Basics of Mechanical Engineering-2

Prof. J. Ramkumar

Prof. Amandeep Singh Oberoi

Department of Mechanical Engineering

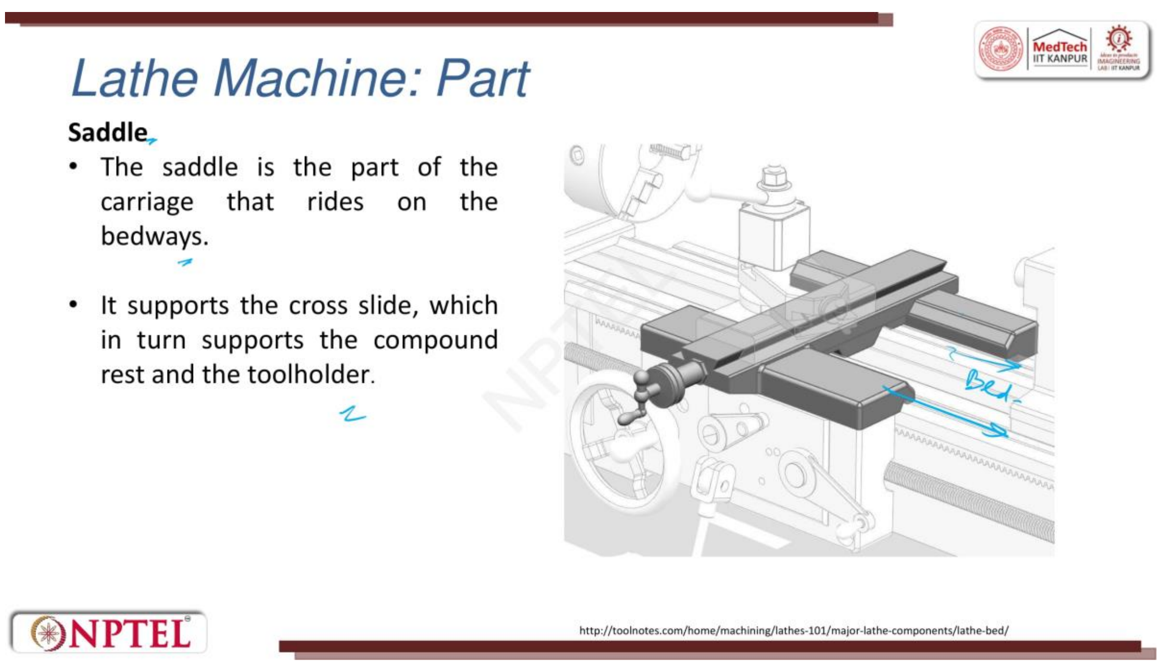
Indian Institute of Technology, Kanpur

Week 09

Lecture 37

Basics of Machining (Part 5 of 7)

Welcome to the continuing lecture on Basics of Machining.



Next, the saddle. The saddle is a part of the carriage that rides on the bed. So, this is the bed. This is the bed. This is the saddle. It supports the cross slide, which in turn supports the compound rest and the tool holder.

Lathe Machines: Type

- Turret Lathe Machine
- Speed Lathe Machine
- Engine Lathe Machine
- CNC Lathe Machine
- Automatic Lathe Machine
- Tool Room Lathe Machine
- Bench Lathe Machine
- Special Purpose Lathe Machine

So, when you look into the different types of lathe machines, there are turret lathe machines, speed lathe machines, and engine lathe machines, which are very commonly used. The latest advancement is CNC lathe machines. Then, you have automatic lathe machines, tool room lathe machines, and bench lathe machines.

Engine and bench are similar, but the bench is for smaller jobs. Then, there are special-purpose lathe machines. So, these are the different types of lathe machines that exist today.

Lathe Machines: Type

Turret Lathe Machine

- Turret lathes (capstan lathes) are designed for high-volume, duplicated parts production.
- They feature a hexagonal turret for sequential operations (facing, turning, reaming, boring), rotating after each task for mass production.
- With three tool posts, they save time, reduce errors, and are easy to operate, even for less-skilled workers.
- However, they require more floor space and are ideal for large jobs.
- Turret lathes are used in of bolts, nuts, fasteners, engine parts, gears, shafts, precision aircraft parts and medical device components.

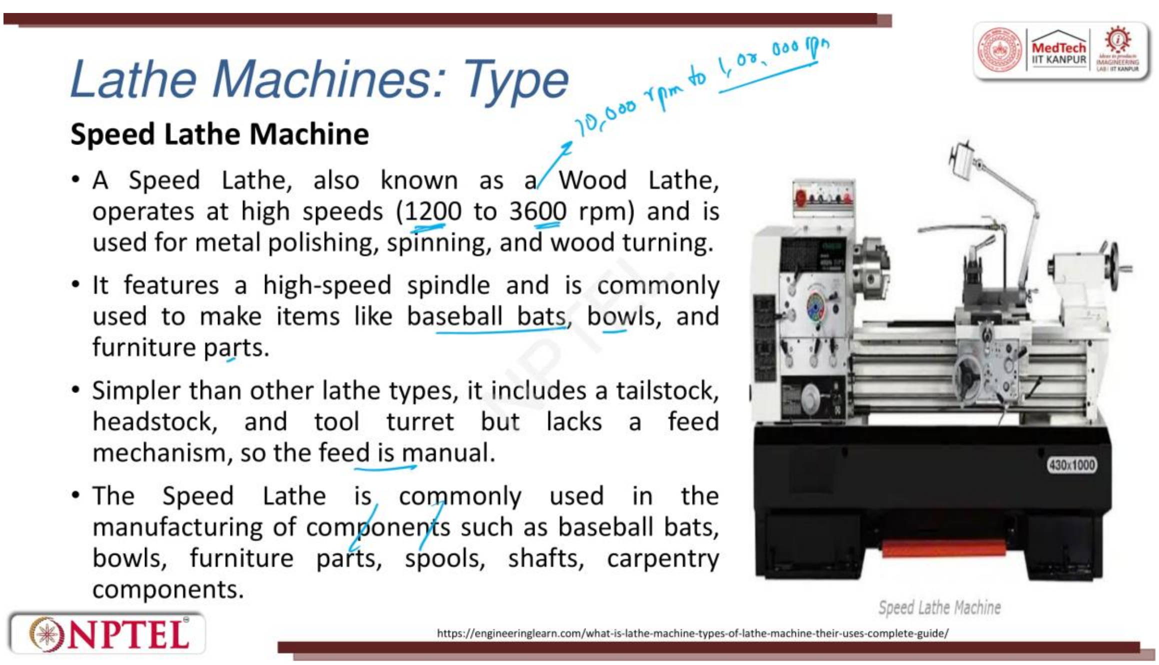
Turret Lathe Machine

https://engineeringlearn.com/what-is-lathe-machine-types-of-lathe-machine-their-uses-complete-guide/

So, the turret-type lathe machine. Here, the turret lathe machine or the capstan lathes are designed for high-volume and duplicate part production. So, here you have a hexagonal turret.

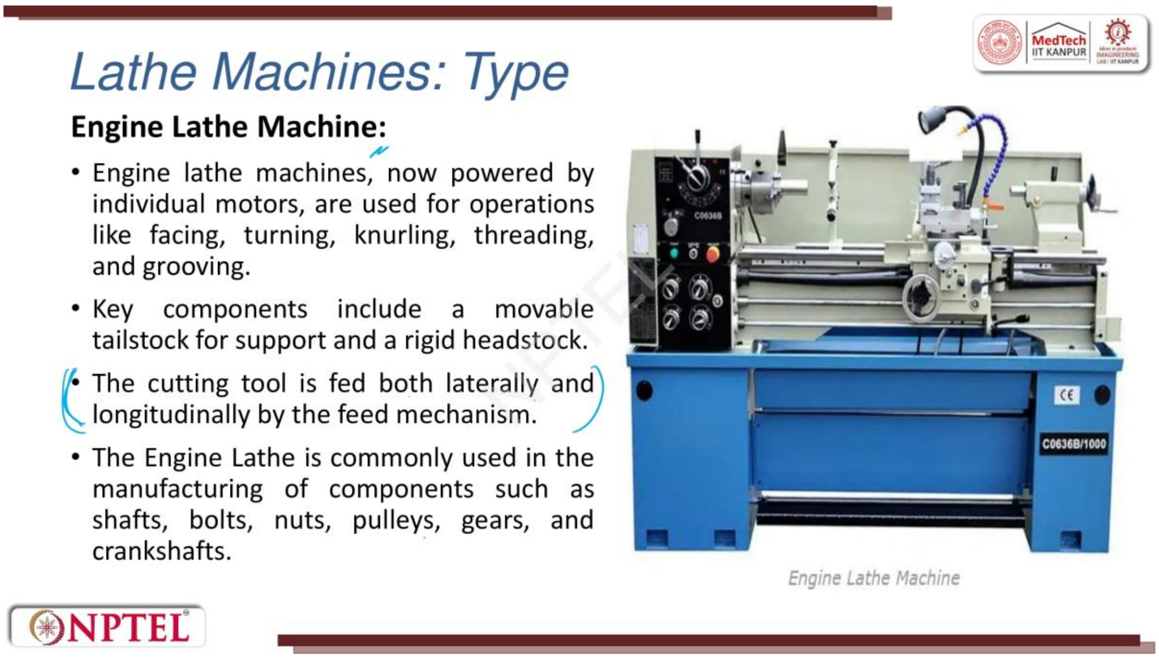
For sequential operation, the hexagon turret, it can be a first tool followed by second tool, third tool, fourth tool, fifth tool, sixth tool, and then when it comes back to the first tool, the next job comes into action. So, it can do facing, turning, reaming, boring, whatever it is, rotating after each task for mass production. With three tool posts, they save time, reduce error and are easy to operate even for less skilled worker. So now, in the tool post when there are three tools there, the positioning accuracy of the tool is increased. Because otherwise what happens, you remove the workpiece, load it into another machine, do an operation, load it to another machine, do an operation.

So, now all the operations are done in a single shot. So, positioning accuracy improves. However, they provide more floor space and ideal for large jobs. Turret lathes are used for making bolts, nuts, fasteners, engine parts, gears, shafts and other precision aircraft parts.



When we look at speed lathes, the speed what we talk about is around about 1200 to 3000 rpm, which is very low, but the name was given in early days, so that still the name is follows.

Today, we talk about somewhere between 10000 rpm to 100000 rpm. So, it features a high speed spindle and is commonly used for making items like baseball bats, bowls and furniture parts. It is simpler than the other lathe machine. It has a tail stock, head stock, a tool turret but lack a feed mechanism so that the feed is done manually. It is commonly used for operations.



So, engine lathe, this is commonly used. The key components are the same. The cutting tool is fed both laterally and longitudinally by a feed mechanism. This is very important for the engine lathe. And they are used for making nut, bolt, pulley, gear, crankshaft, etc.

Lathe Machines: Type

CNC Lathe Machine:

- CNC Lathe machines use Computer Numeric Control for high accuracy and speed.
- They have components like the spindle, tailstock, headstock, chuck, tool turret, and control panel.
- Used in aerospace and automotive industries, they offer high precision and faster operations, even for semi-skilled workers.
- CNC lathes are ideal for mass production and provide better dimensional accuracy than turret and capstan lathes, which lack programming systems.

NPTEL

https://engineeringlearn.com/what-is-lathe-machine-types-of-lathe-machine-their-uses-complete-guide/

This is the latest CNC machines. Here, the control what was given in the manual is now completely changed automatic. So, it still has a spindle, headstock, tailstock, tool turret and control panel. But here the program is written for controlling the movement of the tool with respect to the workpiece.

Lathe Machines: Type

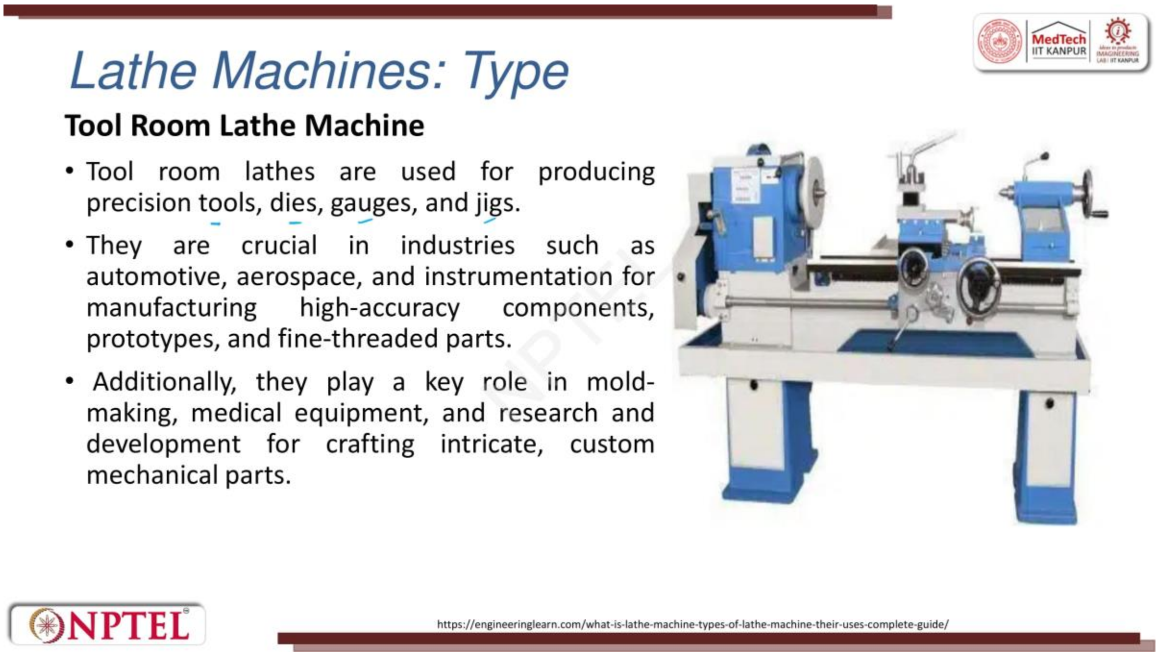
Automatic Lathe Machine

- Automatic lathes operate without manual intervention and are designed for high-speed, heavy-duty tasks.
- Their automated tool-changing mechanism enhances efficiency, allowing continuous operation.
- These machines optimize labor by enabling a single operator to oversee multiple units simultaneously.
- Essential in automotive, aerospace, and electronics, automatic lathes ensure precision, mass production, and cost-effective manufacturing of complex components.

NPTEL

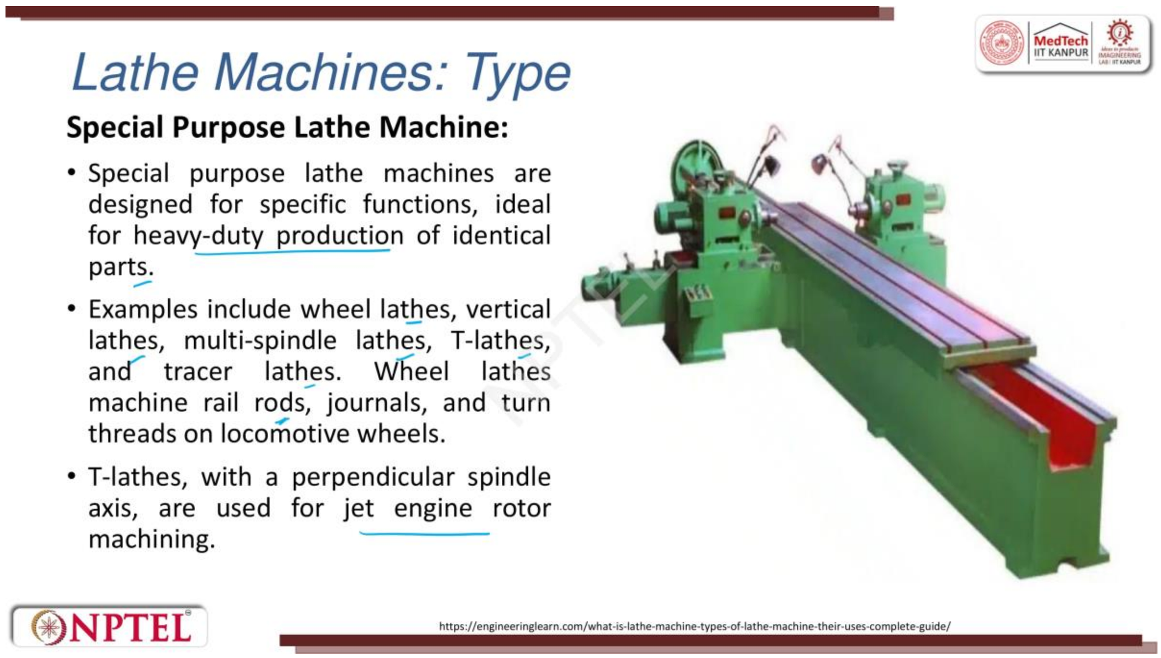
https://engineeringlearn.com/what-is-lathe-machine-types-of-lathe-machine-their-uses-complete-guide/

This is automatic lathe machines. Here, it operates without manual intervention for high speed heavy duty tasks. Here, the tool automatically changes a high efficiency and allows continuous operation. So, here you can try to produce multiple units in set period of time.



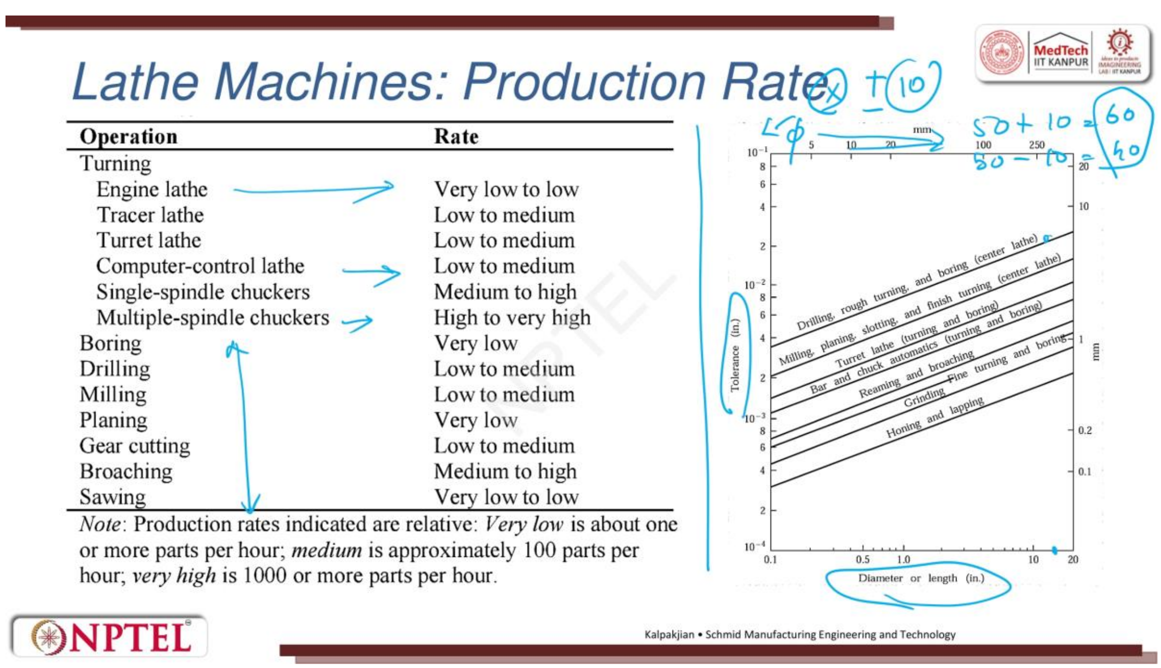
Tool room lathe machines, these are otherwise called as high precision lathe machines. They are used for making high precision tools, dies, gauges and jigs.

Very small and very precise. Bench lathe machines; these are large machines where the distance between the headstock and the tailstock is very large, and you can do big jobs. Here, it is very similar to speed lathes and engine lathes.



This is a special-purpose lathe. So, special-purpose lathes; here, you can modify the existing lathe machine itself to meet the requirements.

So, special-purpose lathe machines are designed for special functions, ideal for heavy-duty operations for specific parts. They are used for making wheel lathes, vertical lathes, multi-spindle lathes, T-lathes, tracer lathes, and wheel lathes, which are used for machining rail rods, journals, and turning threads, etc. The T-lathes with the perpendicular spindle axis are used for making jet engine rotary machine parts.



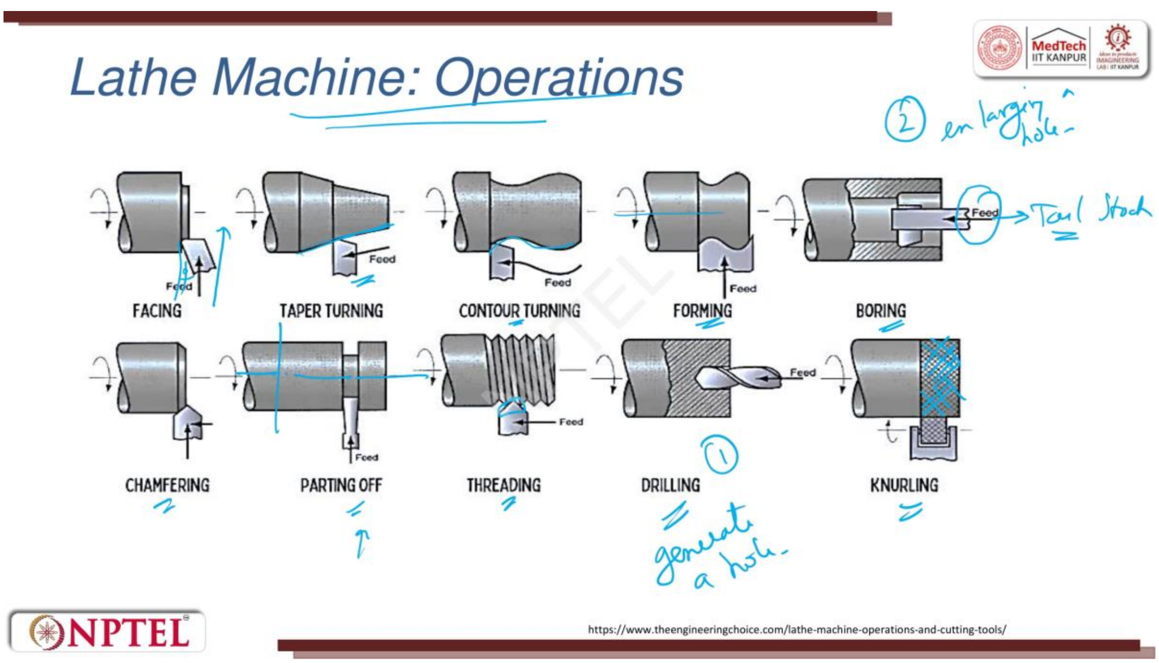
Operation	Rate
Turning	
Engine lathe	Very low to low
Tracer lathe	Low to medium
Turret lathe	Low to medium
Computer-control lathe	Low to medium
Single-spindle chuckers	Medium to high
Multiple-spindle chuckers	High to very high
Boring	Very low
Drilling	Low to medium
Milling	Low to medium
Planing	Very low
Gear cutting	Low to medium
Broaching	Medium to high
Sawing	Very low to low

Note: Production rates indicated are relative: *Very low* is about one or more parts per hour; *medium* is approximately 100 parts per hour; *very high* is 1000 or more parts per hour.

So, if you look at the production rate, These are the various lathes that are used. For example, computer-controlled lathe machines are used for low to medium volume. When we try to have multi-spindle chuckers, they are used for high to very high machining. When we try to use engine lathes, they are for very low to low-rate machining processes. So, these are all the other processes that can be done on a lathe machine: boring, drilling, milling, planing, gear cutting, broaching, and sawing. So, here, this figure clearly shows the tolerance we get for the diameter or the length of the job.

So, as the length of the job increases, the tolerance level increases. That means there is a possibility of defects occurring. What is tolerance? Tolerance is when you allow a deviation, like x plus 10. So, this x is the diameter of the shaft.

And this 10 is the deviation that is allowed. For example, if the diameter is 50 millimeters, you can have plus 10 or minus 10. So, the tolerance range is from 40 to 60. The given tolerance here is 20 millimeters. So, the part can have a maximum diameter of 60 or a minimum of 40. So, as the length increases, the diameter increases, and the tolerance also increases.



These are the various operations you can perform on a lathe machine. You can perform a facing operation. Facing is when you position the tool at an angle. This angle can be 45 degrees or any other angle, and the movement is perpendicular to the workpiece.

So, now what happens? The face is machined. Since it is rotating and there is axis symmetry, from the exterior end you only have to go to the half. The rest of the half is already machined when you move from here to here. You can put the angle, you can put the tool at an angle.

And then you can try to generate a feed, taper surface. So, taper surfaces are generated when this motion and the cross-slide motion—linear motion and the cross-slide motion—are given together, we try to generate a feed motion. You can also try to generate a profile in the turning. So, that is called contour turning. And if you have a tool which has the shape of the form to be generated on the workpiece, then that is called the forming tool.

If you want to bore a hole, that means in the shaft you want to make a hole. So, then we call it a boring operation. So, if you take an analogy, there are a lot of tunnels to be made for metro as well as for bridges. They try to bore. So, in a bore, what happens?

The tool is attached to the tailstock. And the feed is given to the tailstock. The spindle rotates. So, as it moves inside, the tool comes in contact with the workpiece, and then you proceed. So, for a boring operation to happen, there has to be a drilling operation first.

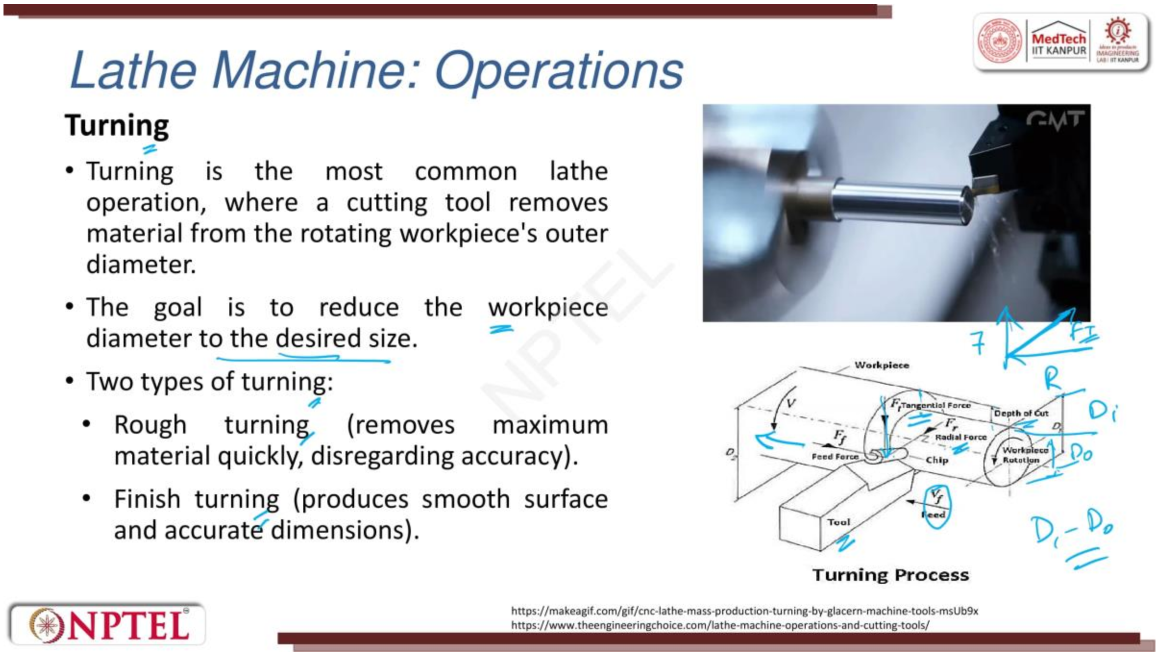
This is first, and this is second. Drilling is to generate a hole. Boring is enlarging a hole. And then, if the workpiece or the tool is at an angle, we try to make a chamfer. Taper is this; chamfer is very slight and is given on the top.

Then, we can also try to do parting off. Parting off means here you are trying to cut the shaft into parts. It is called parting off. So, here there will not be any feed along the workpiece, but it will be perpendicular to the axis. You get a feed.

So, that leads to the parting operation. Then, we saw that using the lead screw, you can try to generate threads. The profile of the thread will be generated on the tool. When it sinks, it tries to generate the tooth profile. I have already talked about drilling.

So, knurling is an operation in which we try to generate a crisscross hatch pattern. This crisscross hatch pattern is generated by a tool that has two wheels. They try to plunge on top of the workpiece and then create this profile. Knurling is not a subtractive process. Knurling is again something like a metal-forming process in which you try to deform the material to generate the diamonds on top of it.

This is used for better gripping. These are all the operations that you can try to do on a lathe machine. Here, we have tried to talk about turning.



So, turning is an operation that is used to reduce the workpiece diameter to produce the desired shape. So, there are two types of turning: one is called rough turning, and the other is called finish turning.

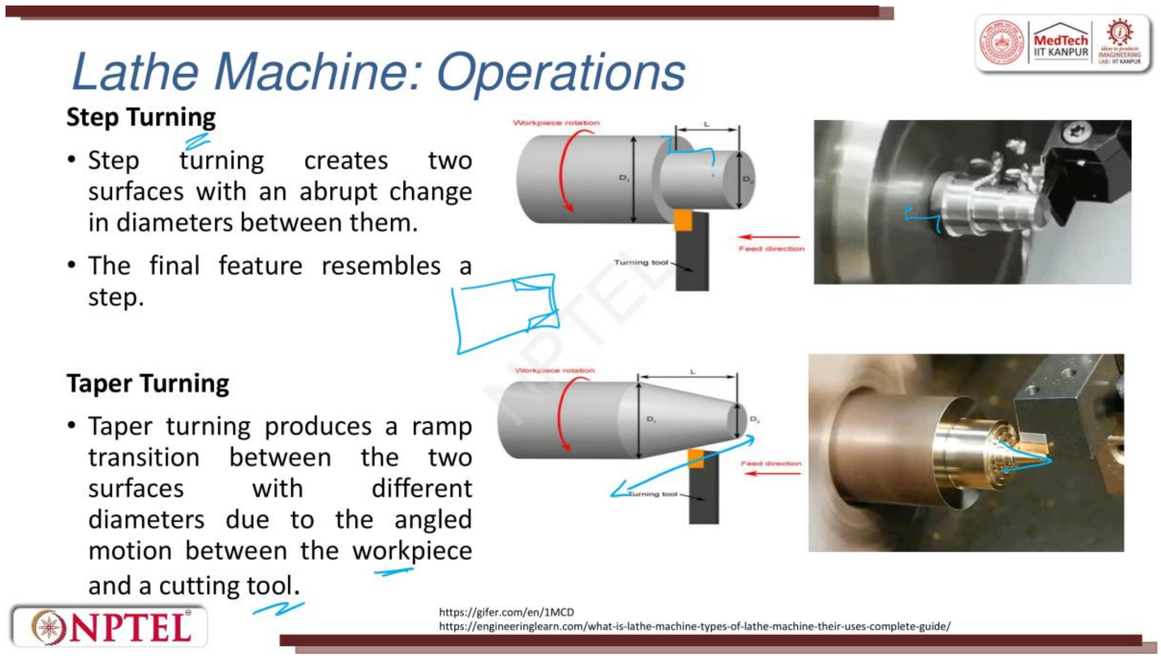
In rough turning, we try to extract the maximum material; the feed rates are higher, and the depth of cut is higher. When we try to do fine turning, the feed rates are smaller, and the depth given is also very small. The sense will be one-tenth; the rough turning you will try to do in a finish such that you get a smooth surface finish. So, this is the tool, and now we are trying to see what all the forces involved are. This is a tool.

The tool is moving in this direction. So, it is a turning operation. So, here, whatever we give or the resistance we get is called the feed force. Then, whatever we try—this is feed. Whatever is perpendicular to the workpiece we do is called the radial force.

The resistance given to the tool, which comes from the other direction, is called the radial force. So, this is the radial force, the opposite of whichever is the feed force. And a resolution of these two at an angle of 45 degrees is the tangential force, right? So, this is the radial force; this is the feed force; the tangent of it is Ft, which is the cutting force. That is the tangential force.

The tangential force also comes, which tries to move the workpiece down. The tangential force is otherwise called the cutting force. The workpiece rotates. This is the depth of cut. The change in diameter is called the depth of cut.

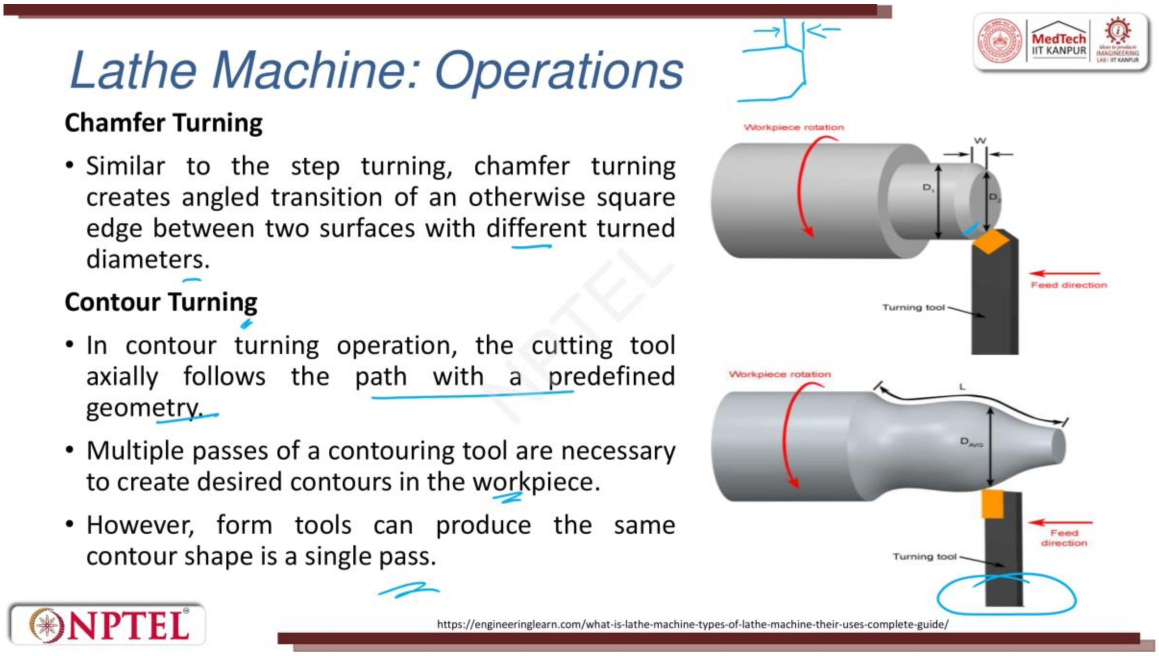
The initial diameter was Di. The final dimension, let us assume, is Do. So, Di minus Do will be the depth of cut. I have explained all the forces that are there. I have left one, which is the feed rate or the velocity of feed, Vf, which comes along the direction the feed force is given.



This is step turning. Step turning is when you want to have multiple diameters on the same shaft, creating a step-like appearance. So, this is a step. So, this is called step turning. Reducing the diameter to two different diameters on a single shaft is step turning.

What is taper turning? Taper turning is when you rotate the tool post to a particular angle, then you give a feed and also move the cross blade. Taper turning produces a ramp transition between two surfaces. So, it is an intermediate step. First is a full shaft, then a step shaft, and now it is a ramp that is being generated.

Taper turning produces a ramp transition between two surfaces with different diameters due to the angled motion between the workpiece and the cutting tool. So, this is taper turning. You can have forward taper turning or reverse taper turning.

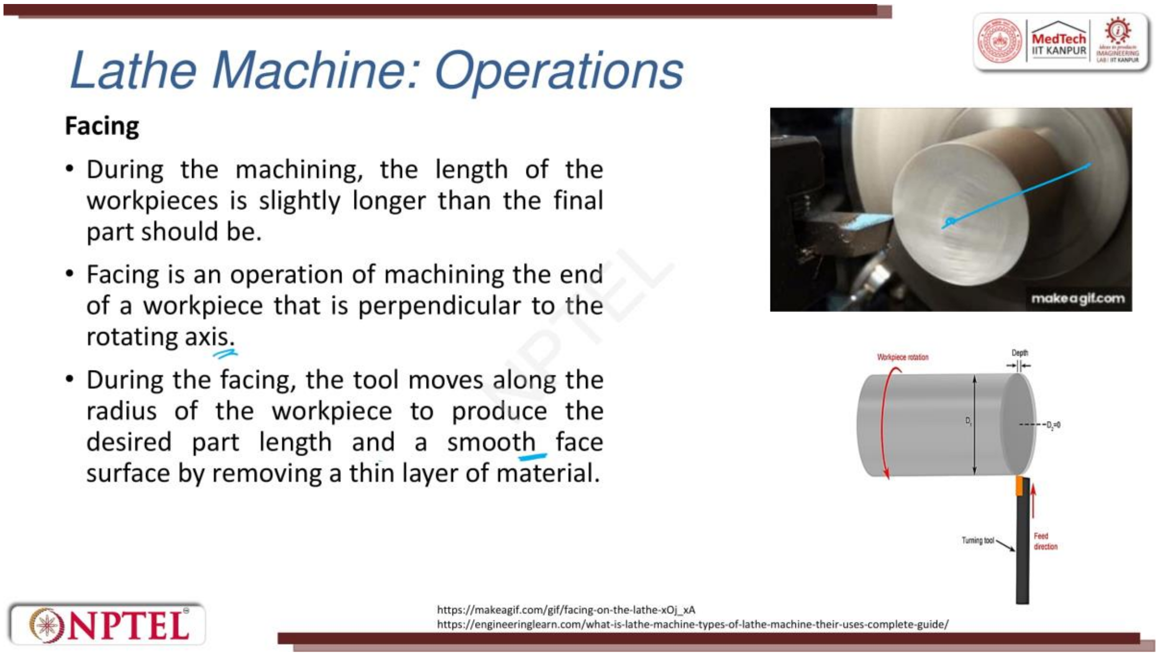


This is the chamfering operation. The chamfering operation is similar to step turning. Chamfering creates an angled edge. So, at the edge, we create a small bevel. This is called a chamfer. Similar to step turning, chamfering creates an angled transition on an otherwise square edge between two surfaces with different turned diameters. What are conical turning or contour turning?

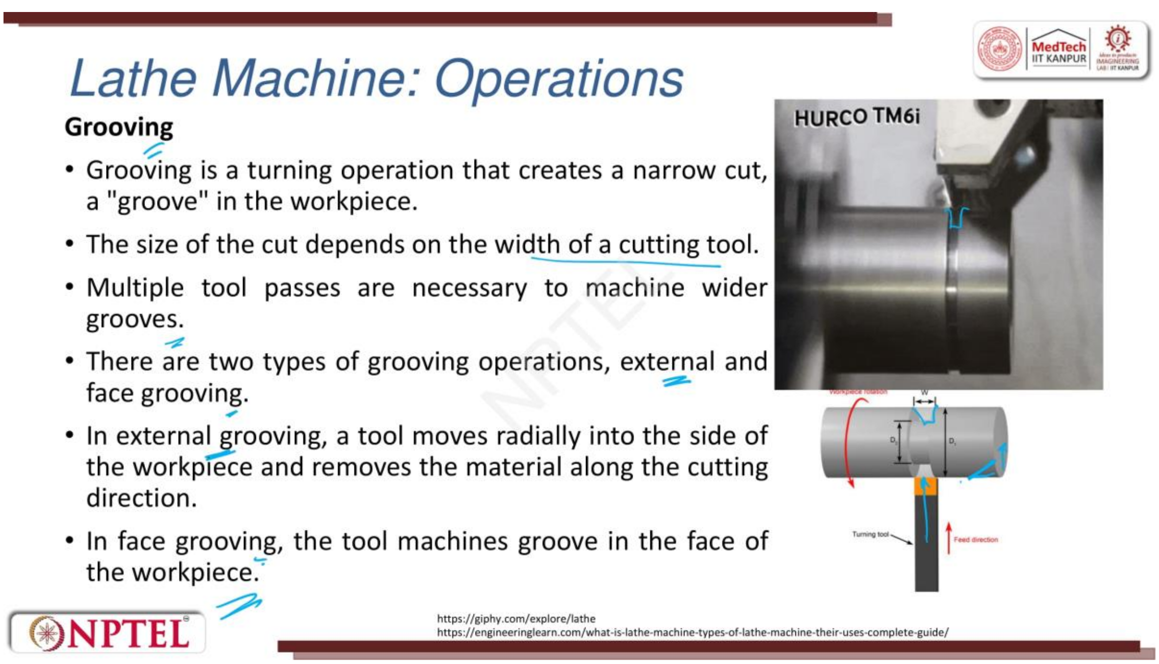
The contour turning is when you try to give a profile to the tool post. The tool moves in and out, thus creating a contour. That is called contour turning. In contour turning operation, the cutting tool follows the path with the predefined geometry. So, you can have a dial or a mandrel that comes in contact with a template.

So, that is in turn transferred to the tool post. The tool moves up and down to create the contour. Multiple passes of a contour tool are necessary to achieve the desired dimension. However, form tools are designed to generate it in a single pass. Facing is done.

So, here this is the axis. So, we are moving perpendicular to the axis. The tool is fed at an angle, and you try to create a face. During machining, the length of the workpiece is slightly longer than the final finished part should be.



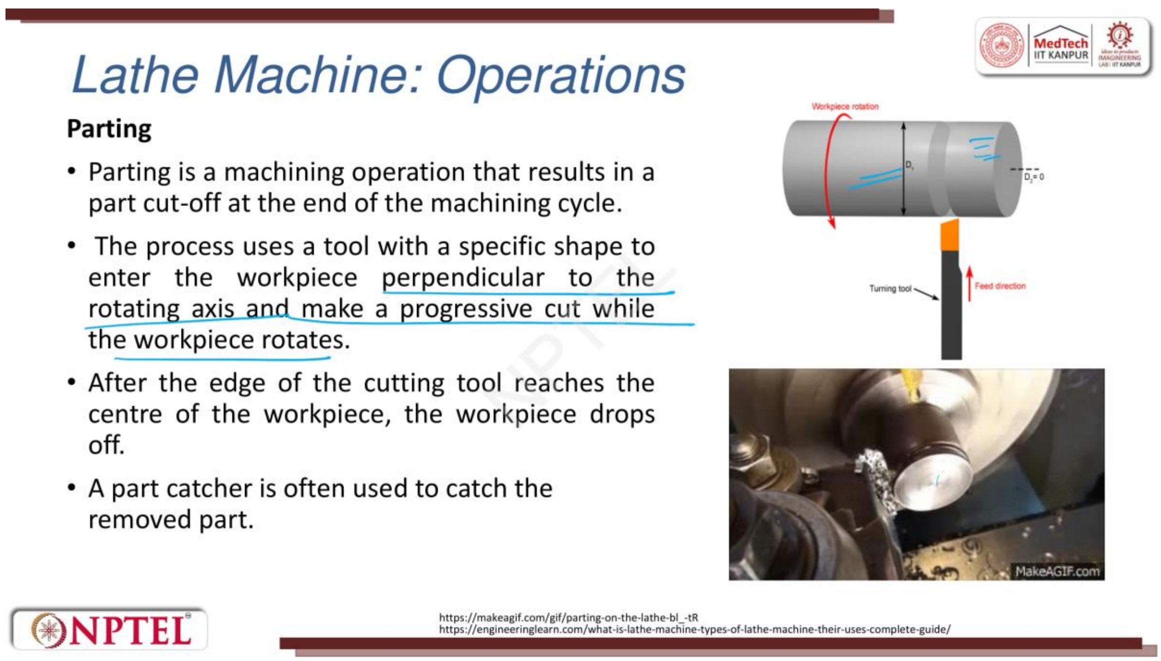
Facing is an operation performed at the end of the workpiece, perpendicular to the rotating axis. During facing, the tool moves along the radius of the workpiece to produce the desired path length and a smooth surface.



Grooving is something that is done in between. So, you can try to see: facing was this, turning was reducing the diameter, and taper turning was bringing the angle. Now, grooving is moving, offset it from that distance, and then move only perpendicular to a certain depth such that a groove is created. The size of the cut depends on the width of the cutting tool.

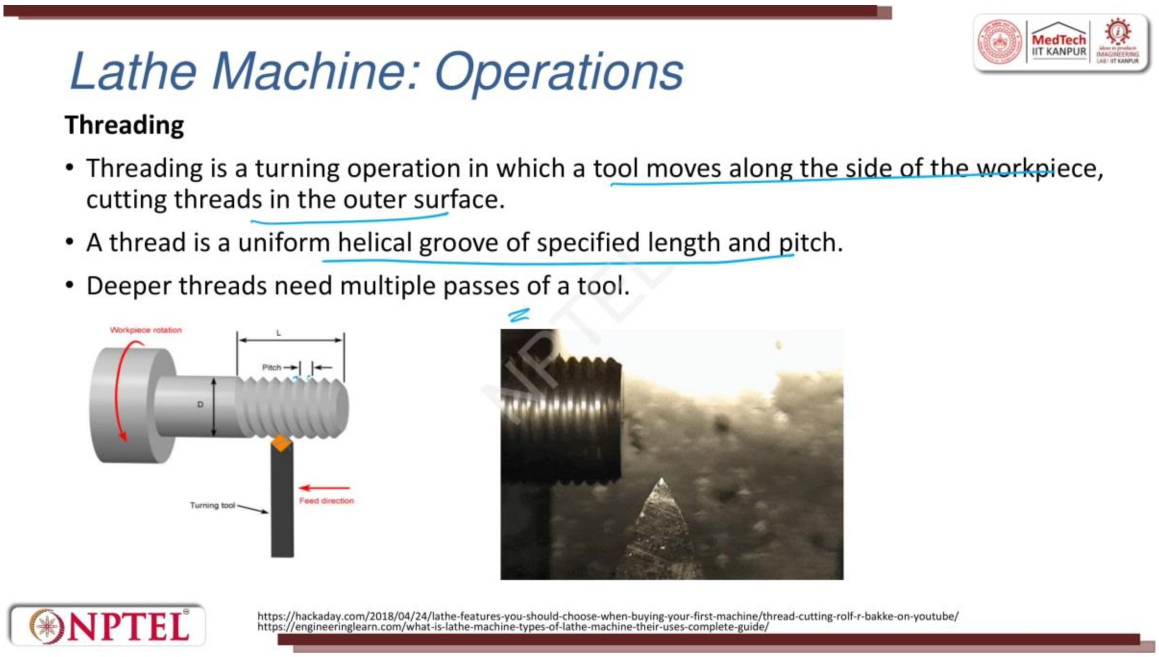
Multiple tool passes are necessary to create a groove. There are two types of groove operations. One is external grooving and face grooving. In external grooving, the tool moves radially into the side of the workpiece and removes the material along the cutting operation. That is called external grooving.

Face grooving is when the tool machines grooves on the face of the workpiece; this is called face grooving. So, this is external grooving, as shown to you.



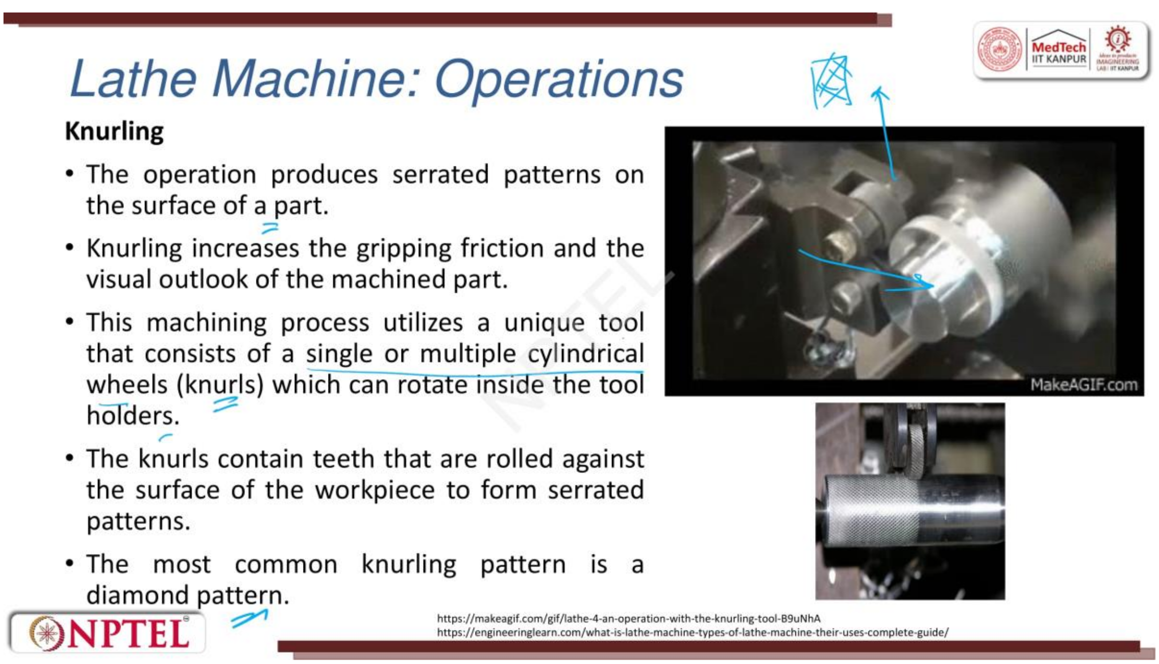
Then, parting or grooving: when the advancement happens along the depth, it tries to remove this material from the shaft. So, that is called parting. Slicing, something like slicing: you rotate it and then remove it.

That is called parting. The process uses a tool with a specific shape to enter the workpiece perpendicular to the rotating axis and makes a progressive cut while the workpiece rotates. So, this is used for cutting the workpiece, and the tool falls down. You can see here the workpiece is cut, and the disc has fallen down. The part catcher is often used to catch the part without damage.



This is the threading operation. Threading; you can see here there is a feed rate. There is a feed rate, and there is a pitch. So, there is a pitch between the threads, which is controlled by the tool geometry and the feed movement applied. Thread cutting is an operation in which the tool moves along the side of the workpiece, creating threads on the outer surface.

You can have internal threads or external threads. You can also have multi-start threads if desired. The threads consist of uniform helical grooves with a specified length and pitch. Deeper threads require multiple passes to generate.



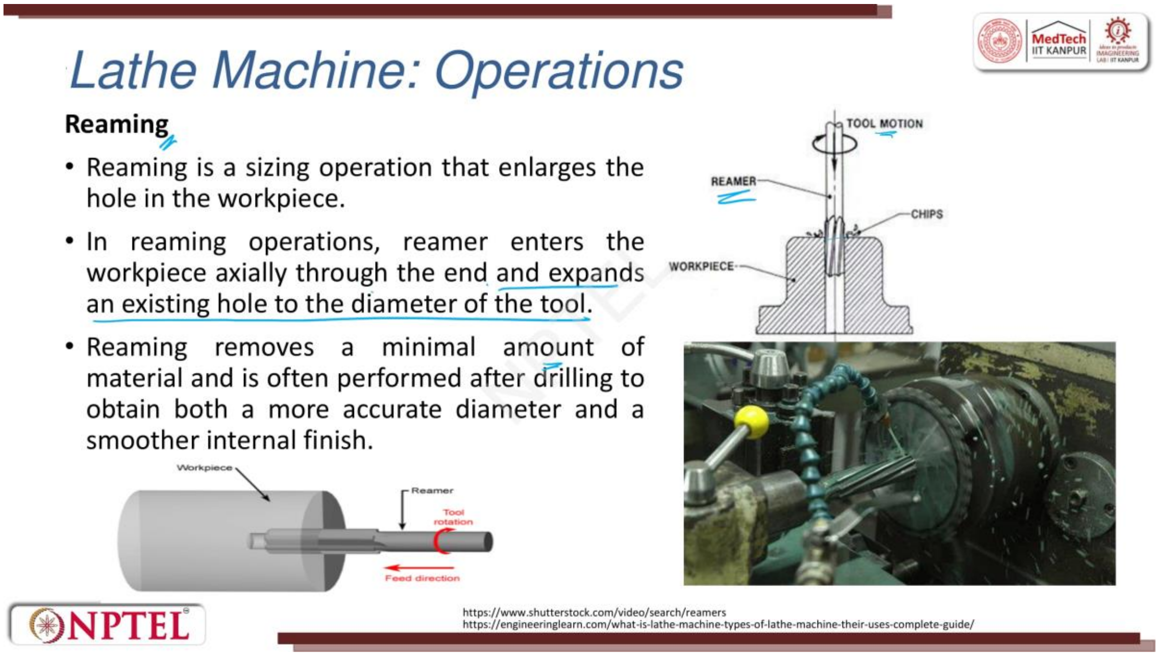
Knurling operation, as I said, involves a tool that moves around. So, you can see here this is a knurling tool which has a diamond-like cut on it. So, now what we do is, we just plunge this on top of a workpiece to create the knurling profile.

The operation produces a serrated pattern on the surface of a part to give a better grip. The machining process utilizes a unique tool which consists of a single or multiple cylindrical wheels called knurls, which can rotate inside the tool holder. The tool holder is there.

Inside the tool holder, these knurls plunge against the workpiece to perform this operation. So, they create diamond-like patterns.

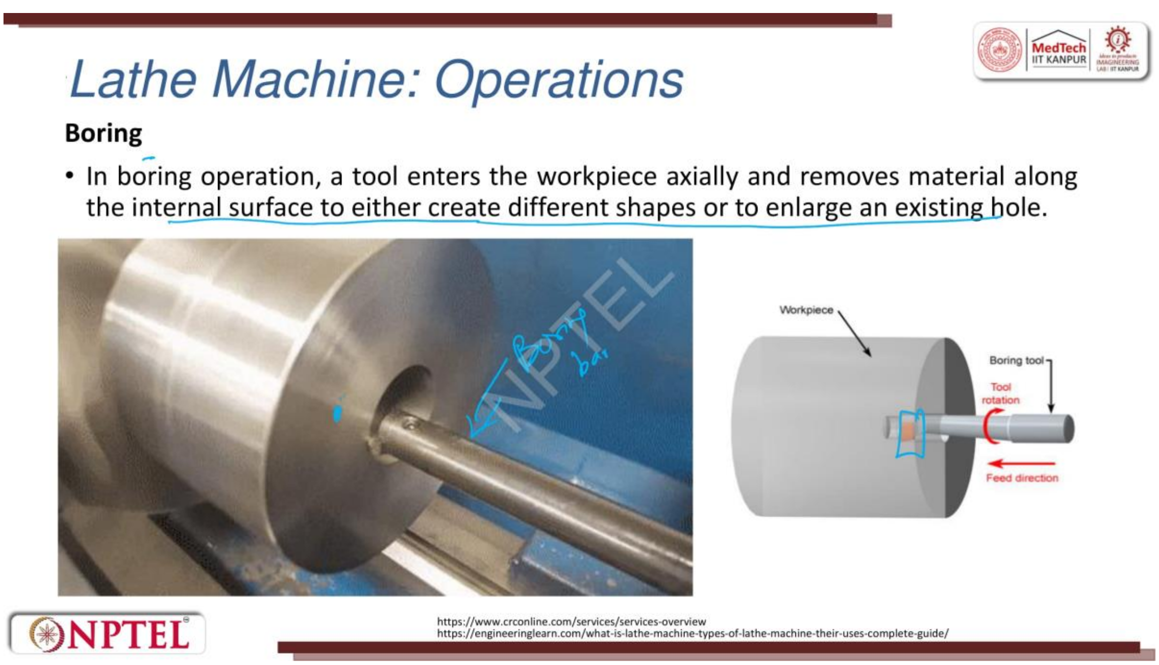
Drilling: the workpiece is rotating, and at the tailstock, you place a spindle and then mount a drill there. So, now you manually rotate the tailstock so that the drill advances and creates a hole. The drilling operation removes material from inside the workpiece.

The result of drilling is a hole with a diameter equal to the size of the drill bit used. The drill bit is usually positioned either on the tailstock or on the tool post. You can start removing material to create a hole.



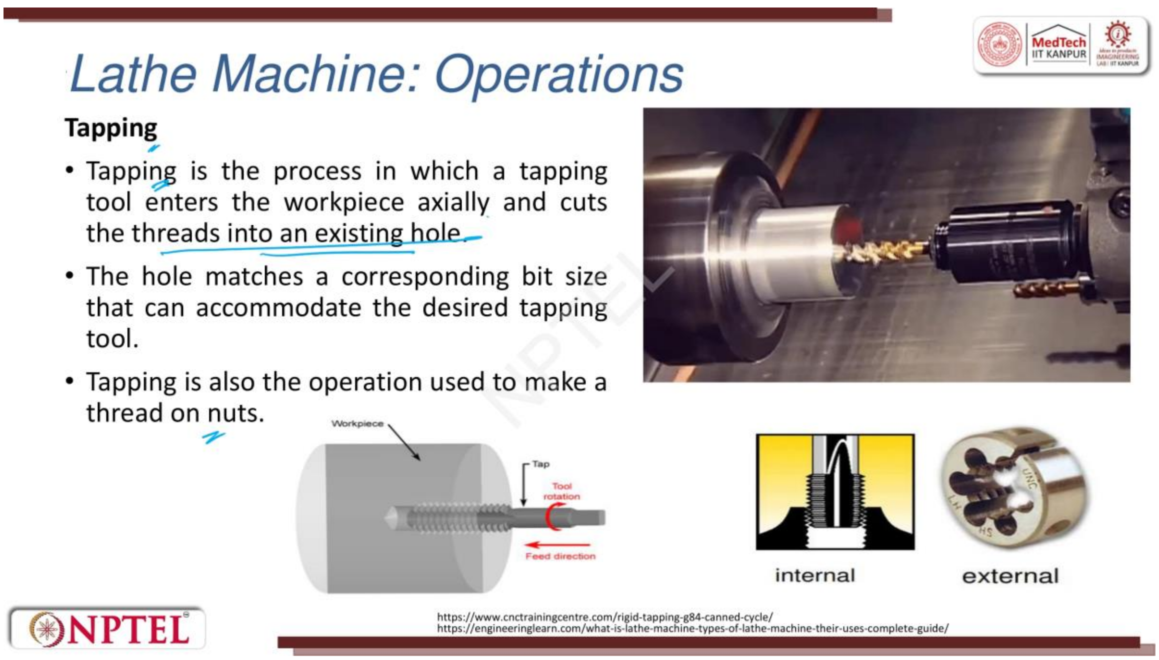
What is reaming? The hole creation is done by the drilling process. Reaming is an enlarging process. So, if you want to enlarge a hole and improve the hole's geometric profile, Then, we do reaming. So, in reaming here, the tool rotates, and the workpiece is mounted on the spindle. The tool rotates, the reamer enters, and then it tries to create or enlarge the hole.

Reaming operation: a reamer enters the workpiece axially through the end and expands the existing hole to the diameter of the tool. It removes a very minimal amount of material compared to drilling.



So, boring is an operation where, again, the tool is mounted on the tool post or it can be mounted on the tailstock. So, you can try to advance the bore. This is a boring bar. This is a boring bar. So, this boring bar has a boring tool here.

As a boring tool, this is the tool where rotation is given to the boring bar, and the feed direction is provided, so it enters and then tries to create or enlarge a hole. In the boring operation, the tool enters the workpiece axially and removes material along the inner surface to either create different shapes or enlarge the existing hole. So, reaming and boring will not generate a hole. Whatever hole was generated in drilling can be enhanced or improved by reaming and boring operations.

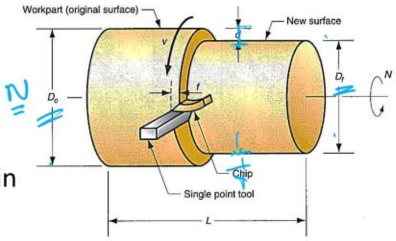


There is also a tapping operation. So, here what we do is replace the reamer with a tapping tool, which enters the workpiece and cuts a thread into the existing hole. A hole is there. Now, you insert a tap and create an internal thread.

The threading operation is external. The tapping operation creates an internal thread. The hole matches the corresponding bit size that can accommodate the desired tapping tool. It is used for making threads on nuts.

Lathe Machine: Cutting Analysis

- The rotational speed in turning is related to the desired cutting speed at the surface of the cylindrical workpiece by the equation.



$$\boxed{N = \frac{v}{\pi D_o}}$$

$V = \pi D_o N$

m/min

where N rotational speed, rev/min; v cutting speed, m/min (ft/min); and D_o = original diameter of the part, m (ft).

- The turning operation reduces the diameter of the work from its original diameter D_o to a final diameter D_f, as determined by the depth of cut d:

$D_o = D_f + 2d$

$$\mathbf{D_f = D_o - 2d}$$

- The feed in turning is generally expressed in mm/rev (in/rev). This feed can be converted to a linear travel rate in mm/min (in/min) by the formula

$$\mathbf{f_r = N f}$$

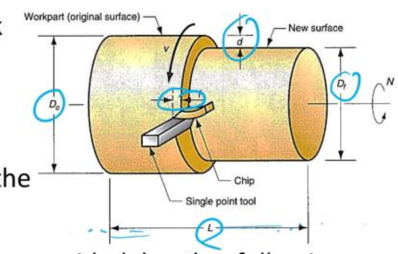
where f_r = feed rate, mm/min (in/min); and f = feed, mm/rev (in/rev).

NPTEL

M.P. Groover, *Fundamental of modern manufacturing Materials, Processes and systems, 4ed*

Lathe Machine: Cutting Analysis

- The time to machine from one end of a cylindrical work part to the other is given by



$$\boxed{T_m = \frac{L}{f_r}}$$

where T_m = machining time, min; and L length of the cylindrical work part, mm (in).

- A more direct computation of the machining time is provided by the following equation:

$$T_m = \frac{\pi D_o L}{f v}$$

- The volumetric rate of material removal can be most conveniently determined by the following equation:

$$\boxed{\mathbf{R_{MR} = vfd}}$$

R_{MR} = material removal rate, mm^3 /min (in^3/min).

NPTEL

M.P. Groover, *Fundamental of modern manufacturing Materials, Processes and systems, 4ed*

Now, let us try to see a little bit of the cutting analysis. So, the rotating speed in turning is related to the desired cutting speed at the surface of the cylindrical workpiece, which is given by this equation. So, here I can rewrite the equation: $v = \pi DN$. So, where π is the circumferential constant, D is the original diameter, and N is the rotational speed in RPM. So, if you want to find the cutting speed v, it is nothing but $\pi \times D \times N$, where N is the speed, v is the cutting speed in meters per minute, and D_o is the original diameter D. So, when I try to reduce a diameter from D_o to D final, it can be done like this.

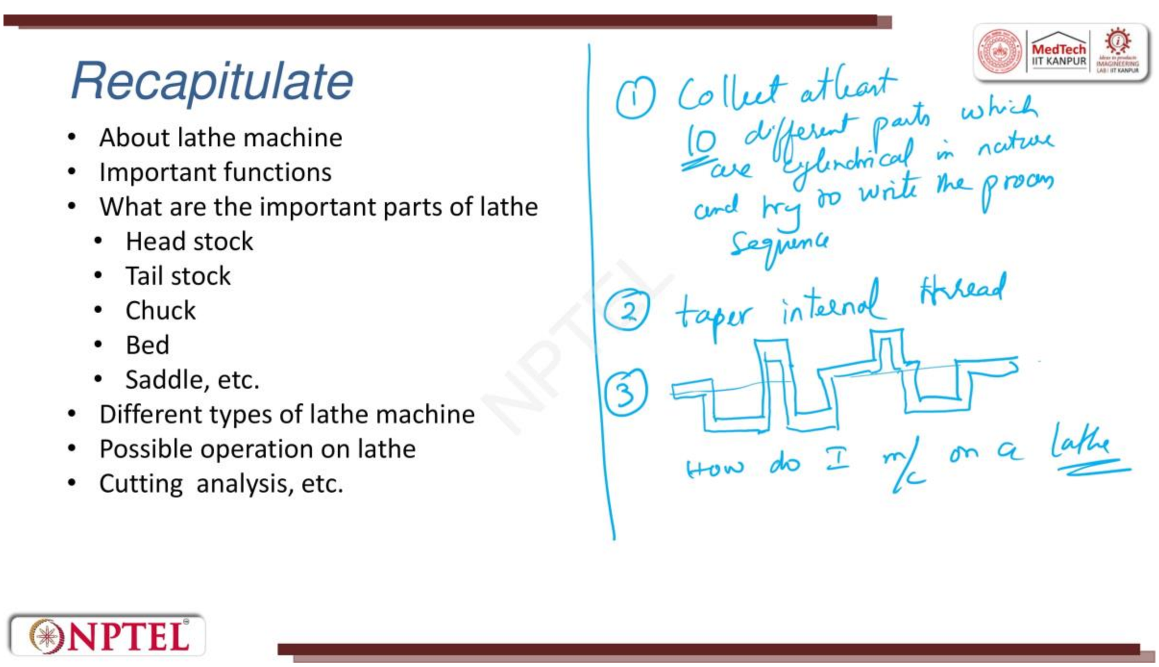
$$D_o = D_f + 2d$$

Do is the original diameter. So, if you see here, D is this.

So, 2D. So, the Do is here, and Df is here and 1D, 2D. This is 2D. So, the turning operation reduces the diameter of the workpiece from its original diameter Do to DF. So, the relationship between Do and Df is Do = Df + 2D. If we want to find out the feed in turning, it is generally expressed in millimeters per minute or millimeters per revolution. The feed rate is nothing but N times the feed. What is N? N is the rotational speed or the spindle speed.

So, the time of machining will be the length of machining divided by the feed rate, which will give you the time of machining. So, if you see here, Do, DF, DF, and this is L. Now, the time required for machining will be the length of cut divided by the feed rate FR. So, in order to have a more direct computation of the machining time, this T will be pi Do into L by fv, which is the feed rate. So, feed rate multiplied by velocity. The volumetric rate of material removal will be v into f into d. This is the volumetric rate of material removal.

So, with this, you will try to get a feel for what the process parameters are and how these process parameters affect the machining time.



So, friends, to recap what we saw in this lecture. So, before we recap, let us try to do these three simple exercises. First, collect at least ten different parts that are cylindrical in nature and try to write the process sequence. What comes first, what comes next, and what are all the operations involved? Collect ten different parts. For example, camshaft, crankshaft, or whatever you want to do.

Next, try to visualize: if I wanted to have a taper internal thread, first of all, is it possible? If possible, how will you mount the tool, and what will be the processing sequence for generating taper internal threads? The third thing is, if I wanted to have a shift, for example, let us take a crankshaft, something like this. So, there is an offset. If there is an offset, how do I machine it on a lathe machine?

So, these are the three things which I would request you to try and practice or observe, and then you will understand how we go about it. Well, friends, before I wind up this lecture, here is a recap of what we have done till now: first, we tried to understand a lathe machine, then we covered its important functions and the major parts of a lathe machine. Then, we discussed different types of operations in a lathe machine and different types of lathe machines. Finally, we did some cutting analysis, wherein we tried to identify process parameters that can be used for machining. And finally, we correlated all these process parameters with respect to time.

So, we learned how to calculate the machining time.

Thank you.