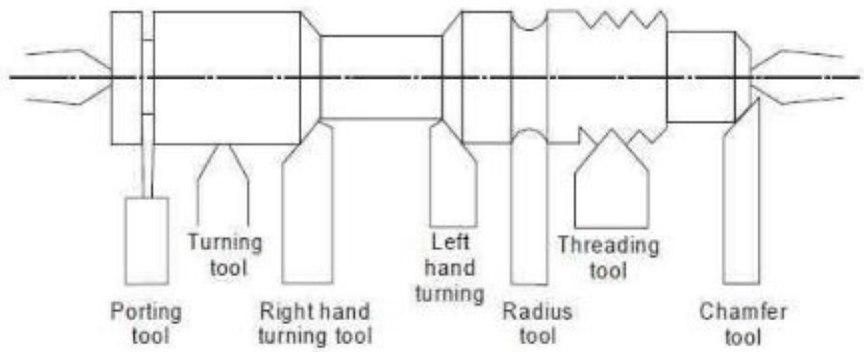
Lathe Operations

For performing the various machining operations in a lathe, the job is being supported and driven by anyone of the following methods.

1. Job is held and driven by chuck with the other end supported on the tail stock centre.
2. Job is held between centers and driven by carriers and catch plates.
3. Job is held on a mandrel, which is supported between centers and driven by carriers and catch plates.
4. Job is held and driven by a chuck or a faceplate or an angle plate.

The above methods for holding the job can be classified under two headings namely job held between centers and job held by a chuck or any other fixture. The operations performed in a lathe can be understood by three major categories



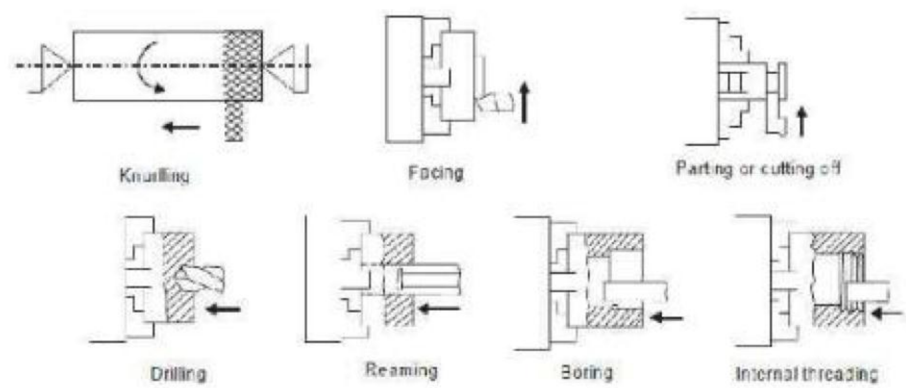
Lathe operation

(a) Operations, which can be performed in a lathe either by holding the workpiece between centers or by a chuck are:

- Straight turning
- Shoulder turning
- Taper turning
- Chamfering
- Eccentric turning
- Thread cutting
- Facing
- Forming
- Filing
- Polishing
- Grooving
- Knurling
- Spinning
- Spring winding

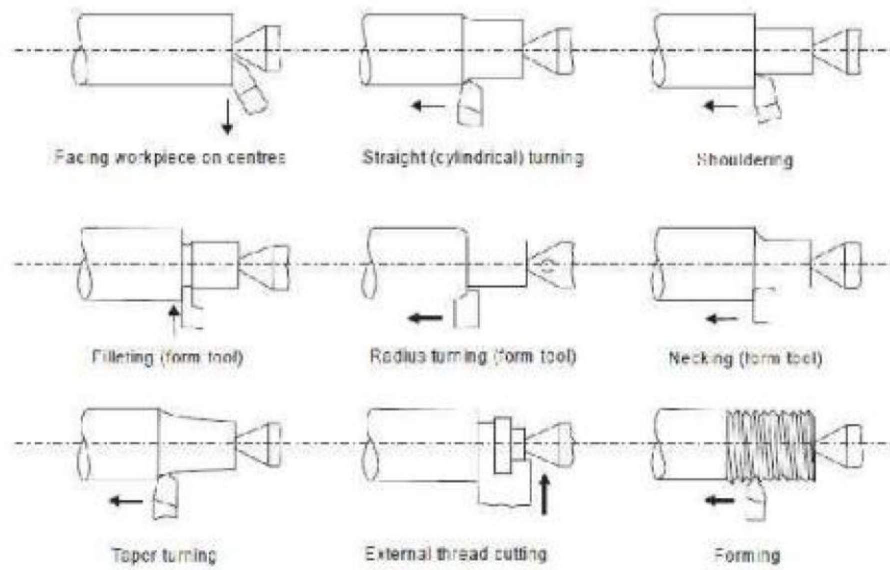
(b) Operations which are performed by holding the work by a chuck or a faceplate or an angle plate are:

- Undercutting
- Parting-off
- Internal thread cutting
- Drilling
- Reaming
- Boring
- Counter boring
- Taper boring
- Tapping



(c) Operations which are performed by using special lathe attachments are:

- Milling
- Grinding



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

Tapers and Taper Turning

A taper is defined as a uniform increase or decrease in diameter of a piece of work measured along its length. In a lathe machine, taper turning means to produce a conical surface by gradual reduction in diameter from a cylindrical job. Taper in the British System is expressed in taper per foot or taper per inch.

$$\textbf{Taper per inch} = (D - d)/l$$

Where,

- D = is the diameter of the large end of cylindrical job,
- d = is the diameter of the small end of cylindrical job, and
- l = is the length of the taper of cylindrical job, all expressed in inches

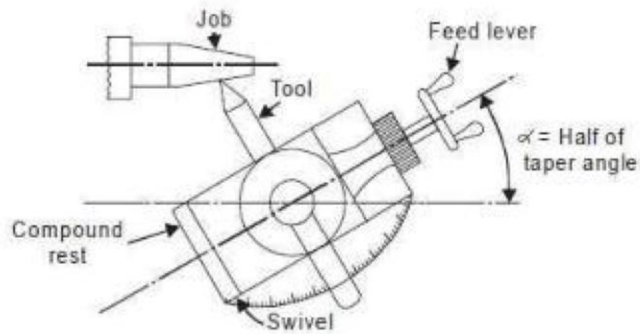
When the taper is expressed in taper per foot, the length of the taper l is expressed in foot, but the diameters are expressed in inches. A taper is generally turned in a lathe by feeding the tool at an angle to the axis of rotation of the workpiece. The angle formed by the path of the tool with the axis of the workpiece should correspond to the half taper angle. A taper can be turned by anyone of the following methods:

1. By swiveling the compound rest,
2. By setting over the tailstock centre,
3. By a broad nose form tool,
4. By a taper turning attachment,
5. By combining longitudinal and cross feed in a special lathe and
6. By using numerical control lathe

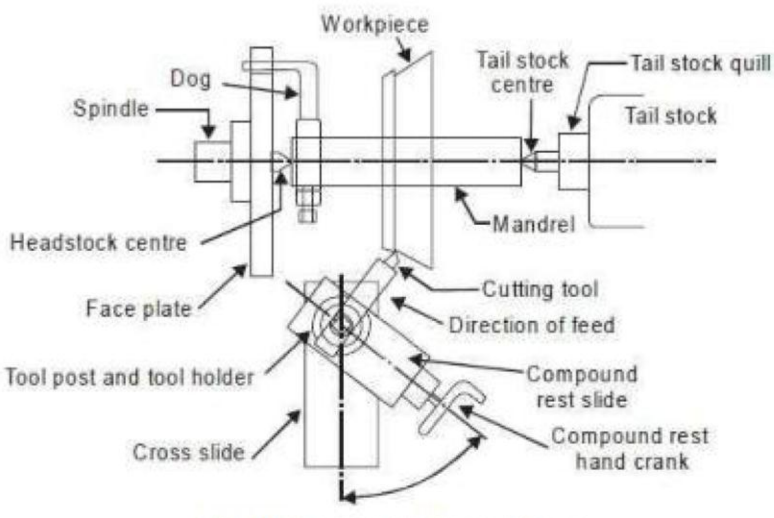
Taper Turning by Swivelling the Compound Rest

This method uses 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 is mounted on the compound rest which is attached to a circular base, graduated in degrees. The compound rest can easily be swiveled or rotated 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 but steep taper because of the limited movement of the cross-slide. The compound rest can be swiveled at 45° on either side of the lathe axis enabling it to turn a steep taper. The movement of the single point cutting tool in this method is being purely controlled by hand. Thus it provides a low production capacity and poor surface finish. The positioning or setting of the compound rest is accomplished by swiveling 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 taper angle can be calculated. The complete setup for producing a taper by swelling the compound rest.



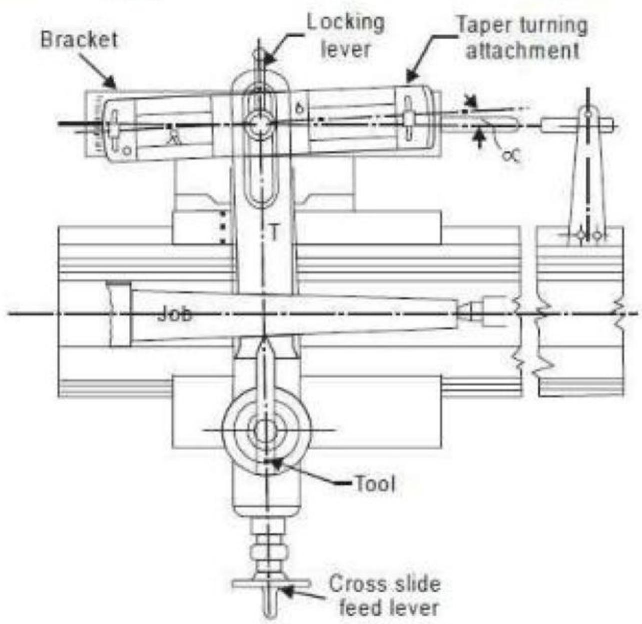
Taper turning by swivelling compound rest



Swivelling compound rest set-up

Taper Turning Attachment Method

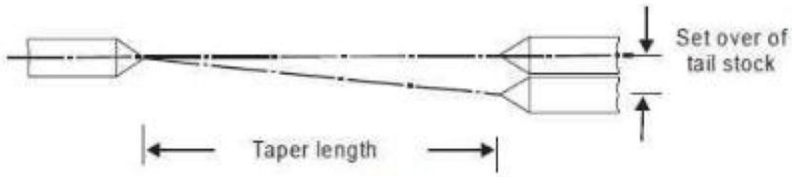
This method is commonly employed for generating external tapers only. In this method, the taper turning attachment is bolted back of the lathe machine has guide bar which may be set at any desired angle or taper. As the carriage moves along the bed length aside over bar causes the tool to move in and out according to setting of the bar. The taper setting on the bar is duplicated on the job or work. The merit of this method is that the lathe centres are kept in alignment.



Taper turning attachment

Taper Turning with Tailstock set over Method

This method is basically employed for turning small tapers on longer jobs and is confined to external tapers only. In this method, the tailstock is set over is calculated by loosening the nut from its centre line equal to the value obtained by formula given below



Tailstock set over

- Tail stock set over = Taper length × Sine of half of taper angle
- $(D - d) / 2 = l \times \sin(a/2)$

Where,

- D = is the diameter of the large end of cylindrical job,
- d = is the diameter of the small end of cylindrical job, and
- l = is the length of the taper of cylindrical job, all expressed in inches,
- a = taper angle

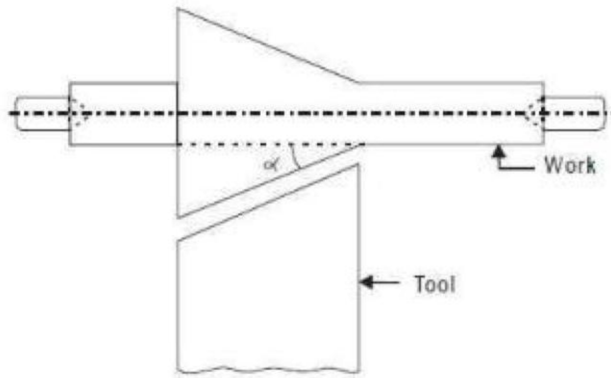
When a part length of the job is to be given taper then tail stock set

$= ((D - d)/2)) \times$ (total length of the cylindrical job/length of taper)

$= l \times \sin(a/2) \times$ (total length of the cylindrical job/length of taper)

Form Tool Method

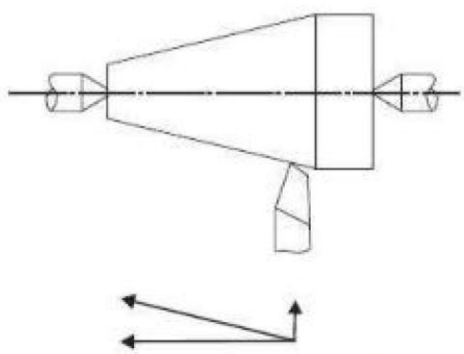
It is limited to short external tapers. The edge tool must be exactly straight for accurate work.



Form tool taper turning

Taper Turning with Double Feeds

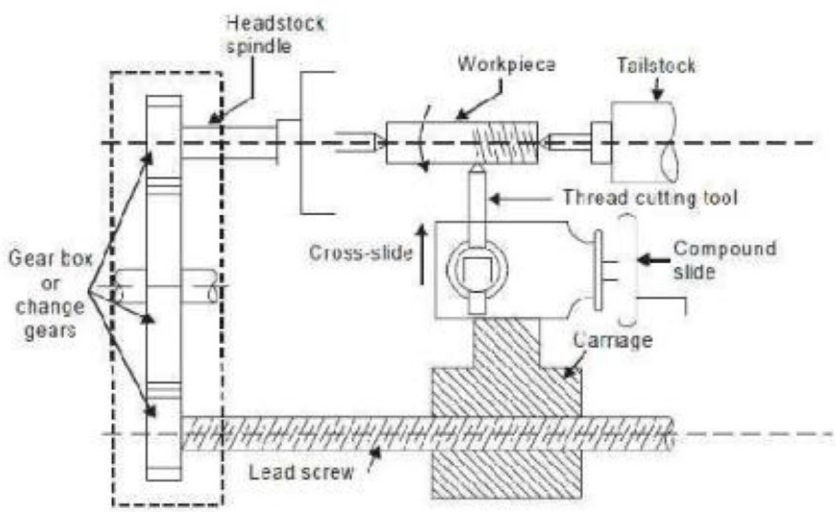
Taper turning can also be cut by combining the two feeds. In this arrangement of taper turning, which is good method of taper turning. In certain lathes both longitudinal and cross feeds may be engaged simultaneously causing the tool to follow a diagonal point which is the resultant of the magnitude of the two feeds. The direction of resultant feed may be changed by varying the rate of feeds by change gears provided inside the apron of the lathe.



Taper turning using combined feeds

Thread Cutting

Thread of any pitch, shape and size can be cut on a lathe using single point cutting tool. Thread cutting is operation of producing a helical groove on spindle shape such as V, square or power threads on a cylindrical surface. The job is held in between centres or in a chuck and the cutting tool is held on tool
post. The cutting tool must travel a distance equal to the pitch (in mm) as the work piece completes a revolution. The definite relative rotary and linear motion between job and cutting tool is achieved by locking or engaging a carriage motion with lead screw and nut mechanism and fixing a gear ratio between head stock spindle and lead screw. To make or cut threads, the cutting tool is brought to the start of job and a small depth of cut is given to cutting tool
using cross slide.



Thread cutting

Cutting Speed

Cutting speed for lathe work may be defined as the rate in meters per minute at which the surface of the job moves past the cutting tool. Machining at a correct cutting speed is highly important for good tool life and efficient cutting. Too slow cutting speeds reduce productivity and increase manufacturing costs whereas too high cutting speeds result in overheating of the tool and premature failure of the cutting edge of the tool.

The following factors affect the cutting speed:

(i) Kind of material being cut,

(ii) Cutting tool material,

(iii) Shape of cutting tool,

(iv) Rigidity of machine tool and the job piece and

(v) Type of cutting fluid being used.

Calculation of cutting speed Cs, in meters per minute

$$Cs = ((22/7) \times D \times N))/1000$$

Where

- D is diameter of job in mm.
- N is in RPM

Feed

Feed is defined as the distance that a tool advances into the work during one revolution of the headstock spindle. It is usually given as a linear movement per revolution of the spindle or job. During turning a job on the centre lathe, the saddle and the tool post move along the bed of the lathe for a particular feed for cutting along the length of the rotating job.

Special Attachments

Unless a workpiece has a taper machined onto it which perfectly matches the internal taper in the spindle, or has threads which perfectly match the external threads on the spindle (two conditions which rarely exist), an accessory must be used to mount a workpiece to the spindle. A workpiece may be bolted or screwed to a faceplate, a large, flat disk that mounts to the spindle. In the alternative, faceplate dogs may be used to secure the work to the faceplate. A workpiece may be mounted on a mandrel, or circular work clamped in a three- or four-jaw chuck. For irregular shaped work pieces it is usual to use a four jaw (independent moving jaws) chuck. These holding devices mount directly to the Lathe headstock spindle.

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

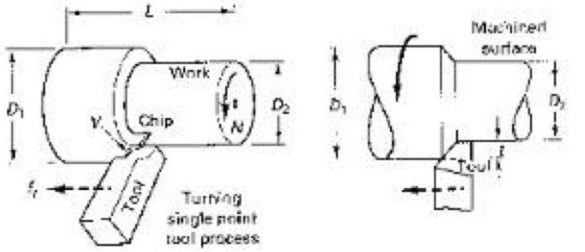
A soft workpiece (e.g., wood) may be pinched between centres by using a spur drive at the headstock, which bites into the wood and imparts torque to it.

Machining time

Machining time is the time when a machine is actually processing something. Generally, machining time is the term used when there is a reduction in material or removing some undesirable parts of a material. For example, in a drill press, machining time is when the cutting edge is actually moving forward and making a hole. Machine time is used in other situations, such as when a machine installs screws in a case automatically. One of the important aspects in manufacturing calculation is how to find and calculate the machining time in a machining operation. Generally, machining is family of processes or operations in which excess material is removed from a starting work piece by a sharp cutting tool so the remaining part has the desired geometry and the required shape. The most common machining operations can be classified into four types: turning, milling, drilling and lathe work.

Calculate Time for Turning

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



A. Machine Speed: To achieve a specific cutting speed:

$$N = \frac{k * V}{\pi * D_i}$$

N = machine speed in revolutions/minute (RPM)

k is a constant to "correct" speed (V) and part diameter (D_i) units

V is desired cutting speed, a Handbook Value

D_1 is largest part diameter (initial size)

V given in surface feet per minute (SFPM), D_1 in inches: k = 12

V given in meters per second (MPS), D_1 in mm: k = 60000

V given in meters per minute (MPM), D_1 in mm: k = 1000

If Cutting Speed for a given RPM rate is desired, solve above equation for

$$V: V = \pi ND/k$$

B. Cutting Time: minutes per operation

CT is cutting time per "pass"

L is length cut

A is "allowance" or starting offset

f_r is machine feed rate units/revolution, a Handbook Value

$$CT = \frac{(L + A)}{f_r * N}$$