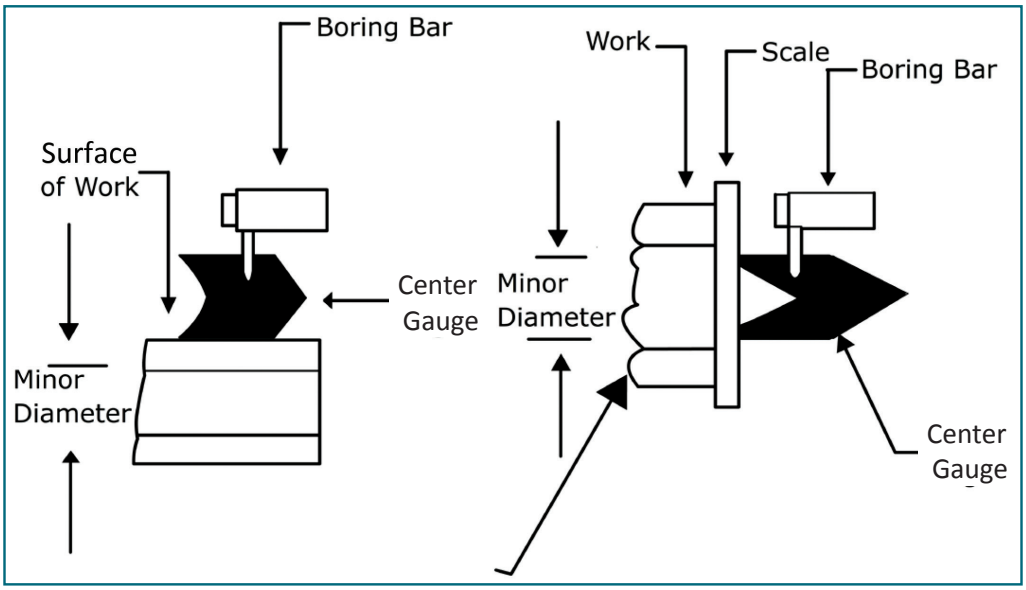
For holes that are too small for a boring bar, an HSS tool bit blank may be ground into a small, single-piece threading tool. These are also commercially available.

Ways to Check Alignment of a Tool Bit

Mohan: I hope you are clear about the tools to cut internal threads, now let me tell you about the ways to check alignment of a tool bit.

Cut Internal Threads

To start with it, we first need to set the tool bit axis vertical to the work for internal threads. The two ways to check alignment of a tool bit for internal threads are:

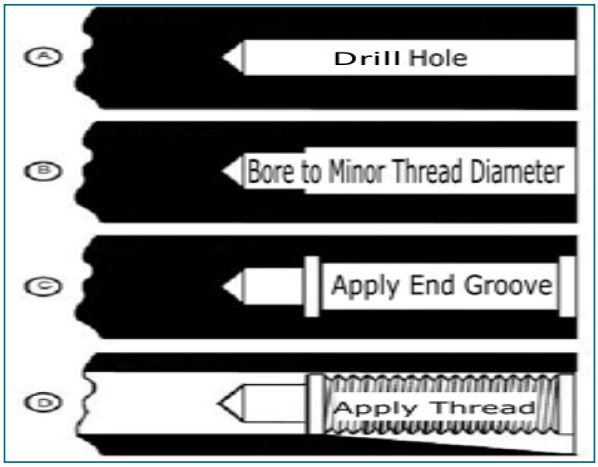


Two Ways to Check Alignment of a Tool Bit For Internal Threads.

Sailesh: Okay.

Mohan: The steps to cut internal right-hand threads are: -

1. Place the work in a chuck, collect or face plate and drill a starting hole for the boring tool.



2. Now start drilling or boring a hole of the threads in the work piece. Remember you can even take this diameter from Machinery's Handbook.
3. If you find that the thread is not running completely through the work then you need to use a square- nose tool inside the work piece to cut a groove where the threads end.

Sailesh: Why is a square- nose tool used?

Mohan: It is a tool that makes it easier to start or stop the threading process as there is no abrupt end of the thread.

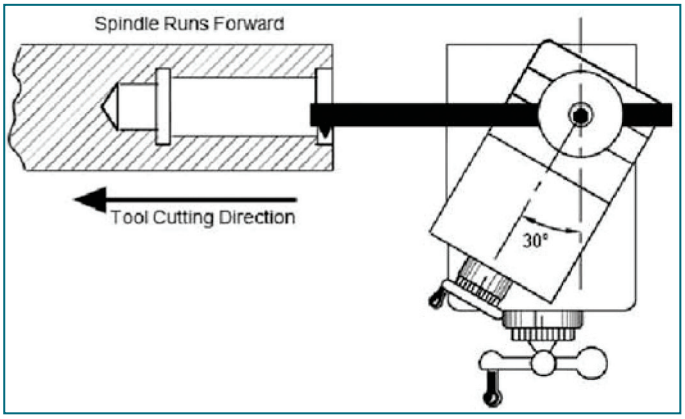
Let me tell you how to proceed:

4. It is important to cut a recess in the open end of the work piece which is equal to the thread's major diameter. This will help you to observe the depth of thread during threading process.

5. You may start threading from either sides that is from outside or from inside the work. If you start threading from inside then you can do away with the problem of not being able to see the internal end of the threads so you know when to open the half-nuts. This method is only suitable on lathes with a long-taper or cam-lock spindle nose which will not unscrew from the spindle when run in reverse under load. You will need to choose one of these methods before installing the boring bar (or single- piece tool bit) with the properly shaped cutting bit in the tool holder, and set the compound at an angle of 30º (In case of a 60º metric thread) to the work axis.

Setup for Cutting Internal Threads Starting from Outside the Work

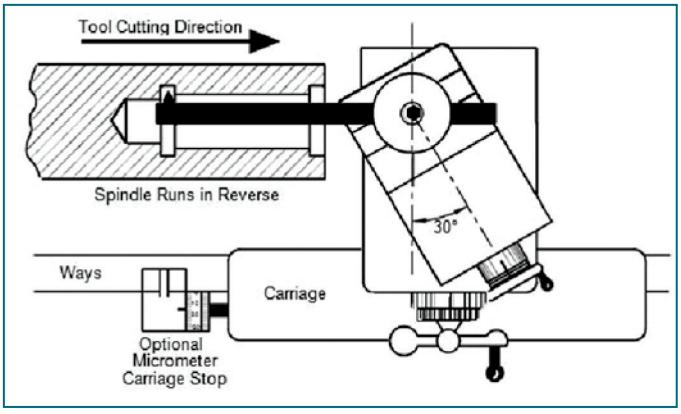
Mohan: These images will help you to understand the set up for cutting internal threads starting from outside the work



Setup for Cutting Internal Threads Starting From the Outside The Work

Setup for Cutting Internal Threads Starting from Inside the Work

Mohan: I hope you are now clear about the setup for cutting internal threads starting from outside the work and now let me tell you about the setup for cutting internal threads starting from inside the work. startingfrom inside the work.



The steps are:

1. Use the tailstock center as a reference to adjust cutter bit height exactly on center. Be sure, that there is sufficient end clearance on the lower face of the cutter bit and there is enough room behind the boring bar and cutter bit. This will assure you that they will not bump into the interior of the work piece when the tool bit is withdrawn from the threads.

2. Fix the spindle and the lead screw to turn forward, that is, to move from right to left. This will enable the tool run into the work.
3. Make the threads like you do in the external threading operations. The only change will be that the cross slide is moved forward to clear the tool, and return the carriage to the starting point. It is helpful to put a mark or piece of tape on the boring tool bit arm or on the ways to indicate the stopping point at the end of the internal threads.
4. Observe the depth of the threads in the lower-level starting area to know when the threads are complete, check with a thread plug gauge or the mating part.

Minimize the length of the unsupported boring bar because it will deflect and restrict you from taking smaller cuts. Plan to make the final pass more than 0.001 inch/ 0.02 mm depth as the spring in the bar will make such a small cut impossible.

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. Explain in detail about the tools to cut internal threads.

2. Describe the ways to check alignment of a tool bit.

3. Write about the setup for cutting internal threads starting from inside the work.

Worksheet

1. Help Sailesh in completing his lines by filling the blank spaces with correct answers:

1. Commercial boring bars with HSS, alloy steel or _____ tool bits are the most common.

   a. Tungsten carbide ☐
   b. Aluminum ☐
   c. HSS tool ☐
   d. Threading tool ☐

2. It is helpful to place a mark or piece of tape on the boring tool bit arm or on the ways to indicate the _____ at the end of the internal threads.

   a. Running point ☐
   b. Stopping point ☐
   c. Bore ☐
   d. Hole ☐

3. _____ makes it easier to start or stop the threading process.

   a. Drill ☐
   b. Square noise tool ☐
   c. Commercial boring bars ☐
   d. HSS tool ☐

Notes

Answers:

1.

1. a.
2. b.
3. b.

Parting

At the end of this module you will be able to:

- identify the tools with reference to the Parting operation;
- perform the Parting operation.

Session Plan	
1	Parting
2	Process of Parting
3	Key Learnings
4	Worksheet

Parting

Sailesh will be learning about parting today. He is excited to learn a new operation of lathe machine.

Sailesh: I have heard about parting what is it?

Mohan: Parting as the name suggests is an operation of cutting off a piece held in the chuck of a lathe.

Sailesh: Oh! It seems to be an interesting operation. I want to know more about it…

Mohan: Sure, Why not! I will tell you about it in detail.

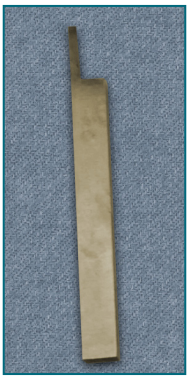
Sailesh: Thank you, Mohan.

Mohan: My pleasure. Parting is the process of parting off some part of the turned part by the way of plunge cut or cutting a deeper external groove.

Sailesh: Is there a special tool used for Parting?

Mohan: Yes, Parting is done with the help of Parting tool. It is a deeper and narrower tool than a turning tool. It is used to make narrow grooves and for cutting off parts.

Look at this image that I have got for you. It is an image of Parting tool but remember you have to select the tool according to the purpose of parting.



Parting Tool

Process of Parting

Sailesh: Mohan I was reading about the precautions for using the parting tool and got confused. Is there any difference between parting and precautions of using parting tool?

Mohan: Yes, the parting and precautions are not at all same. At first, I told you about precautions and now I will tell you about the parting process. After knowing the process your doubts will be cleared.

Do you know that the process of parting is very much alike the process of external grooving?

Sailesh: Oh! Is that so? This means I will be able to understand it easily.

Mohan: Yes I think so too…. Well the process of parting is:

Select the Parting tool carefully.

Fix the selected tool in the tool holder, ensure minimum overhanging of the tool, and set the cutting tip height of the tool with respect to the turning axis of the machine.

Start the machine after selecting the RPM of the machine and ensure correct direction of rotation.

Take a skin touch at the point of parting and measure the placement of the groove position. It can be controlled by the drum movement of the transverse slide.

If the skin touch is ok, lock the carriage for transverse movement and give only the plunge cuts to part away the piece as required.

Sailesh: You are right; the process is similar to the process of external grooving that I have already learnt in previous sessions. It makes it all the easy to remember it.

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. What is parting? What are the precautions that an operator should take care of while parting?

2. Write the process of Parting?

Worksheet

1. *Mohan is testing the knowledge of Sailesh and his friends. Help Sailesh by completing each statement*

   a. Parting is an operation of _____ a piece from apart held in the _____ of a lathe.

   b. The tool that is used for parting operation is called a _____

   c. It is important to ensure that the parting tool is _____ to the _____ of rotation and the tip is the same height as the center of the part.

   d. The best way to hold the tool against the tip is the _____ as the center of the part.

2. *Mohan has asked Sailesh that in the process of parting what is the next step after the step no 3 or 'Start the machine after selecting the RPM of the machine and ensure correct direction of rotation'. Help him by giving suitable answer to the asked question.*

Notes

Answers:

1.

a. cutting off and Chuck.

b. Parting tool.

c. Perpendicular.

d. Same height.

e. External Grooving.

2. *Step no 4 i.e 'Take a skin touch at the point of parting and measure the placement of the groove position. It can be controlled by the Drum movement of the transverse slide.*

Introduction to CNC

At the end of this module you will be able to:

- state the importance of CNC.

Session Plan	
1	Basic Concept of NC
2	Conventional Numerical Control
3	Computer Numerical Control (CNC)
4	Direct Numerical Control (DNC)
5	Applications of NC System
6	Advantages of NC System
7	Limitations of Numerical Control
8	Key Learnings
9	Worksheet

Basic Concept of NC

It's the first time that Sailesh is going to learn about the Numerical control and he is really excited about it. He asks Mohan:

Sailesh: What is NC?

Mohan: NC stands for Numerical Control and it is a method of automatically operating a machine tool by means of numbers, letters and symbols that controls movements through some form of an input medium.

This medium can be a punched tape, a direct wire from computer magnetic tapes and Diskettes.

It is further divided into three parts.

Sailesh: What are they?

Mohan: The common types of NC systems used in machine tools are:

- Conventional NC Computer numerical Control (CNC).
- Direct numerical Control (DNC).

Conventional Numerical Control

Mohan: Sailesh, let me first tell you about the Conventional NC

Sailesh: Okay!

Mohan: It is a hard wired based system, employing integrated circuits which are permanently wired and arranged on printed circuit board. Also this NC system is purely a hard wire based system. It is not easy to change the main features of its machine control unit.

Sailesh: Does that mean that there are no provisions for changing feeds and speed during machining?

Mohan: Yes, that's right!

Computer Numerical Control (CNC)

Mohan: Let's begin with CNC that is Computer Numerical Control.

Sailesh: What is it?

Mohan: It is a soft ware based system, in which the computer replaces the control unit of the conventional NC. It does not carry the hard wired logic systems, and all their functions for controlling the machine tool are performed by the software programme of the Computer.

Sailesh: Is there any special type of computer used for it?

Mohan: The Computer used in it is called mini Computer and its main objective of using this system is to simplify the hard wire of conventional NC and replace it with software. The programme is entered into the computer through tape, or, key board and stored in memory, which can be called whenever required. Programme editing and modification is easy here in CNC systems.

Direct Numerical Control (DNC)

Mohan: I hope you have understood about the CNC and now I will tell you about DNC.

Sailesh: Okay!

Mohan: It is also a computerized Numerical control system, but it differs from a CNC system in that it employs a separately located Central Computer and directly controls several machines simultaneously.

Applications of NC System

Mohan: Now, I will tell you about the applications of the NC and why they have come in practice.

With the increasing competition in industries, there is a continuous endeavor to reduce overall production cost without compromising the quality of the product.

This is possible only by:

a. Increasing production rate

b. Optimizing machining conditions

c. Improving dimensional control

d. Minimizing human involvement in actual processing

e. Reducing non-machining times and similar other measures.

These above requirements have led to wide application of numerical control machines.

Some important NC machine tools in metal cutting are turning center and machining center.

Sailesh: Are there any conditions under which the numerical control system is used?

Mohan: It would be most appropriate to use numerical control under the following conditions:

1. Machining of such components which require 100% inspection.
2. Repetitive production of Precision parts in small and medium lot sizes.
3. When several operations are to be performed in machining of a part.
4. When complex machining operations are involved.
5. When the amount of metal to be remove is high.
6. When economic considerations demand a substantial reduction in "lead time" for the manufacture.
7. When Jig and Fixture involvement is high.
8. When dimensional accuracy of the part is very high.

However, it is not necessary that all the above situations should be present to justify the application of NC system but it is also true that higher the number of such situations present simultaneously the better is the justification for application of NC.

Advantages of NC System

Mohan: Let's discuss the advantages of the NC system.

Sailesh: Okay!

Mohan: The advantages of using a NC system are:

- **Increased productivity**: Due to reduced set up and lead times.
- **Less rejection**: Due to elimination of human errors and high accuracy of NC systems.
- **Increased rate of production**: Due to heavy reduction in non-machining times and optimization of machining conditions.
- **Reduced non-productive time**: Due to use of lesser set ups, lesser setting times, lesser work handling times, facilitation of automatic tool change in some of machines.
- **Reduced inspection requirement**: Due to uniform quality of production parts.
- **Higher accuracy**: Repeatability error is very less in case of NC system.
- **Better quality control**: Due to higher accuracy of NC system, the quality of product is effectively controlled.

- **Ease of complex machining**: NC facilitates performing of complex machining easily, accurately and at much faster rate.
- **Greater utilization of manpower**: At the time of continuous running of NC machines, the operator can be utilized for some other operations such as program check, design study, process planning etc.
- **Reduction of Human errors**: As the machine runs with help of programming, the human error is eliminated.
- **Safety of Operator**: As the control of NC system is located at a distance from the machining area, the operator gets no chance to be exposed to working zone.
- **Safety of machine tool**: There is practically no chance of any damage to the machine tool due to human error since there is almost no requirement of operator involvement during running of machine.
- **Accurate Estimation**: The product cost and rate of production can be accurately predicted and estimated because of programmed machining, which does not allow any deviation of the actual production time from the programmed one.
- **No need of skilled Operator**: Since all the manual skills required in conventional machining are transferred to the machine in NC, the essential requirement of a skilled operator can be easily done away with.

Limitations of Numerical Control

Mohan: Sailesh, now let me tell you the limitations of the NC system.

Sailesh: Okay.

Mohan: The main disadvantages of N/C system are:

1. A relatively high price for the N/C machine.
2. More complicated maintenance, a special maintenance crew is desirable.
3. A highly skilled and properly trained programmer is needed.
4. Redundancy in labor may be there.

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. Classify NC system.

2. What are the advantages of NC systems over conventional system?

3. Write the limitations of NC and CNC system?

Worksheet

1. Read the following questions and choose the correct answer.

1. _____ is the method of automatically operating a machine tool by means of numbers, letters and symbols.

   a. Conventional NC

   b. Computer Numerical Control (CNC).

   c. Direct Numerical Control (DNC).

   d. Numerical control (NC)

2. The Computer used in NC is called _____.
   a. Mini Computer
   b. Electronic system
   c. NC System
   d. N/C machine

3. NC system is purely a _____ system
   a. Hard wire based system
   b. Computer based system
   c. Electronic system
   d. Wire based system

Notes

Answers:

1. d.
2. a.
3. a.

Coordinate Systems Cartesian

At the end of this module you will be able to:

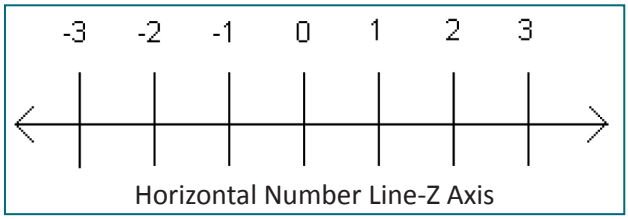
- apply the knowledge of cartesian system to hold the tool in the correct position.

Session Plan	
1	Coordinate Systems- Cartesian
2	3D Coordinate Systems Cartesian
3	Understanding Cartesian Coordinate System With the Help of Figures
4	Identifying the Location of a Point / Address
5	Key Learnings
6	Worksheet

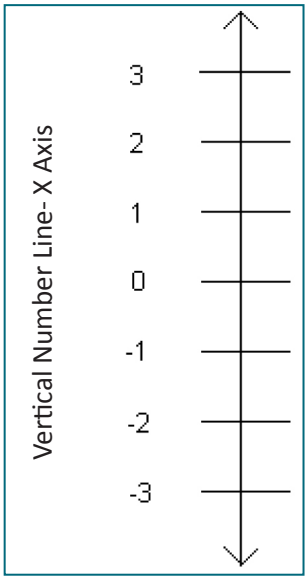
Coordinate Systems- Cartesian

Sailesh: Hi Mohan, I am reading about the Coordinate Systems Cartesian. It seems quite complicated I was wondering if you could help me with the concept.

Mohan: Don't worry Sailesh! It's really quite simple.



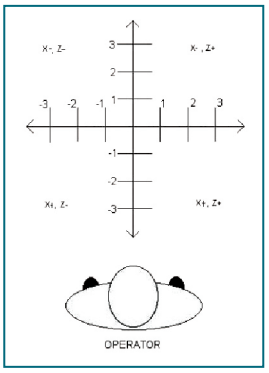
Horizontal Number Line-Z Axis



Imagine, you are sitting in a room and when you look around you'll see at least 6 different planes on our sides, up and down. Above you is the roof and below you is the floor; If you see one of the corners there are 3 lines emerging from a common point. Each of this line denotes an axis/ direction and if you name it you will say X, Y and Z axis.

Let us understand the zero point or a reference point for each axis and the plus and minus side of the same.

Mohan: You are right, Sailesh! See the figure of operators working grid given above. Now, imagine that an operator is standing in front of this figure. He will have X and Y axis in front of him. Now, he can see four quadrants that are formed namely (X Positive Y positive), (X negative and Y positive), (X negative and Y negative) and (X positive and Y negative).



Sailesh: I am getting a little confused. Does it mean that the figure having four quadrants refers to 2D Cartesian coordinate system?

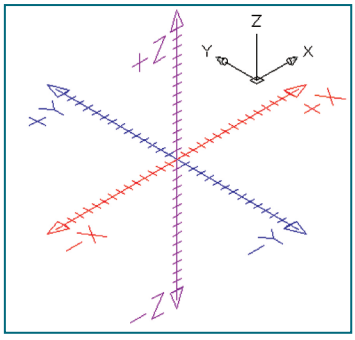
Mohan: Not exactly. When we talk about two axis and four quadrants then surely we are referring to 2D Cartesian coordinate systems.

3D Coordinate Systems- Cartesian

Sailesh: I have learned about 2D Cartesian Co-ordinate systems but I have heard that it is possible to have coordinate Cartesian systems in 3D too? Will you tell me about it?

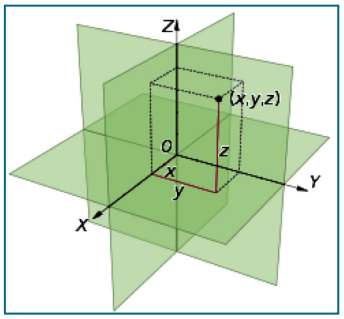
Mohan: Sure, Sailesh! It's quite simple to understand. When we add one more axis called Z axis to the present two axes X and Y, it becomes 3D.

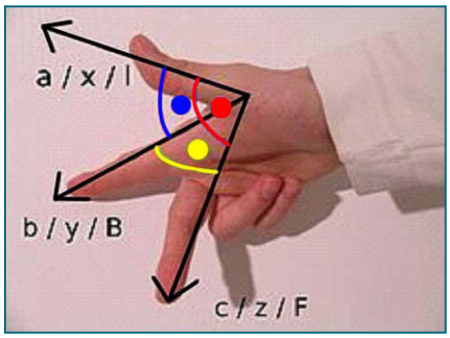
Sailesh: Why do we choose a Cartesian coordinate system?



Mohan: Choosing a Cartesian coordinate system for a three-dimensional space means choosing an ordered triplet of lines (axes), any two of them being perpendicular; a single unit of length for all three axes; and an orientation for each axis. In two-dimensional case, each axis becomes a number line. The coordinates of a point p are obtained by drawing a line through p perpendicular to each coordinate axis, and reading the points where these lines meet the axes as three numbers of these number lines.

Alternatively, the coordinates of a point p can also be taken as the (signed) distances from p to the three planes defined by the three axes. If the axes are named x, y, and z, then the x coordinate is the distance from the plane defined by the y and z axes. The distance is to be taken with the + or − sign, depending on which of the two half-spaces separated by that plane contains p. The y and z coordinates can be obtained in the same way from the (x, z) and (x, y) planes, respectively.

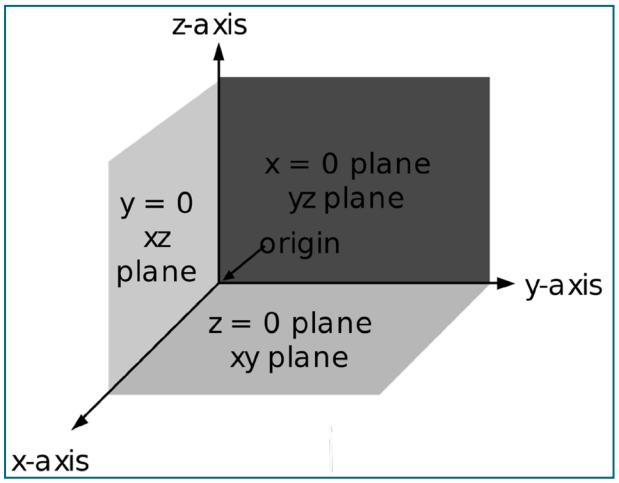




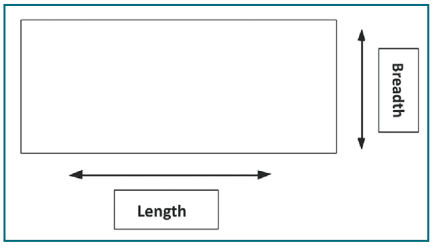
Simple Understanding of 3D Coordinate System

Sailesh: Is it all about 3D Cartesian coordinate systems?

Mohan: Let me show you an image that will help you in getting a clear understanding about 3D Cartesian coordinate systems. The image given below will help you understand the 3D Cartesian coordinate systems.



Understanding Cartesian Coordinate Systems With the Help of Figures

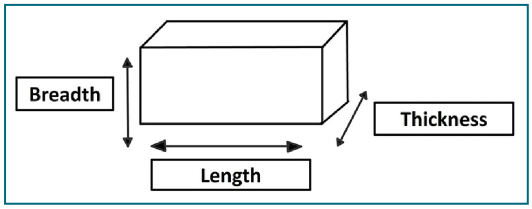


Sailesh: I think that Cartesian Coordinate Systems can be very easily understood with the help of figures, is it true?

Mohan: Yes Sailesh, you are right the shapes can help you in understanding Cartesian coordinate systems in a better way. Let me explain:

The shapes we know which are in two dimensions can be made on a planer surface and have only two dimensions without any thickness/depth.

For Example: A rectangle is a 2 dimensional (2D) shape. A rectangle has two dimensions length and breadth.



Sailesh: It's all about 2D shapes that can be related to 2D Cartesian coordinate systems, what about 3D shapes or 3D Cartesian coordinate system.

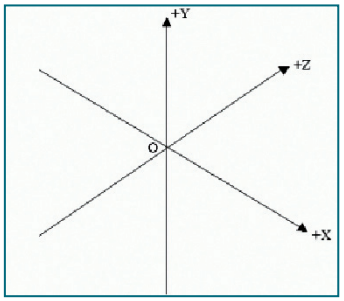
Mohan: If you add a thickness/depth, then it becomes a three dimensional object. A 3D shape as the name suggests is a shape with three dimensions. A 3 dimensional object like this would have 12 sides, (4 x3) whereas a 2D figure would have only four sides (4 x 1).

Identifying the Location of a Point/ Address

Mohan: After knowing about the Cartesian coordinate systems, it is important for you to know about identifying the location of a point or address.

Sailesh: What is that? It sounds interesting.

Mohan: You are aware that a 2D shape is made on a planer surface. Hence, it can be plotted easily on the X and Y axis. In case of 2D shapes, X and Y axis are two perpendicular axes which can be marked as per required dimensions on them. The point where X and Y meet is called origin and is generally marked as O.



Sailesh: Does this mean that the point O will always come in the center?

Mohan: Absolutely right. Each axis has a marking on it used to find the distance between two points. Both the axis run infinitely and includes both positive and negative values.

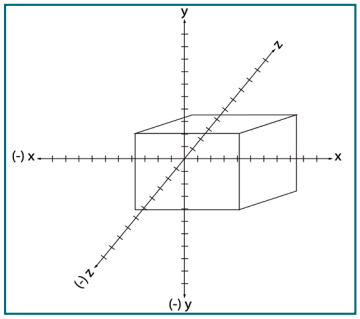
Sailesh: How do we measure in case of 3D shapes?

Mohan: When we want to plot a 3D shape on the axis we need a third axis called the Z axis to accommodate the third dimension for a 3D shape.

Sailesh: I am sorry, I am unable to find the difference I am getting so confused.

Mohan: Relax Sailesh! Don't get confused. A 2D figure can only be drawn on paper and can never be created physically, whereas a 3D figure can be actually a physical object apart from being a 2D object.

We have used a box having 6 faces to find the address of the corners with reference to all the 3 planes X, Y and Z. The address of each corner is seen in the figure below.



Sailesh: I am sorry, I am unable to find the difference I am getting so confused.

Mohan: Relax Sailesh! Don't get confused. A 2D figure can only be drawn on paper and can never be created physically, whereas a 3D figure can be actually a physical object apart from being a 2D object.

We have used a box having 6 faces to find the address of the corners with reference to all the 3 planes X, Y and Z. The address of each corner is seen in the figure below.

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

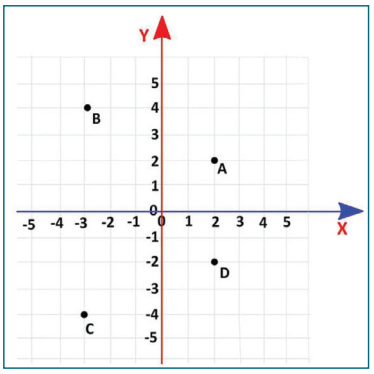
1. Explain the Coordinate systems- Cartesian 2D?

2. Explain the Coordinate systems- Cartesian 3D?

3. How do you identify the location of a point or address?

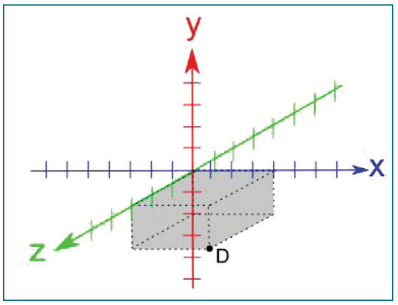
Worksheet

1. *In the figure given below, help Sailesh write down the Cartesian coordinates of the point A,B,C, D.*

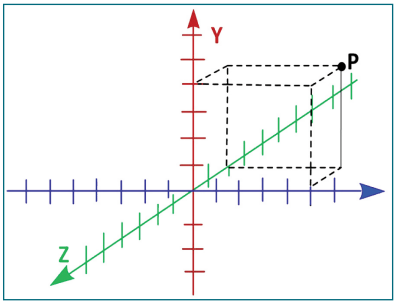


2. *In which quadrant do you find the point (3, -2)?*

3. *If D is a point in 3 dimensions. What are the Cartesian coordinates of the point E?*



4. *What are the three dimensional coordinates of the point F?*



Notes

Answers:

1.

a. (2,2)

b. (-3,4)

c. (-3, -4)

d. (-2,2)

2. 4th quadrant

3. (4,-2,3)

4. (5,4,-2)

Control Systems

At the end of this module you will be able to:

- identify the machine keys;
- start and reference the machine.

Session Plan	
1	Control Panel
2	Reference Mode (Starting and Referencing a Machine)
3	JOG Mode (Manual) Operations
4	Edit Mode
5	Auto Mode
6	Auto SBLK (Single Block) Mode
7	MDI (Manual Data Input) Mode
8	TAPE Mode (DNC – Direct Numeric Control)
9	Key Learnings
10	Worksheet

Control Panel

Today, Sailesh is going to learn about the Fanuc control system. He asks Mohan:

Sailesh: What is a Fanuc control system?

Mohan: Let me explain it to you in a simple. Let's start with a control panel. Do You know what is a control panel?

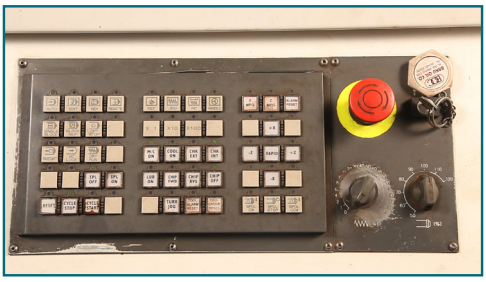
Sailesh: No

Mohan: The control panel consists of keys to communicate with CNC controller and keys/ buttons/ switches for controlling machine movements and functions.

Every machine will have a typical operator panel for CNC controller supplied by Fanuc. Program keying in and certain functional setup are performed using this operator panel. Machine movements, mode selection, turret position selection in JOG (manual) mode, speed and feed override switches along with machine ON/OFF switches are placed in an adjoining panel to the CNC controller operator panel.

Given below is the image of the machine control panel.

I. Machine Control Panel



- Machine ON/OFF switch.
- Hydraulic ON/OFF switch.
- Mode selector switch – there are different modes available in a CNC machine. They are – EDIT, AUTO, AUTO SLK (Single Block), TAPE, MDI (Manual Data Input), MPG, JOG, REF (Reference).
- Manual Pulse Controller (MPG)/ Hand Pulse Generator (HPG)/ Electronic hand wheel – used for moving the slides manually during setup operations by rotating the hand wheel.
- Selector Switch for selecting the axis (X, Z) during MPG mode.
- Selector Switch for resolution of slide movement in MPG mode – 0.001mm/ 0.01mm/0.1mm per pulse of MPG.
- Selector Switch for spindle speed override from 0% to 120 % from the command value.
- Selector Switch for cutting feed override from 0% to 150 % from the command value.
- Selector Switch for rapid feed override from 0% , 50 %, 100% from the command value.
- Push buttons for moving slides manually in all the directions in JOG mode – +X, -X, +Z, -Z.
- Auto cycle ON button.
- Auto cycle HOLD button.
- Auto cycle RESET button.
- Spindle ON button.
- Spindle Hold button.
- Feed HOLD button.
- Feed RELEASE button.
- Turret position indicator.
- Turret position selector for indexing turret to the required position during setup operations in JOG mode.
- Turret Index ON push button.
- LED indication for
  - Reference point establishment
  - Single block mode
  - Optional stop
  - Dry run
  - Lubricant oil low level
- Machine lamp ON/OFF button
- Chip conveyor ON/OFF button

II. CNC control panel

- Monitor – it will indicate current status of the various machine parameters namely, X and Z position, mode in which machine is functioning at present, the part programme step in which presently running, spindle speed, feed rate, tool number, tool offset number, etc.
- Below the monitor there are soft keys (membrane or tactile keys) to select menus for different modes selected. The control panel consists of keys to communicate with CNC controller and keys/ buttons/ switches for controlling machine movements and functions.
- Communication with CNC controller is performed using soft keys below the monitor and right side of the monitor.
- On the right side of the monitor there is a keypad similar to typewriter keypad which is used for keying in part programs and editing them.
- Page UP, page down keys are provided.

Reference Mode (Starting and Referencing a Machine)

Mohan: I hope you are clear about the control panel and its working and now we can start with the reference mode.

Sailesh: Yes. Please tell me about the reference mode.

Mohan: REF Mode is used for establishing the measuring system of the machine in X and Z axis immediately on switching ON the machine. No other activity is done using this mode.

You know there is a simple procedure for referencing a machine which is as follows:-

1. Before switching ON the machine ensure that power input stabilizer is switched ON and RESET.
2. Switch ON the machine and wait for message to appear in monitor.
3. Switch ON hydraulics (notice the noise generated due to hydraulics)
4. Switch ON the machine lamp.
5. Press RESET button.
6. Press feed RELEASE push button.
7. Check the turret position. If the turret is in the right hand top corner, select JOG mode. Manually move the slide pressing -X and - Z push buttons to bring it to middle of the stroke.
8. If the turret is in the middle of the stroke the above step need not be performed.
9. Select REF mode.
10. Press +X button – the slide starts moving up, stops at the machine reference point and LED indication shows the reference point is established in Z direction.
11. Press +Z button – the slide starts moving towards the right side, stops at the machine reference point and LED indication shows the reference point is established in Z direction.

Mohan: Are you clear about it now?

Sailesh: Yes, let's proceed to the next topic.

JOG Mode (Manual) Operations

Mohan: Now, I will explain about the JOG mode (Manual) Operations.

Sailesh: What is that?

Mohan: All operations done manually in a conventional lathe cannot be performed in CNC lathe like using the hand wheel for performing various operations like threading, grooving, etc.

This mode is mainly provided in the CNC lathe only for setup operations like checking spindle rotation, moving slides for measuring tool offsets, indexing tool turret to bring the required tool in position during measuring tool offset, checking the functioning of coolant motor, conveyor and similar accessories and attachments.

The procedure to use JOG mode is:

1. Select JOG mode in mode selector switch.
2. Press +X, -X, +Z, -Z buttons for moving the slides in the respective direction.
3. Press coolant ON/OFF button to execute the function.
4. Press foot switch for clamping de-clamping of work piece in the chuck.
5. To index the tool turret, slides must be in reference position.
6. To index the tool turret to any required position, select the required turret position in the selector switch and press turret index button. Turret will move to the indexed position.

Sailesh: This is very interesting!

Edit Mode

Mohan: Let's start with the edit mode now.

Sailesh: What is it used for?

Mohan: Edit mode is used for keying in a new part program store it inside the CNC and to edit stored programs.

You know the procedure to keying in new part program is simple and is as follows:-

1. Select EDIT mode.
2. Select PGM (program) in menu key below the monitor.
3. Select NEW in menu key below the monitor.
4. Press O using Alpha numeric key pad. O indicates a program.
5. Assign a number and key in the number for the program using alpha numeric key pad.
6. Press INSERT key in alpha numeric keypad.
7. On pressing INSERT key an empty program with the number keyed in is stored and it is ready for editing.
8. Use the alpha numeric key pad for keying in the program.
9. As and when it is keyed in, the program is stored by pressing INSERT button.

Sailesh: Is the procedure to edit a new part program different from editing a stored part program?

Mohan: Yes!

The procedure to select and edit a stored part program is:

1. Select EDIT mode.
2. Select PGM (program) in menu key below the monitor.
3. Select DIR (directory) in menu key below the monitor.
4. Search for the program to be used using UP or DOWN arrow key in alpha numeric key pad.
5. Press Select key from the menu keypad for editing the part program.
6. Check if the program number displayed on the monitor is correct.
7. Use alpha numeric keypad for keying the codes and alter the programme.

Auto Mode

Mohan: Now I have something interesting for you.

Sailesh: Really? What is it?

Mohan: I will explain you about the auto mode now.

Sailesh: What is an auto mode?

Mohan: Auto mode is used for production of work pieces continuously on auto cycles. During auto cycle run the tool offsets can be adjusted manually in the tool offset page on the monitor.

You know the procedure to select and run a stored part program is:

1. Select EDIT mode.
2. Select PGM (program) in menu key below the monitor.
3. Select DIR (directory) in menu key below the monitor.
4. Search for the program to be used using UP or DOWN arrow key in alpha numeric key pad.
5. Select PAGE DOWN key in alpha numeric key pad for bringing the program to AUTO mode run.
6. Check the program number displayed on the monitor for correctness.
7. Pick up the work piece, load it on the chuck, press foot switch for clamping & close the door.
8. Without chuck in clamp position machine will not auto cycle.
9. Press cycle start button for activating the cycle.
10. Do not open the door/ machine Guard till auto cycle is completed (look for patrol light indication for cycle completion).
11. Chuck will not open when the spindle is running.
12. Open the door, press foot switch for de-clamping the work piece.

In Auto mode the variations in the dimensions of work pieces machined are corrected by adjusting the wear offset for compensation of amount of cutting tool wear. This is applied in both X and Z axis depending on the dimension whether diameter or length.

Mohan: The procedure to adjust tool offset (wear) is:

1. Check the dimension of work piece produced. Compare it with drawing dimensions. Calculate the value to be corrected and note down. Either plus or minus.
2. Identify the tool number for which correction has to be done.
3. Altering the dimension of the diameter is done by altering the value of wear offset in X axis for the relevant tool and length has to be adjusted in the Z axis for the relevant tool.
4. Check whether control system is in auto mode.
5. Press OFFSET soft key in the alpha numeric keypad.
6. Press WEAR in the menu keys.
7. Use UP and DOWN arrow keys in the alpha numeric keypad to locate the tool number for which correction has to be done.
8. Position the cursor in the correct axis.
9. Key in the value noted earlier using alpha numeric keypad along with plus or minus symbol.
10. Press +INPUT in the menu key.
11. Now the alterations are saved to run the auto cycle with corrections.
12. The altered settings will be applied in the next auto cycle.

Auto SBLK (Single Block) Mode

Auto SBLK mode is used for testing new part program by executing the part program block wise. Machine stops after completion of instruction in every block and waits for the operator to press the AUTO start button.

Procedure to use SBLK mode:

1. Select EDIT mode.
2. Select PGM (program) in menu key below the monitor.
3. Select DIR (directory) in menu key below the monitor.
4. Search for the program to be used using UP or DOWN arrow key in alpha numeric key pad.
5. Select PAGE DOWN key in alpha numeric key pad for bringing the program to AUTO mode run.
6. Check the program number displayed on the monitor for correctness.
7. Select SBLK mode.
8. Press cycle start button to run the machine in SBLK mode.
9. Press cycle start button after stoppage of machine in every block.

MDI (Manual Data Input) Mode

Mohan: Sailesh, did you understand all that we have studied so far?

Sailesh: yes!

Mohan: That's great so we can now start with the MDI (Manual Data Input) Mode.

Sailesh: What is that?

Mohan: Operations like running spindle at a specific speed, moving slides in X and Z axis, switching ON or OFF of coolant motor, conveyor, etc can be done in MDI mode like in JOG mode by keying in the respective G code/ M code.

MDI mode is used to check the correct functioning of machine, G and M codes by operator or setter.

Sailesh: are there any other uses of the MDI mode?

Mohan: Yes. The MDI mode is used for internal software data, PLC (program logic controller) data, and electrical signals (IN / OUT status) by electronic maintenance personnel. Operator should not open these menus.

Running machine in MDI mode:

1. Select MDI mode in mode selector switch.
2. Use alpha numeric keypad for keying in any G/M codes and numbers.
3. As and when the keys are pressed they appear on the monitor. Check for the correctness.
4. After completing a block of instructions press EOB (End of Block) soft key in the alpha numeric keypad.
5. Press OUTPUT START button in the alpha numeric keypad for execution of this block.
6. Storing of this program is possible by pressing store soft key appearing in the menu soft key.

Tape Mode

Mohan: Last mode is Tape mode.

Sailesh: What is that?

Mohan: TAPE mode is used to run the machine by feeding the program from an external device through RS 232C cables. In this way the machine becomes a slave to external device for receiving instructions.

Sailesh: Is this mode is used for any special purpose?

Mohan: Yes. This is used wherein multiple machines are connected in a workshop or production shop. Through cabling from a central computer. (FMC – Flexible manufacturing Cell)

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. Explain the different keys on the Fanuc control panel.

2. Write the procedure of referencing of machine.

3. Write the procedure to use AUTO mode.

Worksheet

Choose the correct answer.

1. Practice using the Fanuc control panel and set the machine in the following modes:
   a. Reference Mode ☐
   b. JOG Mode ☐
   c. EDIT Mode ☐
   d. AUTO mode ☐
   e. AUTO SBLK mode ☐
   f. MDI Mode ☐

2. _____ is used to check the correct functioning of machine, G and M codes by operator or setter.
   a. MDI mode ☐
   b. Auto mode ☐
   c. Tape mode ☐
   d. Auto SBLK mode ☐

3. _____ key in alpha numeric key pad is used for bringing the program to AUTO mode run.
   a. Page Up ☐
   b. Page Down ☐
   c. Scroll ☐
   d. Shift ☐

4. What does FMC stands for?

   a. Flexible manufacturing Cell ☐
   b. Flexible machine Cell ☐
   c. Flexible manufacturing Computer ☐
   d. Flexible machine Compute ☐

5. _____ is used for establishing the measuring system of the machine in X and Z axis.

   a. Auto Mode ☐
   b. MDI Mode ☐
   c. JOG Mode ☐
   d. REF Mode ☐

Notes

Answers:

1.

1. To be explained on your own
2. a
3. b
4. a
5. d

G Code and M Code

At the end of this module you will be able to:

- understand G codes and their functions;
- understand M codes and their functions.

Session Plan	
1	G Code and Their Functions
2	M Code and Their Functions
3	General Vocabulary for CNC Turning
4	Key Learnings
5	Worksheet

G Code and Their Functions

Today, Sailesh has seen the lathe machine in reality. He is a little confused to see a small chart of codes pasted on the machine. So he asks Mohan:

Sailesh: What is written in this chart? I cannot understand anything written on it?

Mohan: This is a chart of codes called G code.

Sailesh: G codes! What is that?

Mohan: G codes are important codes. They are also called preparatory codes and are commands in a CNC program.

Sailesh: Then why are they called G codes?

Mohan: Very simple, it is because all these codes begin with the letter G.

Sailesh: Oh! I See. Where do I use them?

Mohan: Well Sailesh, G codes are specific general command telling the machine tool what type of action to perform, such as:

- Rapid move (transport the tool through space to the place where it is needed for cutting; do this as quickly as possible)
- Controlled feed move in a straight line or arc
- Series of controlled feed moves that would result in a hole being bored, a work piece being cut to a specific dimension, or a profile a shape added to the edge of a work piece.

Sailesh: What are the common G codes?

Mohan: I will give you a chart of G codes:

Commonly used G-Codes for CNC Lathes are as follows:

00 - Rapid Positioning	G01 - Linear Interpolation
G02 - Circular Interpolation CW	G03 - Circular Interpolation CCW
G04 – Dwell	G07 – Feed rate Sine Curve Control
G10 - Data Setting	G11 - Data Setting Cancel
G17 - XY Plane Selection	G18 - XZ Plane Selection
G19 - YZ Plane Selection	G20 - Input in Inches
G21 - Input in Metric	G22 - Stored Stroke Check ON
G23 - Stored Stroke Check OFF	G27 - Reference Point Return Check
G28 - Automatic Zero Return	G29 - Return from Zero Position
G30 - 2nd Reference Point Return	G31 - Skip Function
G32 - Thread Cutting	G36 - Automatic Tool Compensation
G40 - Tool Compensation Cancel	G41 - Tool Compensation Left
G42 - Tool Compensation Right	G46 - Automatic Tool Compensation
G50 - Coordinate System Setting	G52 - Local Coordinate System Setting
G53 - Machine Coordinate System Setting	G54 – Work piece Coordinate Setting 1
G55 – Work piece Coordinate Setting 2	G56 – Work piece Coordinate Setting 3
G57 – Work piece Coordinate Setting 4	G58 – Work piece Coordinate Setting 5
G59 – Work piece Coordinate Setting 6	G61 - Exact Stop Check Mode
G62 - Automatic Corner Override	G63 - Tapping Mode
G64 - Cutting Mode	G65 - User Simple Macro Call
G66 - User Modal Macro Call	G67 - User Modal Macro Call Cancel
G70 - Finishing Cycle	G71 - Turning Cycle
G72 - Facing Cycle	G73 - Pattern Repeating Cycle
G74 - Drilling Cycle	G75 - Grooving Cycle
G76 - Threading Cycle	G80 - Canned Cycle Cancel
G83 - Face Drilling Cycle	G84 - Face Tapping Cycle
G85 - Face Boring Cycle	G87 - Side Drilling Cycle
G88 - Side Tapping Cycle	G89 - Side Boring Cycle
G90 - Absolute Positioning	G91 - Incremental Positioning
G92 - Threading Cycle	G94 - Face Turning Cycle
G96 - Constant Surface Speed Control On	G97 - Constant Surface Speed Control Off
G98 – Feed rate Per Time	G99 – Feed rate Per Revolution
G107 - Cylindrical Interpolation	G112 - Polar Coordinate Interpolation Mode On

G113 - Polar Coordinate Interpolation Mode Off	G250 - Polygonal Turning Mode Cancel
G251 - Polygonal Turning Mode	

Sailesh: They are so many… do I need to learn them??

Mohan: No, Sailesh! They are pasted on the machine in a form of chart that you can refer any time.

Sailesh: Oh! That is really convenient!

M Code and Their Functions

After learning about G codes, Sailesh is now ready to learn about M codes.

Mohan: Today, I will tell you about M codes.

Sailesh: Are M codes different from G codes?

Mohan: Yes, Sailesh. G codes and M codes are different. The M codes are miscellaneous codes and are commands in a CNC program; they begin with the letter M.

Sailesh: Is there any chart of M codes available on or around the machine?

Mohan: Yes, the chart of M codes is available in the machine room for your reference.

Let me give you a chart of M codes:

Commonly used M-Codes for CNC Lathes

00 - Program Stop

M01 - Optional Program Stop

M02 - Program End

M03 - Spindle Clockwise

M04 - Spindle Counter Clockwise

M05 - Spindle Stop

M07 - Coolant 1 On

M08 - Coolant 2 On

M09 - Coolant Off

M30 - End Program, Return to Start

M98 - Call Subprogram

Sailesh: I think with experience I will memorize these codes.

Mohan: You are right Sailesh! While working on the lathe machine you will memorize these codes without learning them.

Sailesh: Thank you Mohan!

General Vocabulary for CNC Turning

Mohan: Let me give you a chart of general vocabulary for CNC turning machine.

General Vocabulary For CNC Turning	
Vocabulary Term	Definition
Absolute Coordinates	A series of numerical positions that are calculated from a fixed point of origin.
Address	The letter used within a word that signals the essential function of the word.
Axes	An imaginary line that passes through the center of an object. Axes are used to measure the distances of objects in the Cartesian coordinate system.
Block	A single line of the part program that consists of words.
Blueprint	A design representing the dimensions of a specific work piece that is used to create the part program.
Cartesian Coordinate System	The numerical system that describes the location of an object by numerically expressing its distance from a fixed position along three linear axes.
Computer Assisted Programming	The use of computer software that facilitates the development of part programs. These software applications create an image of the work piece and develop the program code from information that the programmer inputs.
Conversational Interface	An interface that asks the user a series of questions and then translates the user's answers into part program code.
Cutter Offset	A predetermined distance from the surface of the work piece that allows for the safe and rapid movement of the cutting tool between cutting operations.
Dry Run	A trial run of the part program without any parts or cutting fluids.
Electronics Industries Association	An organization that sets standards for technology industries.
F code	A word that determines the feed during a cutting operation.
Feed	The rate that the cutting tool travels along the surface of the work piece.
G code	A word that determines the type of operation performed on the machine.
Incremental Coordinates	A series of numerical positions that use the previous position as the point of origin.

M code	A word used to signal an action from a miscellaneous group of commands. M codes change cutting tools, turn on or turn off the coolant, spindle, or work piece clamps, etc.
Machining Center	A mill that is controlled by a computer running programs driven by numerical data. Machining centers are also capable of performing a wide range of hole-making operations.
N Code	A word that acts as the name or title for a program block.
Operator	The person responsible for running the CNC machine on the shop floor.
Part Program	A series of numerical instructions used by a CNC machine to perform the necessary sequence of operations to machine a specific workpiece.
Part Programmer	The person responsible for the creation of a part program. The part programmer translates the workpiece design into program instructions for the CNC machine.
Program Zero	The position that acts as the origin for the part program of a particular work piece. This position is unique to each work piece design, and it is selected by the part programmer.
S code	A word that determines the speed during a cutting operation.
Speed	The rate at which the cutting edge of the tool moves past the work piece surface at the point of contact.
T code	A word that determines which specific cutting tool will be selected during a tool change.
Tool Changer	A device on a machining center that arranges multiple cutting tools in order and then positions these cutting tools for replacement.
Tool Path	A series of program blocks that describes the movements of a single cutting tool.
Turning Center	A lathe that is controlled by a computer running programs driven by numerical data. Sophisticated turning centers can also perform a variety of drilling and milling operations.
Turret	The component of a lathe that holds a number of cutting tools. The turret rotates to place tools in the cutting position.
Word	The pairing of a letter address and a numerical value.
X code	A word that describes a specific position along the X-axis.
X-axis	The linear axis representing motions and positions that travel the longest distance parallel to the worktable.
Y code	A word that describes a specific position along the Y-axis.

Y-axis	The linear axis representing motions and positions that travel the shortest distance parallel to the worktable.
Z code	A word that describes a specific position along the Z-axis.
Z-axis	The linear axis that represents motions and positions perpendicular to the worktable. The Z-axis is always parallel to the spindle.

Sailesh: You have earlier used many of these terms while telling various operations of lathe machine.

Mohan: Yes, that's true. You will also get familiar with these terms as and when you work on the machine.

So, friends! Remember these are important codes and vocabulary terms which will be used every time you work on a lathe machine.

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. What are G codes? Write about the functions of any 5 G codes.

2. What are M codes? Write about the functions of any 5 M codes.

Worksheet

1. *Mohan is testing the knowledge of Sailesh and his friends on the general vocabulary. Give definition of any 5 terms given below:*

a.	Blue print	b.	Turret	c.	X axis
d.	Y axis	e.	Word	f.	S code

2. *Sailesh has written some statements complete them by filling in the blank spaces:*

a. _____ is the code for spindle clockwise.

b. _____ is the code for face drilling cycle.

c. G88 is the code for _____.

d. G10 is the code for _____.

e. _____ is the code that allows you to skip function

Notes

Answers:

1.

a. Blueprint -A design representing the dimensions of a specific work piece that is used to

create the part program.

b. Turret- The component of a lathe that holds a number of cutting tools. The turret rotates to place tools in the cutting position.

c. X-axis - The linear axis representing motions and positions that travel the longest distance parallel to the worktable.

d. Y-axis- The linear axis representing motions and positions that travel the shortest distance parallel to the worktable.

2.

a.	M03	b.	G83	c.	Side tapping cycle
d.	Data setting	e.	G31		

Part Program Development

At the end of this module you will be able to:

- explain how a part program works.

Session Plan	
1	Part Program
2	Key Learnings

Part Program

Sailesh is inquisitive to know what a programmer keeps on feeding in the machine. He decides to ask Mohan:

Sailesh: Mohan, I have seen that the programmer enters something very long in the machine. What is that?

Mohan: It is called a part program.

Sailesh: What is a part program? I have never heard about it.

Mohan: The part program is the lifeline for performing an operation on CNC machine. The machine cannot turn the product accurately without the part program.

The program has to be based on the sequence of operations planned. The program necessarily requires the input of tool offsets and tool numbers to be fed for the program to function completely to make the work piece as per requirement.

Following is the program that has been written to turn this particular piece in three different settings:

S.No : O0400	**1ST SET UP PROGRAM**
G28U0.W0	Reference Point Return
N3	Sequence Number
T0303	Tool no 3 with Offset no 3
G50S2000	Maximum R.P.M clamp at 2000
G96S150M4	Constant surface Speed with Direction of Rotation of Spindle Anti-clockwise at cutting speed 150m/min
G0Z50	Rapid Movement of tool in Z axis 50mm distance before work piece zero point.
M8	Coolant on
X52.0	Rapid Movement of tool in X axis 52mm Dia.
G0Z2.	Rapid Movement of tool in Z axis 2mm distance before work piece zero point.
	Facing and OD Turning Operation Starts

G94X0Z0.5F0.15	Facing Cycle Starts with feed 0.15mm / revolution
Z0.1	Tool moves in Z axis 0.1mm distance before work piece zero point.
Z0	Tool moves in Z axis to work piece zero point.
G0X51.0Z2.0	Rapid movement of tool in X to 51mm Dia and 2mm distance in Z direction before work piece zero point.
G71U0.3R0.5	Canned cycle for roughing starts with depth of cut 0.3 mm R is escaping amount.
G71P10Q20U0.2W0.05F0.15	P is the start point of profile Q is end point of profile U&W finishing allowance in X and Z axis direction.
N10G01X42.5F0.15	Linear movement of tool in X axis to 42.5 mm dia with feed 0.15mm/ revolution.
G01Z-75.0	Linear movement of tool in Z axis to 75mm distance after Work Piece zero point.
G01X48.5	Linear movement of tool in X axis with 48.5mm dia.
N20G01Z-90.0	Linear movement of tool in Z axis to 90 mm distance after work piece zero point.
G0Z10.	Rapid movement of tool in Z direction to 10 mm distance before work piece zero point.
G0X100.Z100.M9	Rapid movement of tool in X axis to 100mm Dia and Z direction to 100mm distance before work piece zero point and coolant off .
M5	Spindle Stop
G28U0W0	Reference Point Return
M1	Optional Stop.
	End of Facing and OD Turning Tool Program
	Start of Centre Drill program
N4	Sequence No
T0404	Tool no 4 with Offset no 4
G97S1000M4	Constant surface Speed cancel and Spindle Rotation Anti Clockwise with 1000rpm
G0X0Z10	Rapid Movement of X axis to Zero and Z axis to 10mm distance before Work Piece Zero Point.
G01Z2.F.5	Linear Movement of Z axis to 2mm distance before Work Piece Zero Point with feed 0.5 mm./revolution
G1Z-5.0F0.02	Linear Movement of Z axis to 5mm distance after Work Piece Zero Point with feed 0.02 mm./revolution
G01Z5.0F0.5	Linear Movement of Z axis to 5 mm distance before Work Piece zero Point with feed 0.5 mm./revolution
G0Z20.	Rapid Movement of Z axis to 20mm distance before Work Piece Zero Point.
M5	Spindle Stop.
M9	Coolant Off
G28U0.W0.	Return to Reference Point.

	End of Central Drill Program
	Start of Drilling Program.
N7	Sequence Number
T0707	Tool no 7 with offset No 7.
(Drill 20.0MM)	
G97S350M04	Constant surface Speed cancel with Spindle Rotation Anti Clockwise at 350r.p.m.
G0X0.Z5.M8	Rapid Movement of X axis to zero and Z axis to 5mm distance before Work Piece Zero point and coolant Off.
G1Z1.F.1	Linear Movement of Z axis to 1mm distance before Work Piece Zero Point with feed 0.1mm/Revolution.
G1Z-41.0.F0.02	Linear Movement of Z axis to 41mm distance after Work Piece Zero Point with feed 0.02 mm/Revolution.
G1Z2.F.8	Linear Movement of Z axis to 2mm distance before Work Piece Zero Point with feed 0.8mm/Revolution.
G0Z20.M5	Rapid Movement of Z axis to 20mm distance before Work Piece Zero Point and Spindle Stop.
M9	Coolant Stop.
G28U0.W0.	Return to Reference Point
M1	Optional Stop.
	End of Drilling Program.
	Start of Boring Tool Program.
N2	Sequence number.
T0202 (BORING)	Tool no 2 with offset no 2.
G50S2000	Maximum R.P.M Clamp at 2000
G96S150M3	Constant surface Speed with Spindle Rotation Clockwise & cutting speed 150m/min.
G0Z50	Rapid Movement of Z axis to 50mm distance before Work Piece Zero Point.
M8	Coolant On
G0X19.5	Rapid Movement of X axis to 19.5mm Diameter.
Z2.0	Rapid Movement of Z axis to 2mm distance before Work Piece Zero Point.
G90X20.5Z-26.95F0.1	Turning Cycle Starts with First Cut at 20.5mm dia and feed 0.1mm/ Revolution.
X21.0	2nd cut at 21mm dia
X21.5	3rd cut at 21.5mm dia.
X22.0	4th cut at 22.0mm dia
X22.5	5th cut at 22.5mm dia.
X23.0	6th cut at 23 mm dia.
X23.5	7th cut at 23.5mm dia.
X24.0	8th cut at 24 mm dia.

X24.5	9th cut at 24.5mm dia.
X25.0	10th cut at 25 mm dia
X25.5	11th cut at 25.5mm dia.
X26.0	12th cut at 26 mm dia.
X26.5	13th cut at 26.5mm dia.
X27.0	14th cut at 27mm dia.
X27.5	15th cut at 27.5mm dia.
X28.0	16th cut at 28mm dia.
X28.5	17th cut at 28.5mm dia.
X29.0	18th cut at 29mm dia.
X29.5	19th cut at 29.5mm dia.
X30.0	20th cut at 30.0mm dia.
X30.5	21st cut at 30.5mm dia.
G0X34.6	Rapid Movement of X axis to 34.6mm dia.
G1Z0.1F0.1	Linear Movement of Z axis to 0.1mm distance before Work Piece Zero point with 0.1mm/Revolution.
G1X31.0C1.8F0.1	Linear Movement of X axis of 31 mm dia with chamfer 1.8mm and Feed 0.1mm/Revolution.
G1Z-27.0	Linear movement of Z axis to 27 mm distance after Work Piece Zero Point.
G1X20.0C1.0	Linear movement of X axis to 20mm distance with chamfer 1.0mm.
G1Z-28.5	Linear movement of Z axis to 28.5mm distance after Work Piece Zero Point.
G1U-0.2F0.1	Linear movement of X axis to 0.2mm downwards with feed 0.1mm/ Revolution.
G0Z10.M9	Rapid Movement of Z axis to 10 mm distance before Work Piece Zero point and coolant off.
M5	Spindle Stop.
G28U0W0	Return to Reference Point.
	End of Boring Tool Program.
M30	Program stop and rewind. – Change of Side of Workpiece.
S.No : 00401	2nd SETUP PROGRAM
	Start of Facing , O.D. Turning and Taper Turning Program
G28U0.W0	Return to Reference Point
N3	Sequence number
T0303	Tool no 3. With offset no . 3
G50S2000	Maximum R.P.M clamp at 2000
G96S150M4	Constant surface Speed and Spindle Rotation Anti Clockwise with cutting speed 150m/min.
G0Z50	Rapid Movement of Z axis to 50 mm distance before Work Piece Zero Point.
M8	Coolant On
X52.0	Tool Moves in X axis to 52mm dia.

G0Z2	Rapid Movement of Z axis to 2 mm distance before Work Piece Zero Point.
G94X0Z0.5F0.15	Facing Cycle starts with feed 0.15mm /rev and Z moves to 0.5mm before Work Piece Zero Point.
Z0.1	Tool moves in Z axis to 0.1mm distance before Work Piece Zero Point.
G0X51.0Z0.5	Rapid Movement of X axis to 51mm dia and Z axis to 0.5mm before Work Piece Zero Point.
G71U0.8R0.5	Canned Cycle for Roughing starts with depth of cut 0.8mm Radius is escaping amount.
G71P10Q20U0.2W0.05F0.15	P is start point Q is end point of profile U&W finishing allowance in X and Z axis direction.
N10G01X28.0F0.15	Linear Movement of tool in x axis to dia 28mm with feed 0.15mm/ Revolution.
G01X38.0Z-30.0	Taper Movement of tool in X and Z axis.
G01Z-80.0	Linear movement of Z axis to 80 mm from Work Piece Zero point.
G01X48.0	Linear Movement of X axis to 48mm dia.
N20G01Z-92.0	Linear Movement of Z axis to 92mm from Work Piece Zero point.
G0Z10	Retract of tool in z axis to 10mm before Work Piece Zero point.
X30.0	Rapid movement in X axis to dia 30mm.
G0Z2	Rapid movement in Z axis to 2mm before Work Piece Zero point.
G94X0Z0F0.15	Facing Cycle Starts with feed 0.15mm/Revolution.
G1X27.5F1.0	Tool Moves in X axis to dia 27.5 mm with feed 1.0mm/Revolution.
Z0.2	Tool Moves in Z axis to 0.2 mm before Work Piece Zero point.
G01Z0F0.2	Linear movement of Z axis to zero point of Work Piece with feed 0.2mm/Revolution
G01X38.0Z-30.0F0.15	Taper movement of X and Z axis with feed 0.15mm/Revolution
G01Z-80.0	Linear movement of Z axis to 80mm from Work Piece Zero point.
G01X47.5C1.0	Tool moves in X axis to dia. 47.5 mm with chamfer C=1mm.
G01Z-95.0	Linear movement of Z axis to 95mm from Work Piece Zero point.
G0U5.0	Rapid incremental of 5mm in X axis. (U = used for incremental value).
G0X100.Z100.M9	Rapid movement of tool in X axis to dia 100mm and in Z axis to 100mm before Pork Piece Zero point.
M5	Spindle Stop
G28U0W0	Return to Reference Point.
M1	Optional Stop.
	End of Facing , O.D.Turning and Taper Turning Program.
	Start of Center Drilling Program
G28U0.W0.	Return to Reference Point.
N4	Sequence No
T0404	Tool no 4 with offset no 4.
CENTER DRILL	

G97S500M04	Constant surface Speed cancel and Spindle Rotation Anti Clockwise at 500 rpm
G0X0.Z5.M8	Rapid Movement of X axis to zero point and Z axis to 5mm before Work Piece Zero point and Coolant On
G1Z1.F.1	Linear Movement of Z axis to 1 mm before Work Piece Zero Point with feed 0.1mm/Revolution.
G1Z-5.0F0.02	Linear Movement of Z axis to 5mm from Work Piece Zero point with feed 0.02mm/Revolution.
G1Z2.F.8	Retract of drill in Z axis to 2mm before Work Piece Zero point with feed 0.8mm/Revolution.
G0Z20.M5	Rapid Movement in Z axis to 20 mm before Work Piece Zero point and Spindle Stop.
M9	Coolant off
G28U0.W0	Return to reference point.
M1	Optional stop.
	End of Center Drilling Program.
	Program stop and rewind. – Change of Side of Workpiece.
S.No : 00402	3rd SETUP PROGRAM
	Start of OD Turning Tool Program
G28U0.W0.	Return to Reference Point
N3	Sequence Number
T0303	Tool number 3 with offset number 3.
G5 0S2000	Maximum R.P.M clamp at 2000
G96S150M4	Constant surface Speed and rotation of spindle anti clock wise with cutting speed 150m/min.
G0Z50.	Rapid movement of Z axis 50mm distance before work piece zero point.
M8	Coolant ON.
X51.0	Rapid movement of X axis to 51mm dia.
G0Z2.	Rapid movement in Z axis 2 mm distance before work piece zero point.
G94X28.0Z0F0.15	Facing cycle
G0Z2.0	Rapid movement in Z axis to 2mm distance before work piece Zero point.
X36.0	Rapid movement in X axis to 36 mm dia
G01Z0.1F0.2	Linear movement in Z Axis to 0.1mm distance before work piece zero point with feed 0.2 mm/revolution
G01X38.0C1.0F0.15	Linear movement in X Axis to 38mm dia with chamfer 1 mm with feed 0.15mm/revolution.
G01Z-50.0	Linear movement in Z Axis to 50mm distance after work piece zero point.
G01X42.0C1.5	Linear movement in X Axis to 42mm dia with chamfer 1.5 mm

G01Z-75.0	Linear movement in Z Axis to 75mm distance after work piece zero point.
G01X48.0C1.2	Linear movement in X Axis to 48mm dia with chamfer 1.2mm
G01Z-77.0	Linear movement in Z Axis to 77mm after work piece zero point.
G0U5.0	Retract of tool 5mm in X axis
G0X100.Z100.M9	Rapid movement of X and Z axis to 100mm each before work piece zero point & coolant OFF
M5	Spindle stop.
G28U0W0	Return to reference point.
M1	Optional Stop
	End of OD turning tool program
N7	Start of OD Grooving program
G0T0707	Tool number 7 with offset number 7
G50S1200	Maximum R. P. M. clamp at 1200
G97S500M3	Constant surface Speed cancel with Spindle Rotation Clockwise at 500 rpm
G0Z50.	Rapid Movement in Z axis to 50 mm distance before Work Piece Zero point.
M8	Coolant ON
X49.0	Rapid Movement of X axis to 49 mm dia.
G0Z2.	Rapid Movement of Z axis to 2mm distance before Work Piece Zero point.
G1Z-75.0F2.0	Linear Movement of Z axis to 75mm distance with feed 2.0 mm/ Revolution.
G1X48.2F0.5	Linear Movement of X axis to 48.2 mm dia with feed 0.5 mm/ Revolution.
X40.0F0.015	Linear Movement of X axis to 40mm dia with feed 0.015 mm/ Revolution.
G4X0.5	Dwell Time of 0.5 second
G1X49.0F1.0	Linear Movement of X axis to 49mm dia with feed 1.0 mm/Revolution.
G0X60.0	Rapid Movement of X axis to 60 mm Dia.
G0Z50.M9	Rapid Movement of Z axis to 50 mm distance before Work Piece Zero point and coolant Off.
M5	Spindle stop
G0G28U0W0	Return to Reference point
M1	Optional Stop
	End of O.D. Grooving program.
	START OF BORING TOOL PROGRAM
N5	Sequence number
T0505	Tool number 5 with offset number 5
(BORING)	

G50S2000	Maximum R.P.M. clamp at 2000
G96S150M4	Constant surface Speed with Spindle Rotation Anti Clockwise& cutting speed 150m/min.
G0Z50.	Rapid Movement of Z axis to 50mm distance before Work Piece Zero point.
M8	Coolant On.
G0X34.0	Rapid Movement of X axis to 34 mm dia
Z2.0	Rapid Movement of Z axis to 2.0 mm distance before Work Piece Zero point.
G1Z0.1F0.1	Linear Movement of Z axis to 0.1mm before Work Piece Zero point.
G1X31.5C1.0F0.1	Linear Movement of X axis to 31.5mm dia with chamfer 1mm with feed 0.1 mm/Revolution.
G1Z-27.0	Linear Movement of Z axis to 27 mm distance after Work Piece Zero point
G1X20.5C2.0	Linear Movement of X axis to 20.5mm dia with chamfer 2mm
G1Z-30.0	Linear Movement of Z axis to 30mm distance after Work Piece Zero point.
G1U-0.2F0.1	Retract of Tool 0.2mm down in X axis with feed 0.1 mm/Revolution.
G0Z10.M9	Rapid Movement of Z axis to 10mm distance before Work Piece Zero point and coolant Off.
M5	Spindle Stop.
M1	Optimal Stop.
	END OF BORING TOOL PROGRAM
	START OF INTERNAL GROOVE TOOL PROGRAM
G28U0W0	Return to Reference point.
N6	Sequence number
G0T0606	Tool Number 6 with Offset Number 6
G50S1200	Maximum R.P.M Clamp at 1200
G97S500M3	Constant surface Speed cancel with Spindle Rotation Clockwise 500 rpm
G0Z50	Rapid Movement of Z axis to 50mm distance before Work Piece Zero point.
M8	Coolant ON
X20.0	Rapid Movement of X axis to 20mm dia
G0Z2	Rapid Movement of Z axis to 2mm distance before Work Piece Zero point.
G1Z-27.0F2.0	Linear Movement of Z axis to 27mm distance after Work Piece Zero point with feed 2mm/Revolution.
G1X31.0F0.05	Linear Movement of X axis to 31mm dia with feed 0.05mm/Revolution.
X34.0F0.015	Linear Movement of X axis to 34mm dia with feed 0.015mm/ Revolution.
G4X0.5	Dwell Time for 0.5 second

G1X20.0F1.0	Linear Movement of X axis to 20 mm dia with feed 1mm/Revolution.
G0Z50.0	Rapid Movement of Z axis to 50mm distance before Work Piece Zero point.
G0G28U0W0M5	Return to Reference Point and Spindle Stop
M1	Optional Stop
	END OF INTERNAL GROOVE TOOL PROGRAM
	START OF EXTERNAL THREADING TOOL PROGRAM
N1	Sequence number
T0101	Tool number 1 Offset number 1
G97S1500M4	Constant surface Speed cancel with Spindle Rotation Anti Clockwise at 1500 rpm
G0Z50	Rapid Movement of Z axis to 50mm distance before Work Piece Zero Point .
M8	Coolant ON
G0X43.0	Rapid Movement of X axis to 43mm dia.
G01Z5.0F0.5	Linear Movement of Z axis to 5.0 mm distance before Work Piece Zero point with feed 0.5mm/Revolution.
G1Z-45.0F0.5	Linear Movement of Z axis to 45 mm distance after Work Piece Zero point with feed 0.5mm/Revolution.
	THREADING CYCLE STARTS
G92X41.5Z-73.5F1.5	Threading Cycle Starts with pitch 1.5mm and First pass at 41.5mm dia.
X41.0	2nd pass at 41mm dia
X40.9	3nd pass at 40.9mm dia
X40.8	4th pass at 40.8mm dia
X40.7	5th pass at 40.7mm dia
X40.6	6th pass at 40.6mm dia
X40.55	7th pass at 40.55mm dia
X40.52	8th pass at 40.52mm dia
X40.5	9th pass at 40.5mm dia
X40.5	Idle pass at 40.5mm dia
G0Z10.M9	Rapid Movement of Z axis to 10mm distance before Work Piece Zero point and Coolant Off
M5	Spindle Stop
G28U0W0	Return to Reference Point
M1	Optional Stop
	END OF EXTERNAL THREADING TOOL PROGRAM
	START OF INTERNAL THREADING TOOL PROGRAM
N8	Sequence number
T0808	Tool number 8 with offset number 8

G97S2000M4	Constant surface Speed cancel with Spindle Rotation Anti Clockwise at 2000 rpm
G0Z50	Rapid Movement of Z axis to 50 mm distance before Work Piece Zero Point.
M8	Coolant ON
G0X31.0	Rapid Movement of X axis to 31mm dia
Z5.0	Rapid Movement of Z axis to 5mm distance before Work Piece Zero point.
G92X32.0Z-26.5F1.5	Threading Cycle Starts with pitch1.5 and first pass at 32 mm dia
X32.5	2nd pass at 32.5 mm dia
X32.6	3rd pass at 32.6 mm dia
X32.7	4th pass at 32.7 mm dia
X32.8	5th pass at 32.8 mm dia
X32.9	6th pass at 32.9mm dia
X32.95	7th pass at 32.95 mm dia
X32.98	8th pass at 32.98 mm dia
X33.0	9th pass at 33.0mm dia
X33.0	Idle pass at 33.0mm dia
G0Z10.M9	Rapid Movement of Z axis to 10mm distance before Work Piece Zero point and Coolant OFF.
M5	Spindle Stop
G28U0W0	Return to Reference point
M1	Optional stop
	END OF INTERNAL THREADING TOOL PROGRAM
	START OF PARTING TOOL PROGRAM
N7	Sequence number
G0T0707	Tool number 7 with Offset number 7
(PARTING)	
G50S1200	Maximum R.P.M clamp at 1200
G97S500M3	Constant surface Speed cancel with Spindle Rotation Clockwise at 500 rpm
G0Z50.	Rapid Movement of Z axis to 50mm distance before Work Piece Zero point.
M8	Coolant ON
X39.0	Rapid Movement of X axis to 39mm dia.
G0Z2.	Rapid Movement of Z axis to 2mm distance before Work Piece Zero point.
G1Z-15.0F2.0	Linear Movement of Z axis to 15mm distance after Work Piece Zero point with feed 2 mm/Revolution.
G1X38.0F0.1	Linear Movement of X axis to 38mm dia with feed 0.1mm/Revolution.

X33.0F0.015	Linear Movement of X axis to 33mm dia with feed 0.015mm/ Revolution.
G4X0.5	Dwell time for 0.5 second.
G1X39.0F1.0	Linear Movement of X axis to 39mm dia with feed 1mm/Revolution.
G0X60.0	Rapid movement of X axis to 60mm dia
G0Z50.M9	Rapid movement of Z axis to 50mm distance before Work Piece Zero point and Coolant OFF
G0G28U0W0M5	Return to Reference point and Spindle Stop
M30	End of program & rewind
	END OF PARTING TOOL PROGRAM
	END OF ALL PROGRAMS

Sailesh: Thank you Mohan. Hey friends, setting the part programme is the programmer's task so, we don't have to do anything with it. This is session is just to tell you that the machine operates with a particular program. This program is both case and space sensitive, a minor mistake in the program can change the entire working of the machine. So, we need to be very careful with it.

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. What is a Part program?

Notes

Throw Away Insert Tools

At the end of this module you will be able to:

- choose the appropriate throw away insert tools.

Session Plan	
1	Throw Away Insert Tools
2	Types of Throw Away Insert Tools
3	Throw Away Bits
4	Key Learnings
5	Worksheet

Throw Away Insert Tools

Sailesh is looking at various kinds of tools kept in the workshop. He is confused about, the function of each tool. He asks Mohan to help him.

Sailesh: Mohan, I know these tools are used in Lathe Machine, but can you tell me what are they called?

Mohan: Yes, that's right; these tools are called Throw Away Insert Tools.

Sailesh: What throw away!! Do we need to throw these tools?

Mohan: No, no Sailesh! Throw away insert tools are turning operation tools. They are considered a very important input for quality as well as efficiency of turning apart from the purpose/shape of turning required.

Types of Throw Away Insert Tools

Mohan gives Sailesh more information on the throw away Insert tools.

Sailesh: Mohan, can you tell me which tool should I select?

Throw Away Insert Tool Bit

Mohan: The shape of tools, geometry and use depends upon material from which they are made, following basic types of tools are used for turning operations:

1. High speed steel tools.
2. Tungsten carbide tools
3. Ceramic tools
4. Diamond tools

Sailesh: The selection of perfect type of throw away insert tool helps us to improve the quality of the finished product. Am I right Mohan?

Mohan: Yes, you are right to some extent. The requirement of machining has improved for delivery of higher accuracy, faster operations, and very good finish. Hence for most of the operations tungsten carbide tool bits are preferred tooling material. In common language they are also known as Throw Away Insert Tools.

Throw Away Bits

Mohan: The throw able bits are available in:

1. Tungsten carbide bits
2. Titanium nitride coated tungsten carbide bits
3. Ceramic bits.

Throw Away Insert Tool Bit

Sailesh: I think I have seen Tungsten carbide bits in our workshop.

Mohan: Yes Sailesh; the tungsten carbide bit is very popular and is used for almost all applications.

For harder materials you need to use Titanium nitride coated tungsten carbide bits because the normal bits find it difficult to machine and ceramic bits are used for hardened material turning.

Sailesh: Are all these types throw able ?

Mohan: Exactly. All the three types that I have mentioned are throw able bits that cannot be reground and reused. Thus, once the edge wears out or breaks the only option is to replace the same.

Sailesh: Ah… That's why they are called Throw Away insert tools…

Mohan: Yes, I can see now that you have understood the concept well.

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. What are throw away tools? Name them.

2. What are throw away bits and its types?

Worksheet

1. *Sailesh is trying to explain the Throw Away Insert Tools to his co workers. State whether his statements are true or false:*

   a. Throw away insert tools are important input for quality as well as efficiency of turning apart from the purpose/shape of turning required. _____

   b. Tools shape geometry and use depends upon material from which they are made. _____

   c. High speed steel tools are a type of basic throw away insert tools. _____

   d. In operations tungsten carbide tool bits are the preferred tooling material. _____

   e. Ceramic bits are used for hard materials. _____

2. *Sailesh is testing the knowledge of all workers so he asks a few questions. Help him by giving appropriate answer in just one or two words.*

   a. Which kind of bit is used for hardened materials?

   b. Which is the most commonly used bits that are also known as normal bits?

   c. Can any of the throw able bits be re-grounded and reused?

   d. Titanium nitride coated tungsten carbide bits are used for which materials?

   e. What is the only option in case its edge breaks or wears out?

Notes

Answers:

1.

a.	True	b.	True	c.	True
d.	True	e.	False		

2.

a.	Ceramic bits	b.	Tungsten carbide bit.	c.	No
d.	Harder materials.	e.	Replace the bit		

Mounting of Tool Holders on Turret

At the end of this session, you will be able to:

- explain features of tool turret with regard to mounting of various tools, external tool holders and boring bars;
- mount tool holder on the Turret.

Session Plan	
1	What is a Turret
2	Mounting of Tool Holders on the Turret
3	Key Learnings
4	Worksheet

What is a Turret

Today, Mohan takes Sailesh to a lathe machine to show him its various parts. On seeing the lathe machine, Sailesh gets excited and asks Mohan about it:

Sailesh: Mohan what is this wheel kind of thing on the machine?

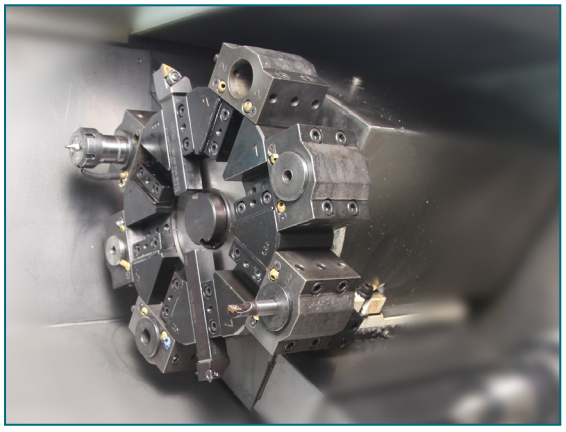
Mohan: It is called a Turret.

Sailesh: What is a Turret and for what is it used?

Mohan: Turret is a freely indexed movable wheel where different type of tooling can be fixed and on revolving the turret; the required type of tool comes into operation.

Sailesh: That's interesting!

Mohan: The good part of Turret wheel is that its circumference is divided in such a way that after indexing each tool will exactly realign itself matching the center axis of the lathe machine.



Turrent CNC Lathe

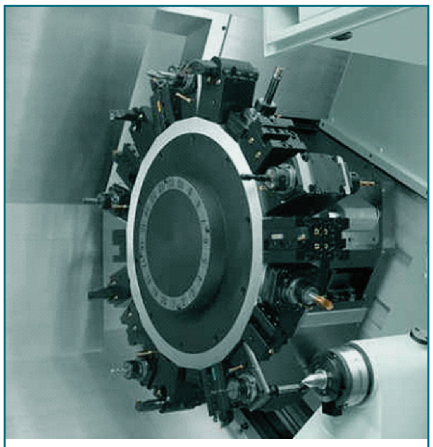
Mounting of Tool Holders on the Turret

After learning about Turret, now it's time to know about the mounting of tool holders on the Turret.

Mohan: Do you know that the turret does not recognise the difference between the tool tip height with respect to its center?

Sailesh: Then how do we work with it?

Mohan: As the turret does not recognize the difference between the tool tip height with respect to its center while setting the tools on Turret the operator must take utmost care. It is his work to ensure that the cutting tip of each tool perfectly matchs the center axis of the lathe.



CNC Lathe

The setting has to be done in such a way that irrespective of the tool being on turret it comes to the point of operation. The setting must be done specifically for this operation and this tool.

Sailesh: Oh! That means setting is it right is extremely crucial. As an operator I must work with utmost care and caution.

Mohan: Yes, you are absolutely right!

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. What is a Turret?

2. Describe the role of an operator in mounting of Tool Holders on Turret?

Worksheet

1. *Complete the following statement given by Sailesh about turret and mounting of tool holders on turret to all his co workers:*

   a. _____ is a freely indexed movable _____.

   b. On _____ the turret the required type of tool comes into operation.

c. The good part of the turret is that it's _____ is so divided that after indexing each tool it will realign itself matching the center axis of the lathe machine.

d. The turret does not _____ the difference between the _____ height with respect to its center.

e. An operator must ensure that the _____ of each tool shall perfectly match the _____ of the lathe.

2. *Sailesh has made some statements after what he has learnt today. Please help him by telling if the statements are correct or not. Put a 'yes' in front of the correct ones and 'no' in front of the wrong statements:*

a. Turret can be symbolized to a wheel.

b. After indexing you need to realign the turret matching the center axis of the lathe machine.

c. The Turret automatically recognizes the difference between the tool tip height with respect to its center.

d. An operator has nothing to do but just check if everything is working fine when working on lathe machine.

e. It is the duty of an operator to make sure that the cutting tip of each tool comes to the point of operation.

Notes

Answers:

1.

a.	Turret, and wheel	b.	Revolving	c.	Circumference
d.	Recognize, and tool tip	e.	Cutting tip , center axis		

2.

a.	Yes	b.	No	c.	No
d.	No	e.	Yes		

Tool Number and Offset Number

At the end of this session you will be able to:

- identify various turning tools with respect to operation;
- identify the mounting positions and the numbering system.

Session Plan	
1	Tool Offset Settings
2	Tool Number and Offset Numbers
3	Key Learnings
4	Worksheet

Tool Offset Settings

Sailesh has been observing the CNC Lathe in operation, he notices the turret moving in front and going back to its resting position after each operation. He wonders how the machine takes its position, and how it determins the exact position of the tool. He sees the operator punching some numbers, but it makes little sense to him. He decides to ask Mohan.

Sailesh: Mohan, while Suresh was working on the CNC machine, I noticed how the turret goes back after every operation. Also, I'm quite puzzled as to what he keeps punching in the controller.

Mohan: Sailesh, That's a very intelligent question. The turret goes back to its resting position after each operation. Suresh must have been doing the tool offset setting.

Sailesh: Tool offset Setting? What is that?

Mohan: It's quite simple! You see, in order to actually achieve the depth fed as per the program we must set each tool touch position with respect to the work piece. This can be achieved by setting individual tool offset position for each tool in the program of the controller.

Remember every time you change the work piece position or the tool position, the tool offset will have to be redone.

Simply said, Tool Offset Settings are the settings done on the CNC machine for each tool mounted on the tool turret. This is also called the reference point or the zero setting for each tool with respect to the work piece.

Each tool tip will touch the work piece at the point where it has to perform the operation. Feed this reading in the controller. This point acts as the reference point or zero setting for the controller. The controller understands this as start point for that particular operation. This operation needs a repetition every time you remove or refit the tool or the work piece.

So Sailesh, why don't you recapitulate the common tools used in lathe operations?

Sailesh: Sure Mohan, now I'm quite familiar with the lathe operations.

(Sailesh starts listing the operations one by one.)

- OD and facing turning tool
- Taper turning tool
- Empty tool (There is no tool fitted in this machine)
- Drill
- Internal boring tool
- Central drill
- Internal groove cutting tool
- Empty tool (There is no tool fitted in this machine).

Mohan: That's impressive Sailesh. I'm proud of you.

Tool Number and Offset Numbers

Sailesh: Mohan, how will the machine know which tool is fixed in which slot?

Mohan: A very intelligent question Sailesh. I'm glad you asked. This is where tool number comes into play.

Mohan: It is important to know about the tool numbers. The Tool number is the slot number in which a particular tool is inserted in the turret. Remember to mention it in the program so that the controller knows as to which tool is available in which slot. The program is automatically amended so that the operation to be performed is performed by the right tool.

Sailesh: So, we need to feed the tool number in the controller and the program will move the tools accordingly.

Mohan: Exactly Sailesh, for instance the OD turning carbide bit is in turret slot number 1. The program will move the OD carbide bit fixed in slot number 1 every time the OD turning operation is recalled.

Once the tool is fixed in a particular slot number, it can even be recalled to perform two or more operations depending upon the type of operation. For example, facing, OD turning and taper turning all can be done by using the same tool. So every time either of these operations needs to be performed, this particular tool will be put to use.

Sailesh: Well friends, this was a critical session. A little more practice on setting the tool offset will make things much clearer. So while I go set the tool offset for my next lathe operation why don't you get some practice of your own.

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. What is tool offset settings?

2. What are tool numbers?

3. State the procedure to set the offset position for a tool.

Worksheet

1. *Sailesh is asking his friends a few question about tool offset numbers, help him by filling the correct answer in the blanks:*

   a. Tool Offset Settings are particular settings that are fed in the _____.

   b. Tool Offset Settings are fed in the controller by making a _____.

   c. Tool Offset setting helps the controller to understand the _____ for a particular operation.

   d. The tool offset numbers are numbers that indicate the _____ on the _____.

   e. The tool offset numbers are_____ indicating its exact location.

2. *Comment on the statement given by Mohan, if they are true or false:*

   a. Tool offset setting and Tool offset number have no difference.

   b. For a particular operation, you need to feed the program accurately by feeding the correct tool offset number to get accurate operation.

   c. Every tool in the turret has a tool number which helps an operator to organize tool data.

   d. For facing operation; a skin touch is given on the face to take the touching of the tool on the work piece.

   e. For OD turning a skin touch is given on the OD to take the touching of the tool on the work piece.

Notes

Answers:

1.

a.	Controller	b.	Reference point	c.	Starting point
d.	Location and Turret	e.	Numbers		

2.

a.	False	b.	True	c.	True
d.	True	e.	True		

Work Holding

At the end of this session, you will be able to:

- choose the appropriate method for clamping (work holding).

Session Plan	
1	Work Holding
2	Job holding Methods/ Devices
3	Key Learnings
4	Worksheet

Work Holding

Sailesh is reading a book about points to remember while working on a lathe machine. He wants to know more about work holding.

He starts reading about work holding. While he is reading, Sailesh's friend Kartik comes into the room and requests him to read aloud so that he can also listen.

Kartik is unable to understand some of the words, so Sailesh calls Mohan for help.

Sailesh: We were reading about the lathe machine. Sadly, we have got stuck on the topic named 'work holdings' Will you please help us?

Mohan: Surely, I will help you. Let me tell you about work holding in simple language.

I am sure that both of you understand that a lathe machine has vast number of applications. In order to facilitate the easy machining of work piece, it is important to hold the piece tightly and securely. For this purpose various types of accessories are used to facilitate easy holding of the work piece. This process or action is called work holding.

Sailesh: Mohan, is work piece an object on which turning operation is to be performed?

Mohan: Exactly! Is everything clear up to this point?

Sailesh: Yes, thank you Mohan.

Job holding Methods/ Devices

Sailesh: Job Holding devices or method means, devices or methods that help you in job holding. Isn't it?

Mohan: Correct Sailesh! Let me tell you in detail. As we know the turning operation sequence has to be planned based on the component drawing. It all depends after seeing the component drawing i.e if the operation will be performed in one setting or multiple settings. Moreover, the component drawing is the basis for deciding what type of job holding device will be used on the machine.

Sailesh: So are there various types of job holding devices or methods?

Mohan: Yes Sailesh, there are four types of job holding devices or methods. They are:

1. 3 jaw self centering chuck.
2. 4 jaw independent jaw movement chuck.
3. Face plate with center for turning between center to center.
4. Collets.

Sailesh: Oh! That means based on the type of job holding required for particular operation the job holding devices or method is fitted on the head stock of the machine.

Mohan: Yes that's right!

Always keep in mind the important criterion of work holding is that while in operation when the machine is making the job rotate and the cutting operation is taking place, the holding should be strong enough to withstand the load from the cutting tool to the job/ work piece.

Sailesh: Does the operator need to know all this?

Mohan: Yes Sailesh. The operator has to understand and plan the amount of material to be held in the chuck or collet or whatever the holding device is used.

Sailesh: That makes me feel important and special...

Mohan: Of course!

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. What is a Work Holding?

2. Name the different types of job holding devices or methods?

Worksheet

1. *Help Sailesh by telling if the statements given below are right or wrong:*
   a. Jaw self-centering chuck is among the 4 devices or methods used for job holding.
   b. Turning operation sequence has to be planned based on the component drawing.

2. *Can you tell Sailesh the number of types of devices or methods for job holding?*

_____

3. *Fill in the blanks with correct answers to complete the statement.*
   a. The component drawing is helpful to see if the operation will be performed in _____ setting or _____ settings.

4. *Fill in the space in the sentence with appropriate answer to give it a meaning:*
   a. _____ is the basis of deciding what type of job holding device will be used on the lathe machine.

Notes

Answers:

1.

a. Right

b. Right

2. *4 (four)*

3. *One and multiple.*

4. *Component Drawing.*

Work Piece Setting

At the end of this session, you will be able to:

- clamp the tools and hold the jobs;
- check the position of the job.

Session Plan	
1	Work Piece Setting
2	Importance of Work Piece Setting
3	Process of Work Piece Setting
4	Key Learnings
5	Worksheet

Work Piece Setting

Sailesh is revising the topic Work Holding from the book on lathe. He comes across a topic named Work Piece Setting and asks Mohan to explain.

Sailesh: I remember that you mentioned about Work Piece in Work Holding. Is Work Piece always the same in all lathe operations?

Mohan: Yes, Work Piece is always the same. OK, Can you tell me what do you understand by a Work Piece?

Sailesh: Hmm... According to me a Work Piece is an object on which turning operations are to be performed.

Mohan: Very good. Do you remember what I told you about component drawing?

Sailesh: Yes. All the turning operations sequences have to be planned on the basis of the component drawing. It is the component drawing that helps to decide if the operation will be performed in one setting or multiple settings.

Mohan: Great! Sailesh, I am pleased to know that you remember everything that I told you.

Sailesh: Thank you.

Importance of Work Piece Setting

Sailesh: I have a question, why do we take care of work piece setting so much? Is it that important?

Mohan: Yes, Sailesh. For the various operations that can be performed in turning, the Work Piece Setting plays a very important role:

1. Dimensional accuracy of the final component.
2. Finish of the final component.

3. Optimization of machining operation.

4. Wear and tear of tool and machine.

Always keep in mind that every job related to lathe machine is important and one should be very careful with it.

Sailesh: Yes you are right, I will take care.

Process of Work Piece Setting

Sailesh: Mohan can you tell me the process of Work Piece Setting so that I am able to get best results.

Mohan: Sure, Sailesh. I was about to tell you. In order to get the best results, optimum care has to be taken in the following area of the process of Work Piece Setting:

1. Holding of the job on the chuck, collet or between centres. The amount of material held inside the chuck collet should be enough to bear the tool cutting load coupled with amount of stock removal per cut at the selected RPM.

2. Clearances from other parts of the machine. The work piece setting has to be planned in such a way that once the job has started, the work pressure on the same after the process of machining should not either make the work piece touch or collide with any other part of machine or become weak enough to fall out of the job holding.

3. Allowance for measurement of the dimensions while the Work Piece is loaded on the machine.

The Work Piece has to be set in such a manner that while the job is held on the machine, measurement of the important dimensions can be conveniently taken without removal of the part from the machine as far as possible.

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. What is Work Piece Setting?

2. Give the importance of Work Piece Settings?

3. Describe the process of Work Piece Setting and the care needed in various areas to get best results?

Worksheet

1. *Help Sailesh by telling him if the statement is correct or incorrect:*
   a. Does the Work Piece mean the same in both Work Holding and Work Piece Setting? _____
   b. The component drawing has no relation with Work Piece Setting. _____
   c. Work Piece Setting is important if you want to get dimensional accuracy of the final component. _____
   d. Work Piece Setting has nothing to do with wear and tear of tools and machine. _____
2. *Help Sailesh to fill in the blanks with the correct answers:*
   a. Optimum care is required when you are holding the job on the _____, _____ or between centres.
   b. Optimum care is needed towards _____ from other parts of the machine.
   c. Work Piece Setting plays an important role in _____ of machining operation.
   d. The _____ drawing helps to decide if the operation will be performed in _____ settings or _____ settings.

Notes

Answers:

1.

a. Correct b. Incorrect c. Correct

a. Incorrect

2.

a. Chuck, collet b. Clearances. c. Optimization

d. Component, One and Multiple.

Fault Tracing Techniques

At the end of this module you will be able to:

- identify the error code;
- perform the operation to rectify the error.

Session Plan	
1	Fault Tracing Technique
2	Drive Failure
3	Vibration Fault
4	Key Learnings
5	Worksheet

External Threading

Sailesh has been told by his friend working in some other workshop that their lathe work got stopped due to some fault. Sailesh is worried and asks Mohan how to trouble shoot various common faults.

Sailesh: What is are the kinds of faults that we may encounter while working on a lathe machine?

Mohan: Before telling you about the faults and the technique to trace or trouble shoot faults, I would like to tell you something about lathe machine. CNC lathes are lathe machines with a computer interface to make all the operations guided by a program for following pre-planned sequence of operations.

Sailesh: Yes, I remember that CNC lathes are complicated machines. CNC, or computer numerical control, lathes use a computer and a program to cut raw materials to the desired shapes.

Mohan: That's right. Now, let me tell you about the fault tracing or trouble shooting of lathe machine. If there is any issue with the operation with respect to the program, the program refuses to let the operation be performed and sounds an alarm to inform that something is not OK

Sailesh: Now, I understood the meaning of trouble shooting. Can you tell me when we need to do trouble shooting?

Mohan: Yes, there is a situation when a lathe stops suddenly or does not start when the start button is pressed; there is some kind of conflict. You must read the information displayed to fix the problem and clear the alarm before proceeding.

Always reset the home position before restarting. If you do not reset the home position then you will have problems with the accuracy of the machine. When the machine is turned on, it has to have a reference point, which is also known as the machine's home position.

Sailesh: Thank you Mohan.

Drive Failure

Sailesh asks Mohan to tell him about a specific fault and how to trouble shoot it.

Sailesh: I have understood everything that you told me about the fault tracing technique but it would be good if you could tell me about some real life situation of a fault and how to trouble shoot It.

Mohan: Sure, Sailesh! I will give you an example of Drive Failure.

Sailesh: What is drive failure? What will happen in the failure phenomenon in drive failure?

Mohan: The failure phenomenon in drive failure is that the machine starts automatically run, CRT display 401 alarm. In this the machine goes beyond the control of the controller or operator and will not take any command.

Sailesh: Oh God! It sounds scary!! How will we analysis it and what will be the treatment process in this case?

Mohan: Well, Sailesh there is nothing to get scared of. You will see FANUCOM appear alarm no 401 is the meaning of "axis servo drive VRDY signal is disconnected, the drive is not ready".

Now let me tell you the failure analysis and inspection:

a. According to the actual configuration of the meaning of the fault and the machine tool servo feed system maintenance in the following order were checked and confirmed.

b. Check L / M / N axis servo drive, drive status indicator PRDY, VRDY are off.

c. Check the servo drive power ACl00V, ACl8V were normal.

d. Measure the driver control panel auxiliary control voltage is ± 24V, ± 15V exception.

Based on the above examination, initially identify a fault with the drive control power.

a. Carefully check the input power, found on the X-axis servo drive. Input power fuse resistance is greater than 2MΩ, far beyond the specified value.

Sailesh: How will I do troubleshooting in that case?

Mohan: In this case, after replacing the fuse again to measure DC auxiliary voltage ± 24V, ± 15V back to normal, the status indicator PRDY, VRDY recovery normal, re-run the machine, 401 alarm disappears.

Sailesh: It sounds easy in comparison to the problem.

Vibration Fault

Mohan: To give you clearer information about the fault tracing technique, I will give you another situation called Vibration Fault.

Sailesh: Vibration fault! What is that?

Mohan: The fault phenomenon of vibration fault is during the Z axis movement, vigorous vibrations appeared, the CNC system does not give any alarm, but the machine does not work.

Sailesh: What is the failure analysis and inspection of Vibration fault?

Mohan: After careful observation and inspection, it is found that movement in a small area (less than 2.5mm), in the Z-axis, the machine is working smoothly without vibration, but more than the above range, machine gives violent vibration.

According to the analysis of this phenomenon, the position control part of the system and servo drive itself on the preliminary determination did not show any fault and may be it has some failure in the position detection device, the pulse encoder.

Taking into account the machine is a semi-closed loop structure, maintenance was tried by replacing the motor, and it was determined that the cause of the failure was due to the defective pulse encoder.

Sailesh: How will I understand the root cause of the malfunction?

Mohan: To get a better understanding of the root cause of this malfunction, following analysis and testing was carried on:

1. The servo drive main circuit was powered manually and the motor shaft was run, checked the system and found that regardless of the motor forward, reverse, system monitor is able to correctly display the actual position value, indicating that the position encoder signal output were being correctly interfaced.
2. Since the machine Z-axis ball screw pitch is of 5mm, as long as the Z-axis moves around 2mm i.e. vibration occurs, therefore, the cause of the fault may be associated with the actual position of the motor rotor, i.e. pulse encoder rotor position detection signal is not functioning .
3. According to the above analysis, taking into account the Z-axis can be normal movement of about 2.5mm, is equivalent to the actual motor rotation of 180 °, therefore, to further determine the fault location on the rotor position detection signal was reversed.
4. In accordance with the above example, following the same method, remove the pulse encoder; replace the same with a new one and the problem will be solved.

Further examination of the encoder revealed the encoder internal output driver IC has been damaged.

Sailesh: After analyzing and testing its time to do actual trouble shooting. I don't know how to do the trouble shooting of vibration fault? Will you tell me?

Mohan: After replacing the integrated circuit, re-install the encoder, and press on the cases of the same method of adjusting the angle of the rotor, the machine started working normally.

Sailesh: This seems to be a manageable thing.

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. What is fault tracing technique?

2. Explain drive fault in detail and the procedure to troubleshoot it.

3. Explain the vibration fault in detail and the procedure to troubleshoot it.

Worksheet

1. *Mohan is testing the knowledge of Sailesh and his friends by asking them what is the full form of CNC lathe machine.*

2. *Sailesh is asking his friends what is the important thing to do before restarting the lathe machine?*

3. *Tell Sailesh what will happen in case the operator has not properly set the home position while restarting the lathe?*

4. *Sailesh has given statements to test the knowledge about fault tracing technique. Complete the statements by filling up the blank spaces:*

   a. When a lathe stops suddenly or does not start when the start button is pressed; there is some kind of _____.

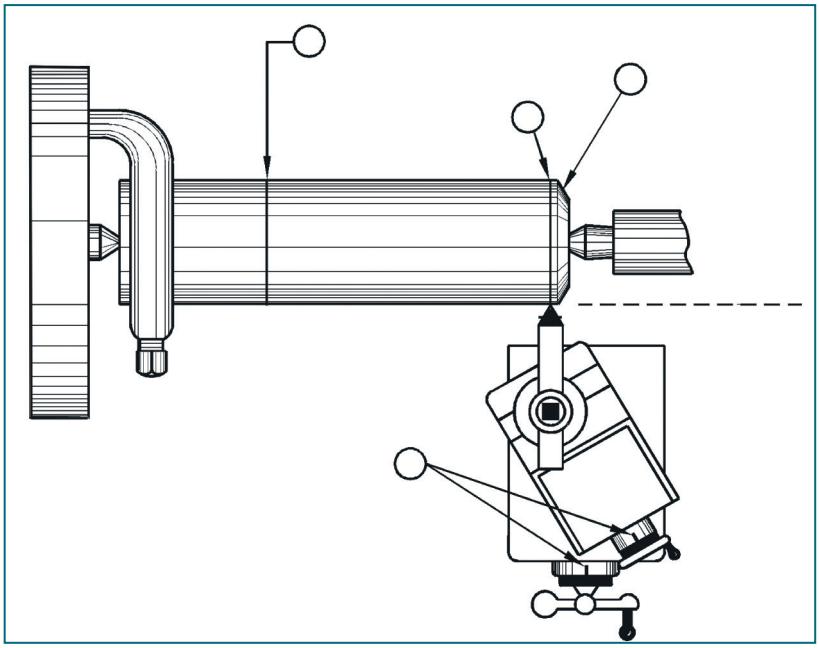
   b. In drive fault, the machine goes _____ the control of the controller or operator and _____ any command.

   c. While doing failure analysis and inspection in the drive fault, it is important to check the servo drive power _____ were normal.

a. The fault phenomenon of vibration fault is during the _____, vigorous vibration appeared; the CNC system does not give any alarm, but the machine _____ work.

a. After replacing the _____ circuit, re-install the _____, and press on the cases of the same method of adjusting the _____ of the rotor, the machine started working normally.

5. *Sailesh has made the following image for preparation of thread and label various parts marked:*



Notes

Answers:

1. *Computer Numerical Control lathe machine.*
2. *To reset the home position before restating.*
3. *You have not properly set the home position when you have restarted the lathe, then you will have problems with the accuracy of the lathe machine.*
4.
   a. Conflict.
   b. Beyond, will not take.
   c. ACI00V, AC18V.
   d. Z axis movement, does not.
   e. Integrated circuit, encoder, angle.

Routine Preventive Measures

At the end of this module you will be able to:

- state the do's and don'ts of daily and weekly maintenance.

Session Plan	
1	Preventive Maintenance
2	Routine Preventive Maintenance for Lathe
3	Key Learnings
4	Worksheet

Preventive Maintenance

Mohan is taking a session to emphasis on the importance of routine preventive maintenance of lathe machines. He says:

Mohan: Preventive maintenance is of utmost importance. It is something that everyone should take care of.

Sailesh: Mohan, what do you mean by preventive measures? Can you tell me in detail?

Mohan: Sure Sailesh. Preventive maintenance is the planned maintenance of a plant`s infrastructure and equipment with the goal of improving equipment life by preventing excess depreciation and impairment.

Sailesh: What can be included in the preventive maintenance?

Mohan: Preventive maintenance includes adjustments, cleaning, lubrication, repairs, replacements, and the extension of equipment life.

Do you know why is preventive maintenance stressed on so much? Well, it's because through routine preventive maintenance we can avoid many unwanted events or accidents.

Routine Preventive Maintenance for Lathe

After the lunch break the session on routine preventive maintenance continues... Mohan introduces this part of the session by saying that routine preventive maintenance of lathe is highly important and should be well taken care of...

Sailesh: How is preventive maintenance linked to CNC machine or lathe machine??

Mohan: The CNC machines are based on the linear and rotary movements supported by very precise bearings. The accuracy level obtained because of these borings demand very regular and systematic preventive maintenance of the slides and spindles.

The operation of turning generates lots of burrs, material waste, magnetically charged particles apart from dust and unwanted materials from atmosphere, coolant and materials being used in the operations.

Sailesh: I was thinking, what will happen if there were no unwanted inputs?

Mohan: Generally, if these unwanted inputs were not present, then the machine slides, spindles and bearings would be very well protected and auto lubricated.

During operation all these unwanted materials can seep in the slides, and bearings. Therefore regular cleaning and external oiling becomes mandatory as part of daily practice of preventive maintenance.

Sailesh: Daily practice! That means we need to clean and lubricate the machine daily?

Mohan: Yes, as a routine before calling off for the day, machine must be wiped off all the chips, burrs, water, coolant, oil and any other material not required. A thin layer of lubricating oil should be spread on the exposed slides, the joints of the spindles and bearing slides.

Always remember before staring the machine afresh; oil must be pumped in through the auto oil feed system to the extent that the minimal quantity of oil is seen bleeding from the slides and spindle ends.

Thorough vigil has to be kept for any sort of unwanted/ abnormal noise from any part of the machine.

Sailesh: What should we do in case any sort of noise from the machine is heard?

Mohan: In case, you hear any kind of noise from the machine then the maintenance engineer should be called immediately. Any electrical switch / membrane switch if not functioning or shows delayed functioning should be immediately reported to the maintenance engineer for immediate action.

Sailesh: That is highly informative, interesting and important! I will always remember and implement it. Friends, I would request you to do the same. It's beneficial to us in all ways!

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. What do you mean by preventive measures?

2. Describe what you understand by routine preventive measures in respect to lathe machine.

Worksheet

1. *Sailesh has asked his co workers to make a list of the things that will be included in the preventive measures. Help him by making list for him.*
2. *Mohan said that preventive measure were important. Why?*
3. *Fill in the blanks with correct answers:*
   a. The CNC are based on the _____ and _____ movements.
   b. The operation of turning generates lot of burrs, _____ charged particles apart from dust and unwanted materials.
   c. If the unwanted inputs are not there machine _____, _____ and _____ are very well protected and auto-lubricated.
   d. Machine should be wipe off all the _____, _____, _____, _____, _____and any other not required material.

Notes

Answers:

1. *Adjustments, cleaning, lubrication, repairs, replacements and the extension of equipment life.*
2. *To avoid or prevent unwanted events or accidents.*
3.
   a. Linear and Rotary.
   b. Magnetically.
   c. Slides, spindles and bearing.
   d. Chips, burrs, water, coolant and oil.

Stress Management

At the end of this module you will be able to:

- reduce, prevent, and cope with stress.

Session Plan	
1	What is Stress
2	Causes and Symptoms of Stress
3	Controlling Stress
4	Key Learnings
5	Worksheet

What is Stress

Walking into the workshop Sailesh immediately senses the confusion in the air. Unlike other days the workshop was in a chaotic state. Everybody had gathered around one of the benches. On delving deeper into the situation he learns that a worker, named Prakash had fainted. He was immediately rushed to the hospital and everybody returned to their stations. Sailesh was disturbed by the whole incident. Finding Mohan alone in the staff room he decides to have a heart to heart chat with him.

Sailesh: What was that all about? Will Prakash be fine? How did it happen??

Mohan: Don't get so worked up Sailesh. Prakash has been stressed for quite some time. He has just had a baby and was working overtime to meet his family needs. I could see it coming...The drop in his performance, family problems, overwork are all causes of stress.

Sailesh: Stress? What does that mean exactly?

Mohan: How do you feel when you know you are going to be tested or have to deal with a serious problem? Have you ever suddenly felt sick in your stomach and not been able to eat?

Sailesh: Yes often!

Mohan: Well, that is because of stress. Stress is our body's response to change. Our body responds by releasing hormones such as adrenaline and cortisol. These hormones cause our breathing and heart rate to speed up, and our blood pressure to rise. These reactions help us to react quickly and effectively to get through the situation at hand. This is also called the 'fight or flight' response.

Sailesh: Mohan, are you implying that stress is good for us?

Mohan: Surprisingly, yes! Optimum amount of stress helps to motivate us to get a task finished, or perform well. But, the problems come when stress is chronic and our body remains in high gear, off and on, for days or weeks at a time. Constant stress may then interfere with our ability to get on with our normal life.

So you see Sailesh, not all stress is bad. The key is to manage stress properly. Unhealthy responses to stress may lead to health problems.

Causes and Symptoms of Stress

Mohan: Remember Sailesh, what can be stressful to one person may not be to another. Stress can come from happy events (job promotion, buying a new house) as well as unhappy events (illness, overwork, family problems). As a CNC machinist, small mistakes at the job front can be stressful. In addition to this the daily grind of – broken tools, scrapping an otherwise good part, an angry boss and sometimes even unhappy customers can cause stress. It is important for you to understand what is causing stress for you—it is the first step for dealing with stress.

Stress affects each of us in different ways. Here are some physical signs and emotional signs that indicate stress:

- It can make you feel angry, afraid, excited or helpless.
- It can make it hard to sleep.
- It can give you aches in your head, neck, jaw and back.
- It can make you feel overwhelmed and out of control.
- It can make it difficult for you to concentrate.
- It can even weaken your immune system.
- It can lead to habits like smoking, drinking, overeating or drug abuse.

Sailesh: Yes, I often tend to have a throbbing head right before an exam when I was in school.

Mohan: Yes Sailesh… that may be because you're tensed. Sometimes you may not even feel it at all, even though your body suffers from it. We can help ourselves handle life's various demands by learning to identify stress and deal with it as soon as possible.

Controlling Stress

Sailesh began to think about times when he may have felt stressed. The list was endless...But, how was he to reduce stress and its impact on his life! He wonders how Mohan does it…He always seems so calm and composed. Sailesh decides to pick Mohan's brains on some stress management techniques.

Sailesh: How do I cope with stress?

Mohan: Once you know how stress makes you feel, it's important to have strategies for soothing or calming yourself down. Taking steps to manage stress will help you feel more in control of your life. Here are some good ways to cope.

- Check your self-talk – Saying things like: 'I can't cope', or 'I'm too busy', or 'I'm so tired', or 'It's not fair' can make you feel worse. Instead try saying soothing, calming things to yourself like 'I'll do the best I can', 'calm down' or 'breathe easy'.
- Make sure you are eating healthy food. Some foods with high fat, sugar, caffeine and salt can contribute to stress.
- Take time to do activities you find calming or uplifting, such as listening to music, walking or dancing.
- Avoid using alcohol, tobacco or other drugs to cope.
- Make time for yourself - take 15 to 20 minutes a day to sit quietly, relax, breathe deeply and think of a peaceful situation.

- Engage in physical activity regularly. Do what you enjoy — walk, swim, ride a bike or do yoga.
- Build a support system. Talking with friends or family about problems you have or decisions that you must make.
- Identify areas of agreement and disagreement and look for options to resolve the conflicts.
- Improve your ability to see options and resources that can be used to solve problems.
- Plan ahead - allow enough time to get the important things done. Remember, stress multiplies when you leave things for the last minute.
- Try to get 6 to 8 hours of sleep each night.
- Last but not the least - Laugh.

For workers everywhere, workplace stress has increased tremendously. Fortunately, Sailesh and you have learned to handle stress in healthy ways. Discover what is causing stress in your life and test to see which stress buster works wonders for you.

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. How can identifying the source of stress help you better cope with it?

2. Did you discover some ways to cope with stress that were not included in this session? What were they?

Worksheet

1. *Sailesh has had a stressful day at work.*

   a. What should he do to relieve his stress?

      i. He should shout at friends / colleagues. ☐

      ii. He should relax and think of a pleasing situation. ☐

      iii. He should continue thinking that today is his unlucky day. ☐

   b. Which of these symptoms show that Sailesh is stressed?

      i. He had a sound sleep last night. ☐

      ii. He has been feeling dizzy and exhausted. ☐

      iii. He has been laughing a lot lately ☐

   c. Which of these could be a possible cause for Sailesh's stress?

      i. He exercises daily. ☐

      ii. He has been eating healthy food. ☐

      iii. He has faced major changes at work. ☐

2. *Sailesh seems to have got some of his facts wrong. Help him by writing 'True' against the correct statements and 'False' against the ones you think are wrong.*

   a. The use of alcohol and nicotine products is the best stress buster.

   b. Humor and spiritual inspiration can control stress.

   c. Positive self-talk can help you deal with a stressful situation.

   d. What is stressful to one person may not be to another.

   e. Planning ahead and organizing tasks is one of the main causes of stress.

Notes

Answers:

1.

a. ii

b. ii

c. iii

2.

a. False

b. True

c. True

d. True

e. False

Work Ethics

At the end of this module you will be able to:

- demonstrate good work ethics.

Session Plan	
1	What are Work Ethics
2	Indicators of Bad Work Ethics
3	Indicators of Strong Work Ethics
4	Personal Grooming
5	Key Learnings
6	Worksheet

What are Work Ethics

Manav has just returned to work after taking un-informed leave of two days. It has also come to Mohan's notice that Manav has been taking unacceptably long lunch breaks and also leaves early on most days. Though he is very competent in his work he his frequently leaves early. His short work hours have put great pressure on his overworked colleagues. Mohan decides to talk to Manav's about his attitude.

Mohan: Manav, why didn't you inform me about your leave?

Manav: Mohan, given that I had pending leave, I didn't think it's important.

Mohan: Well it does Manav. We had to complete an order for an important customer and your absence put extra pressure on the others. Not only was the work flow disrupted but all of us had to put in extra time to get the job done on time. With a little planning this could have been avoided.

I have also noticed that you often come to work late. You have also seen loitering about after your allocated lunch time. This shows you lack good work ethics.

Mohan emphasizes how strong work ethic is the key to good performance. He goes on to reiterate the definition of work ethics: **Work ethic is a set of moral principles or standard work norms practices by an employee to promote the overall welfare of the company**. Manav was embarrassed and realized that his tardiness could obstruct his career growth. He apologized profusely and decided to mend his ways.

Indicators of Bad Work Ethics

Mohan: Friends, we saw that Manav was criticized for putting in short number of hours, but what about other indications of negative work ethics. Let's take a look!

Procrastination: Worker's with bad work ethics are often seen frittering away time. Such people wait

for the last minute to finish the work assigned to them and end up turning in low quality work. Friends, you must also take care of the amount of breaks you take.

Irregular Attendance: A worker with bad ethics often uses sick leave and pending leave as a vacation instead of their designated purpose. They often show up late for work.

Politicking and Gossiping: Someone with a negative work ethic is often party to gossiping and office politics. They often seek pleasure in stoking rumors that disrupt the workflow. They are jealous of perceived rivals and instigate others against them. They are selfish and care little for the overall growth of their organization.

Bad Mouthing the Company: A worker with bad ethics also has the habit of talking ill about the manangement and the company. This should be avoided at all costs.

Lack of Discipline: Employees with bad work ethics have little respect for the organization's policies. They do not follow the instructions given to them by their superiors. They also do not follow the rules laid down by the management. Such employees show bad work ethics when they do not take responsibility for their work.

Untidy Appearance: Another indicator of bad work ethics is an untidy appearance. People with bad work ethics always disregard the work dress-code.

Indicators of Strong Work Ethics

Mohan: Friends it's time for an honest evaluation -Do you think you have high standard of work ethics? Here is a list of fundamental workplace ethics that can help you reassess your ethical standards.

Integrity: Honesty is the corner stone of all relationships- be it your relationship with your supervisor or your co-workers. Individuals with high moral standards are trustworthy and reliable. Their consistent performance shows they are dependable and can be turned to in the hour of need.

Sense of Responsibility: A person demonstrating work ethics feel accountable for his job performance. They show up on time every day, put in their best foot forward and complete the job on time – even if it means putting in a few extra hours. They understand how to manage time and how to prioritize work. These employees respect their peers and help where they can, making the workflow smoother.

Initiative: People with good work ethics take initiative-see a job that needs to be done and do it! They often go out of their way to get work done.

Maintain Quality: Those with a good work ethics take extra measures to maintain the quality of their work. These employees show a high level of dedication to the company and are committed to improve the overall quality.

Discipline: An individual with good work ethics respects the company policies. An ethical employee arrives on time and gives advance notice of absence. Positive work ethics also means using leaves for their designated purpose. Such employees are punctual and reliable.

Respect: Those with a good work ethic deal appropriately with superiors, peers and subordinates and treats everyone with respect. They are courteous, do not use offensive language and refrain from gossiping.

Appearance: A person demonstrating good work ethics dresses up appropriately. Such people are always well groomed and hygienic. They wear clothes suitable to the job, task and environment.

Personal Grooming

Ramesh has just walked in 20 minutes late, looking untidy. He had a missing button on his shirt and his hair was messy. Ramesh was good at his job but his untidy appearance was a put off. Mohan decides to talk to Ramesh in private and give him some tips on grooming.

Mohan: Ramesh, I want to compliment you on your work.

Ramesh: Thank you Mohan!

Mohan: You are welcome but there is a slight problem…..

Ramesh: What is the problem, I thought you were happy with my work!

Mohan: The problem is the way you dress.

Ramesh: My dress and appearance? Well how does that matter if I'm good at my job?

Mohan: Did you know 55% of what the other person thinks of you is based on how you present yourself. Poor personal appearance and careless dressing can affect your career negatively.

If you smile and dress neatly – people think that you are pleasant, whereas, if you wear torn and wrinkled clothes people might conclude you're careless and irresponsible. You must look and act positively to project the image you want others to receive.

To appear professional, you must observe good habits of grooming and personal hygiene.

Ramesh: Grooming? What is that?

Mohan: Grooming is the processes of making yourself appear neat and tidy. Grooming involves overall cleanliness and personal hygiene. Here is a grooming check list for you:

- Comb your hair neatly.
- Wash your hair at least once a week.
- Keep your moustache neatly trimmed.
- Shave regularly.
- Brush your teeth twice a day and rinse after every meal.
- Avoid bad breath.
- Keep your nails short and ensure they are grime free.
- Iron your clothes and keep them neat and clean.
- If you are required to wear uniforms, you must take care of your uniform and report any damage to your supervisors.
- Do not wear the uniform outside the work premises.
- If you wear a turban, ensure that it is neatly tied.
- Avoid clothing that can create a safety hazard.
- Make sure your shoes are in a good condition and polished.
- Use mild deodorant to avoid bad body odor.

So there you have a few personal grooming pointers. You'll find that your chances of getting ahead in your professional life increase by following these simple tips.

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. What do you understand by workplace ethics? Why do you think it is important to exhibit work place ethics?

2. Your appearance influences the opinion of others around you. Do you agree? Justify your answer.

Worksheet

1. *Sailesh and his friends have recently started interning as lathe operators. Read the scenarios given below and decide who amongst his friends exhibits good work ethics.*

   a. Chaman has just overheard a piece of juicy gossip about Raghav. He then repeats it to other co-workers.

   b. Ramesh has called in sick because it's a beautiful day and he has decided to go shopping with his girlfriend.

   c. Kartik does not remove the burrs on one or two pieces, hoping the supervisor won't notice.

d. Mahesh has decided to extend his lunch break knowing that his supervisor is away from the workshop.

e. Raman has been sneaking some of the workshop supplies home.

2. *Sailesh knows that his supervisor is coming in late today. He decides to come to the workshop a little late knowing that his supervisor wouldn't be there early. Sailesh is not getting paid for his internship and believes he has nothing to lose.*

a. Is something wrong with this attitude?

_____

b. What do you think Sailesh should have done?

_____

3. *Mohan has been receiving complaints about one of his workers, Dharam. People around him say that he has body odour and it is difficult to work around him. Mohan gives him some tips to practice better hygiene. Dharam decides to heed Mohan's advice but cannot recall some key points. Help him to fill out the lapses in his memory.*

a. Use a _____ to avoid body odour.

b. Clothes should be clean and _____.

c. _____ your hair neatly.

d. Shave _____.

e. Brush your teeth _____a day and _____after every meal.

Notes

Answers:

1. *None of the above.*
2.
   a. Sailesh is exhibiting bad work ethics.
   b. Ideally, he should have informed his supervisor.
3.
   a. Use a _____ to avoid body odour.
   b. Clothes should be clean and ironed.
   c. Comb your hair neatly.
   d. Shave regularly.
   e. Brush your teeth twice a day and rinse after every meal.

Attitude

At the end of this module you will be able to:

- demonstrate positive attitude towards superiors, peers and other service providers.

Session Plan	
1	What is Attitude
2	Why Attitude Matters
3	Cultivate a Positive Attitude
4	Key Learnings
5	Worksheet

What is Attitude

Mohan is about to start his day at work when he hears an argument break out. Much to his chagrin, he learns that it is Dayal shouting at the Chaiwalla for bringing him cold tea.

Although a highly valued member of staff, Dayal had problems communicating with his peers. He tended to be overly autocratic to those subordinate to him. When an idea was presented, he was the first one to say, "This won't work." Fair enough. But he never offered any options. He never looked for alternative solutions. He never gave anyone the benefit of the doubt. The Staff did not stand up against him to avoid the stress of dealing with his confrontation.

Having witnessed the bickering first hand he decided something needed to be done. He had an important role in mind for Dayal but before that his ill-tempered nature had to be controlled. He decides to talk to Dayal.

Mohan: Dayal, I want to have a little chat about your tiff this morning.

Dyal: Oh that!! It was nothing. Suresh, that Chaiwalla brings me cold tea on purpose.

Mohan: If you walk around the workplace with a negative attitude, you are less likely to be approachable. An optimistic attitude on the other hand will enable you to gain the trust and acceptability of your peers and create a better workplace. This means you are bound to be more productive and achieve more.

Dyal: Oh no Sir, you don't know that little brat, he troubles me on purpose.

Mohan: Remember Dayal, you carry your attitude around with you, like a pair of glasses that tints your perception of the world. So, based on how you see the world, you will interpret and react differently to a situation than someone who has a different view. Your tinted glasses i.e. your attitude will affect how you think, how you behave and even how you feel.

Dayal always quick to react, was left with a deep seated thought.

Why Attitude Matters

Mohan: Hello friends I am sure the situation described above gave you some food for thought. Given the choice of dealing with a person like Dayal, who saw the negative in everything or a positive, upbeat person with a "can-do" attitude like Sailesh, whom will you choose?

Most people prefer to be around people with a positive attitude. So developing a positive attitude can help you get more both, professionally and personally. I have listed several benefits to having a positive work attitude.

Having a positive attitude helps you:

- Bounce back from adversities.
- Cope with stressful situations at work
- View failure as a learning experience and adopt new approaches.
- Build self confidence and belief in your own abilities.
- Enhance creativity. Since positive attitudes are at the heart of all innovation.
- Resolve conflicts in your personal and professional relationships.
- Better manage and express your emotions.
- Build empathy and readily see how other people think and feel.
- Keep motivated and keen to learn.
- Inspire and motivate others.
- Become a role model for others and garner more respect.
- Promote a “Can-Do” culture at work.
- Be perceived as a leader and get more special projects to work on.

Remember friends! A positive attitude in the workplace is like walking around with your promotion ticket in your pocket.

Cultivate a Positive Attitude

Mohan: While it is easier to sit and blame Dayal, how many of us actually think about our own attitude at work? Not many I’m sure! After all, we are human. If you pause and think, you will probably come up with dozens of examples where your negative attitude may have caused problems or conflict in your personal and professional relationships. Beware! Your negative attitude can sabotage you reputation. There are, however, a few simple strategies that will help you cultivate a positive attitude at work. Come let’s take a look!

Smile:

It’s truly amazing how powerful a smile can be. It naturally makes you feel more positive and improves other people’s outlook towards you. Even if you don’t feel like it, try to smile regularly throughout the day. Others will respond to you more favorably and you’ll naturally feel more positive.

Look for a solution and not the problem:

Negative people see obstacles; whereas positive people turn every challenge into an opportunity. Try and look for solutions instead of simply saying “That won’t work.” It’s fine to be skeptical, but bring

your own ideas to the table as well. Think of how can you turn the situation around? What are the alternative solutions? How can you make this work?

Respect the people around you:

Your negativity can suck the energy from those around you. Give your peers and subordinates the respect they deserve. If you do not agree with someone talk to that person in private instead of confronting them in public. Talk about the issue at hand and not about the person. Stay away from gossip. Gossiping about one colleague to the other can spread negativity.

Work towards continuous improvement:

Keep yourself updated about the latest development in your work field. Always look for ways to improve your work competencies. Taking courses and reading books can keep you informed and active. That way when the right opportunity comes knocking you will not be caught unaware.

Be constructive:

Look at a problem as an opportunity in disguise. Remember, challenges are a means to grow and achieve success.

Set realistic goals:

Set goals that are achievable. Take your strengths and weaknesses into account, and focus on accomplishing your tasks. Seeing your goals come to fruition is a natural and productive way to cultivate positive attitude.

Be pleasant and polite.

You should be polite to superiors, peers, and subordinates alike. Too often people are only polite to their superiors or those individuals they consider can help them progress in their careers. You will be surprised to learn how a simple cheerful “hello” can instill your day with positivity. Always thank, acknowledge a job well done and celebrate success, no matter how small it is.

So friends what are you waiting for? Practise the tips given in this chapter to develop a positive attitude.

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. How can you demonstrate positive attitude at work?

2. Have you seen someone at your organization whose positive attitude has paid off at work? Tell us his/her story.

3. What are some of the suggestions you will follow to cultivate a positive attitude?

Worksheet

1. *Put a tick mark against the correct option.*

   a. Kartik has a tremendous amount of job expertise but, is bitter and angry all the time. He is good Lathe Operator and believes that everyone else is incompetent. He could have easily become a team leader but has not been promoted. What do you think is the problem?

      i. Bad attitude.
      ii. Incompetent co-workers.
      iii. Jealousy amongst peers.

   b. Suresh has been seen gossiping around. He is polite to everyone while they are with him but that stops the moment they walk away. Do you reckon this person is:

      i. Promoting constructive feedback.
      ii. Simply releasing stress.
      iii. Spreading negativity.

   c. Whenever a technical problem arose, Raju was the man who had the answers. He was gracious and willing to help everyone at any time. Everybody liked working with him. What do you think is the reason?

      i. His ability to solve problems.
      ii. His optimism and positive attitude.
      iii. His good looks.

   d. Dayal has realized that his bad attitude is hindering his job performance. He has decided to change his attitude for the better. He starts by:

      i. Being respectful to his supervisor.
      ii. Being friendly to his co-workers.
      iii. Being polite to superiors, peers, and subordinates alike.

e. You are about to have a heart to heart conversation with Dayal requesting him to change his attitude. Put a tick mark against the advice you would give Dayal and a cross against the ones you will avoid.

i. Attitude is inherent and cannot be changed.

ii. One should have a solution-oriented attitude.

iii. Negative thinking is inevitable when an unpleasant situation disrupts your day.

iv. You should be polite only to your superiors as they are the ones who can add considerable valuable to your career.

v. Positive attitude can help you cope with stressful situations at work.

Notes

Answers:

1.

a. Bad attitude.

b. Spreading negativity.

c. His optimism and positive attitude.

d. Being polite to superiors, peers, and subordinates alike.

Communication

At the end of this module you will be able to:

- practice good oral and written communication skills.

Session Plan	
1	What is Communication
2	Verbal and Non- verbal Communication
3	Barriers to Communication
4	Tips to Communicate Effectively
5	Key Learnings
6	Worksheet

What is Communication

Sailesh has been interning for about three weeks and and has now begun to understand his job. He arrives early at the workshop and gets started on the work piece. He is performing the drilling operation when Suresh (another worker) comes and tells him something. Unable to hear him over the noise caused by the drilling machine, he stops the operations and asks again. Suresh, now at the other end of the workshop shouts out something about changed tolerances in the work piece. Sailesh still was unsure of what he had heard, but was afraid that he would look stupid asking a third time. He decides to carry on with the guesstimated tolerance level. A grave mistake! A whole batch of work pieces were put to waste.

Mohan decides to address this gap in the communication. He agrees that it was not entirely Sailesh's fault but all the same Sailesh must understand the importance of communication in the workplace. He begins with the basic definition of communication:

Communication is a process by which information travels from a source to the receiver. For it to be successful, the receiver must understand the message in the way that the sender intended. Sources and receivers can be individuals or groups.

Whenever we communicate, we follow the steps of the communication process shown below.

The source encodes the communication into a message, and transmits it through a channel. The receiver decodes the message and gives feedback one way or another.

By understanding the steps in the communication process, Sailesh has become more aware of his role in successful communication and recognizes what he need to do to communicate effectively.

Verbal and Non-verbal Communication

Mohan: Sailesh before we move on let me tell you about the types of communication.

Sailesh: Types of communication? That's simple! We use words to express our thoughts, ideas and messages across.

Mohan: Yes that's right. Any form of oral and written communication can be clubbed under verbal communication. Most oral communication involves people speaking to each other, either face to face or on the telephone.

Written communication also uses words but in a written form. For instance, engineering drawings, letters, emails, SMS etc. are all forms of written communication.

Sailesh: Yes, one can only communicate orally or by writing.

Mohan: Not really Sailesh, Communication can be non-verbal, which means that it does not use words to transfer information. Here are some examples:

- **Facial expressions**: We use facial expressions to show how we feel about something.
- **Gestures**: We can use gestures to show how we feel and also to communicate a specific message. For instance, I might hold up the palm of my hand to tell you to stop the operation.
- **Body language**: We use body language to demonstrate our attitudes and emotions. Often we are not even aware that we are doing it. For example, if you are in a conversation with someone and you begin to fidget, this silently communicates that you are not interested in the conversation.

So Sailesh, now you know Communication is more than just verbal. Here is some advice to improve non-verbal communication skills:

- Maintain eye contact.
- Nod your head to convey that you are listening or that you agree.
- Smile and show interest.
- Lean forward to show the speaker you are interested.
- Use a tone of voice that matches your message.
- Portray appropriate facial expressions.
- Avoid distracting actions or gestures.

Barriers to Communication

Mohan: To deliver and receive messages effectively, you must commit to breaking down the barriers to communication.

Sailesh: What is a communication barrier?

Mohan: Communication barriers are like walls that prevent messages from being properly sent or received. Often, due to the nature of the environment or our psychological perceptions we cannot communicate properly.

Let's take a look at this list of communication barriers.

- **Preoccupation**: Sometimes a person may have something else on his mind and may not be able to listen at all.
- **Hostility**: When two people are hostile, each often distorts messages from the other in such a way that hostility further increases.
- **Inarticulateness**: Simple inarticulateness may distort the message.
- **Physical environment**: The environment alone may create conditions under which communication cannot take place effectively. A noisy room may make it impossible to send and receive messages accurately.
- **Status**: The most difficult condition to overcome in communication is that of status. For Instance, while passing information to his superior one tends to screen out information detrimental to their welfare and tend to pass along information that helps their cause.
- **Lengthy messages**: If a message is too lengthy and disorganized the recipient may lose interest in the communication.
- **Poor body language**: Inappropriate body language can often confuse the recipient. For instance, you may be very interested in the conversation, but your yawning and glazed eyes may show that you are bored.
- **Jargon**: Technical jargon, acronyms, slang can disorient the message for those who do not understand them.

Tips to Communicate Effectively

Sailesh: Friends, today I got an insight into how ineffective communication can cause tremendous confusion and result in wasted effort and missed opportunity. With Mohan's help I have come up with some easy tips that will help you develop your communication skills.

- Use proper language—avoid jargon, acronyms, slang.
- Listen and do not interrupt the speaker.
- Avoid distractions. For example, taking personal calls at work, talking to co-workers while interacting with supervisor.
- Ask effective questions if the message is not clear.
- Restate the issue or question to verify understanding.
- Follow instructions promptly and appropriately.
- Request advice and feedback.
- Listen effectively.
- Use appropriate body language.
- Make eye contact.
- Nod and smile to acknowledge points.
- Allow the person to speak, without thinking about what you'll say next.

These tips will help you ensure that you say what you mean, and hear what is intended. So friends, go ahead and practice clear communication.

Key Learnings

Summarise your learnings here. Write your answers in the spaces provided.

1. What is non-verbal communication?

2. Reiterate some of the effective tips for communication.

Worksheet

1. *Mohan has asked Dharam to come up with some effective communication tips. Dharam has come up with a list. Which of these will you strike off as incorrect? Put a (✖) cross mark against these statements.*

   a. Avoiding eye contact and looking away from the speaker. ☐

   b. Lean forward to show the speaker you are interested. ☐

   c. Ask effective questions if the message is not clear. ☐

   d. Avoid noisy environments while communicating an important message. ☐

   e. Do not embarrass yourself by asking questions even if the message is not clear. ☐

2. *What do you think went wrong? Match the communication problem to the cause of the problem.*

a.	Sailesh is talking to Ramesh, but Ramesh is thinking about his mother who was admitted in the hospital yesterday.	i.	Hostility
b.	Sailesh is discussing the working of a new CNC Lathe machine with Sushil, but Sushil cannot hear a word over the clamor of the machines.	ii.	Status

c.	Afraid to portray himself in bad light, Sailesh has omitted to tell Mohan about some work pieces that do not fit the tolerance level.	iii.	Preoccupation
d.	Raghav and Dhruv have just had an argument so when Dhruv asks Raghav to take Hitesh's place at the Lathe Machine, Raghav thinks he is being spited and refuses to do so.	iv.	Physical environment

3. *Put a tick mark (✓) against the correct option.*

a. In the communication process, to encode means to:

i. Block the message

ii. Speak in a secretive and coded language

iii. Translate ideas into a message

b. A message can only be deemed effective when it is

i. Understood by others and produces the intended results.

ii. Communicated face-to-face

iii. Uses complex terms

c. Which of the following is an example of a nonverbal message?

i. Jargon

ii. Eye contact

iii. Yelling

d. Communication barrier does the following:

i. Enhances a message

ii. Causes listeners to listen to messages more carefully

iii. Distorts or interferes with a message

Notes

Answers:

1.

a. ✗ b. ✓ c. ✓

d. ✓ e. ✗

2.

a.	Preoccupation
b.	Physical environment
c.	Status
d.	Hostility

3.

a. iii b. i c. ii

d. iii

Skill India
कौशल भारत-कुशल भारत

सत्यमेव जयते
GOVERNMENT OF INDIA
MINISTRY OF SKILL DEVELOPMENT
& ENTREPRENEURSHIP

N·S·D·C
National
Skill Development
Corporation

Transforming the skill landscape