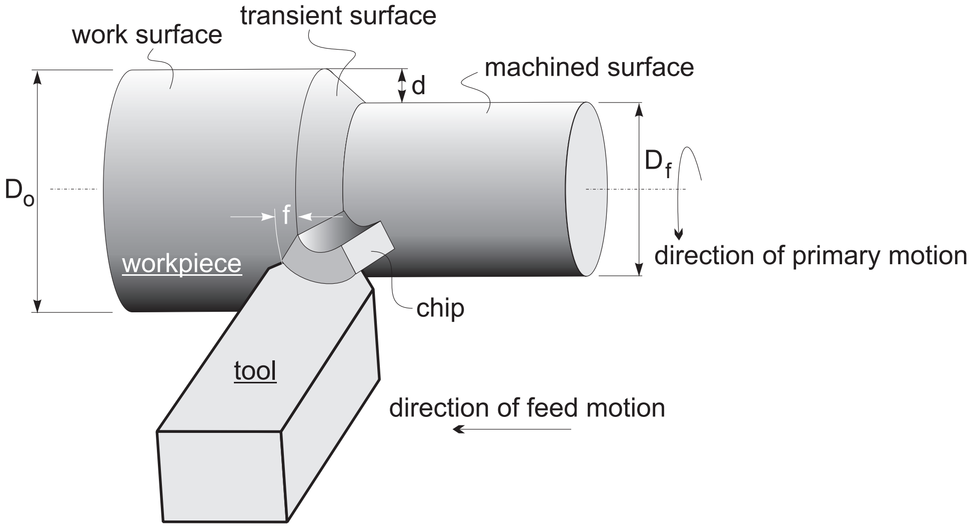
6 MACHINING OPERATIONS

CHAPTER CONTENTS

6.1 TURNING

Introduction

Turning is a machining process to produce parts round in shape by a single point tool on *lathes*. The tool is fed either linearly in the direction parallel or perpendicular to the axis of rotation of the workpiece, or along a specified path to produce complex rotational shapes. The *primary* motion of cutting in turning is the rotation of the workpiece, and the *secondary* motion of cutting is the feed motion.



Turning operation

Cutting conditions in turning

Cutting speed in turning V in m/s is related to the rotational speed of the workpiece by the equation:

$$V = \pi DN$$

where D is the diameter of the workpiece, m; N is the rotational speed of the workpiece, rev/s.

One should remember that cutting speed V is always a linear vector. In the process planning of a turning operation, cutting speed V is first selected from appropriate reference sources or calculated as discussed in *Section 5.10 Selection of Cutting Conditions*, and the rotational speed N is calculated taking into account the workpiece diameter D. Rotational speed, not cutting speed, is then used to adjust lathe setting levers.

Feed in turning is generally expressed in mm tr^{-1} (millimetres per revolution).

The turning operation reduces the diameter of the workpiece from the initial diameter D_o to the final diameter D_f. The change in diameter is actually two times *depth of cut*, d:

$$2d = D_o - D_f$$

The volumetric rate of material removal (so-called *material removal rate*, *mrr*) is defined by

$$mrr = Vfd$$

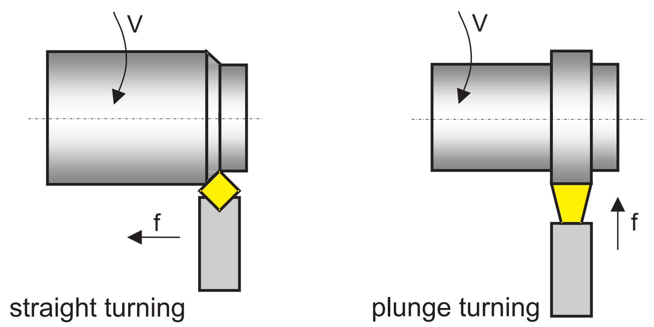
When using this equation, care must be exercised to assure that the units for V are consistent with those for f and d.

Operations in turning

Turning is not a single process but class of many and different operations performed on a lathe.

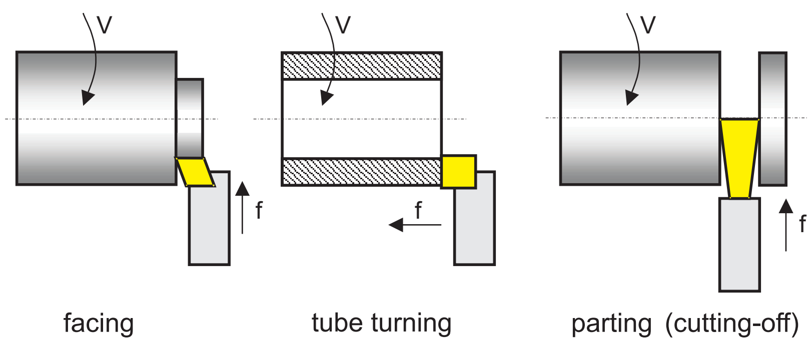
Turning of cylindrical surfaces

The lathe can be used to reduce the diameter of a part to a desired dimension. The resulting machined surface is cylindrical.



Turning of flat surfaces

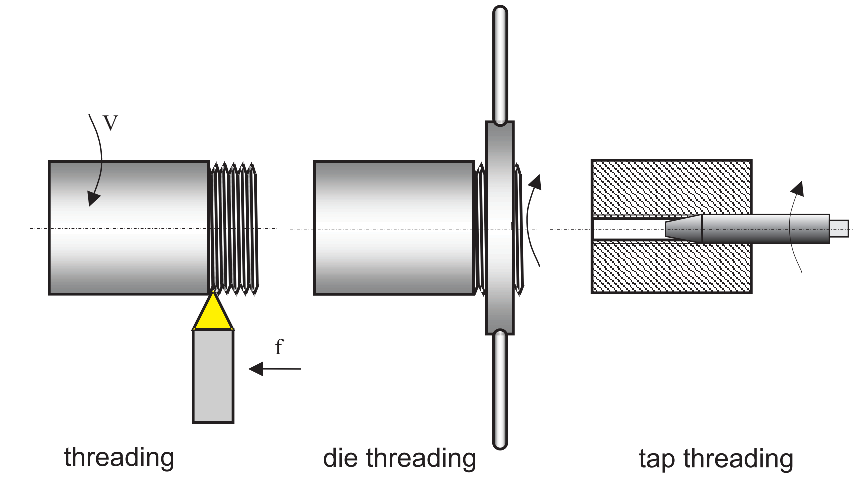
A lathe can be used to create a smooth, flat face very accurately perpendicular to the axis of a cylindrical part. Tool is fed radially or axially to create a flat machined surface.



Threading

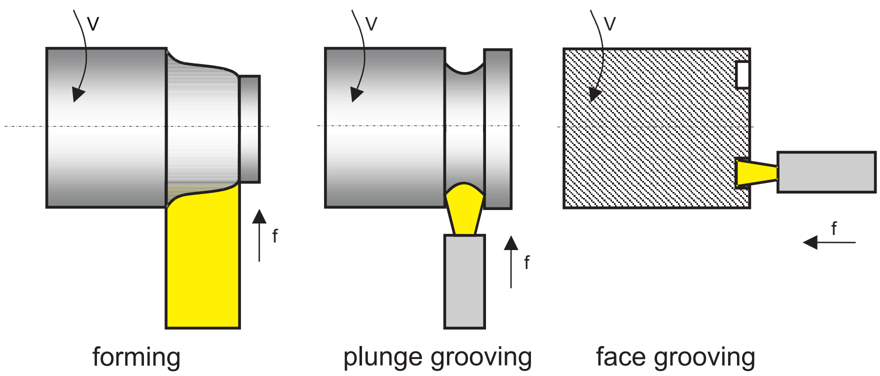
Different possibilities are available to produce a thread on a lathe. Threads are cut using lathes by advancing the cutting tool at a feed exactly equal to the thread pitch. The single-point cutting tool cuts in a helical band, which is actually a thread. The procedure calls for correct settings of the machine, and also that the helix be restarted at the same location each time if multiple passes are required to cut the entire depth of thread. The tool point must be ground so that it has the same profile as the thread to be cut.

Another possibility is to cut threads by means of a *thread die* (external threads), or a *tap* (internal threads). These operations are generally performed manually for smal thread diameters.



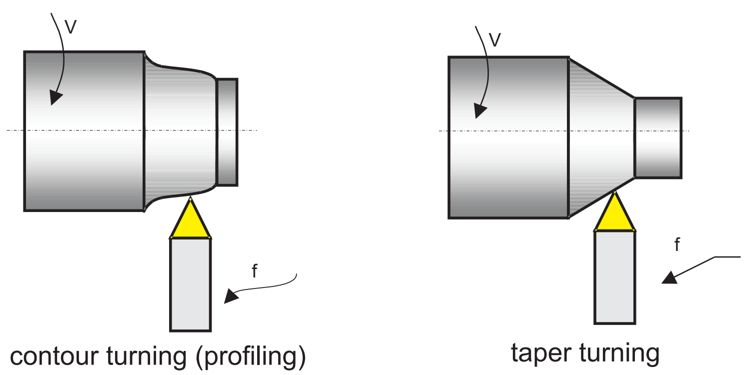
Form turning

Cutting tool has a shape that is imparted to the workpiece by plunging the tool into the workpiece. In form turning, cutting tool is complex and expensive but feed is linear and does not require special machine tools or devices.



Contour turning (profiling)

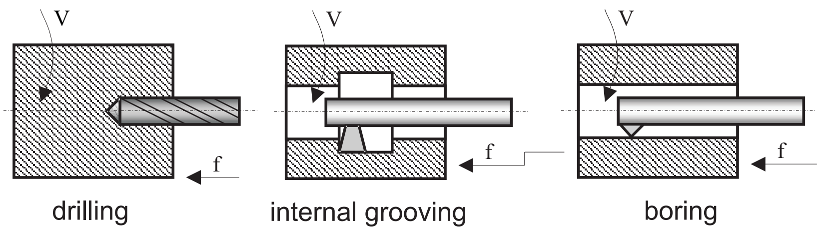
Cutting tool has a simple shape, but the feed motion is complex; cutting tool is fed along a contour thus creating a contoured shape on the workpiece. For profiling, special lathes or devices are required.



Producing tapers on a lathe is a specific task and contour turning is just one of the possible solutions. Some other methods for turning tapers are discussed later.

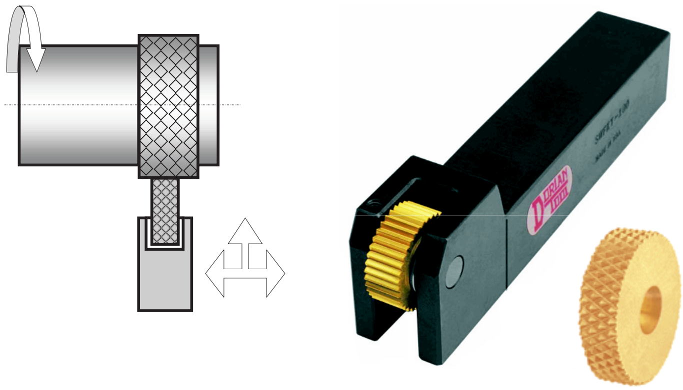
Miscellaneous operations

Some other operations, which do not use the single-point cutting tool can be performed on a lathe, making turning one of the most versatile machining processes.



Knurling

This is not a machining operation at all, because it does not involve material removal. Instead, it is a metal forming operation used to produce a regular crosshatched pattern in the work surface.



(*Left*) Knurling operation; (*Right*) Knurling tool and knurling wheel. Wheels with different patterns are easily available.

Lathes

A lathe is a machine tool that rotates the workpiece against a tool whose position it controls. The *spindle* (see picture in the next page) is the part of the lathe that rotates. Various work holding attachments such as *three jaw chucks*, *collets*, and *centers* can be held in the spindle. The spindle is driven by an electric motor through a system of belt drives and gear trains. Spindle rotational speed is controlled by varying the geometry of the drive train.

The *tailstock* can be used to support the end of the workpiece with a *center*, or to hold tools for drilling, reaming, threading, or cutting tapers. It can be adjusted in position along the *ways* to accommodate different length workpieces. The *tailstock barrel* can be fed along the axis of rotation with the *tailstock hand wheel*.

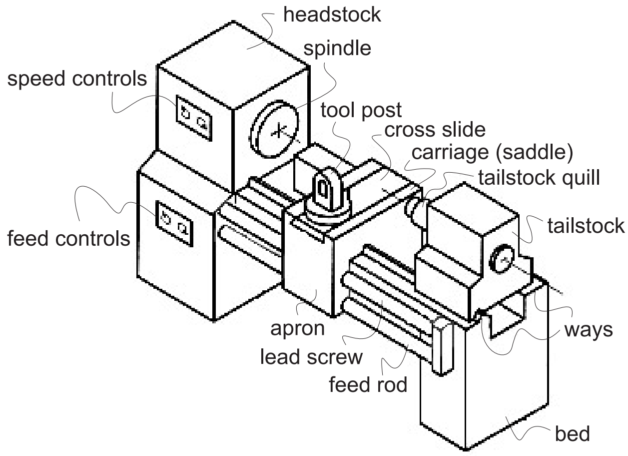
The *carriage* controls and supports the cutting tool. It consists of:

- a *saddle* that slides along the *ways*;
- an *apron* that controls the feed mechanisms;
- a *cross slide* that controls transverse motion of the tool (toward or away from the operator);
- a *tool compound* that adjusts to permit angular tool movement;
- a *tool post* that holds the cutting tools.

There are a number of different lathe designs, and some of the most popular are discussed here.

Engine lathes

The basic, simplest and most versatile lathe. This machine tool is manually operated that is why it requires skilled operators. Suitable for low and medium production, and for repair works.



The principal components of an engine lathe

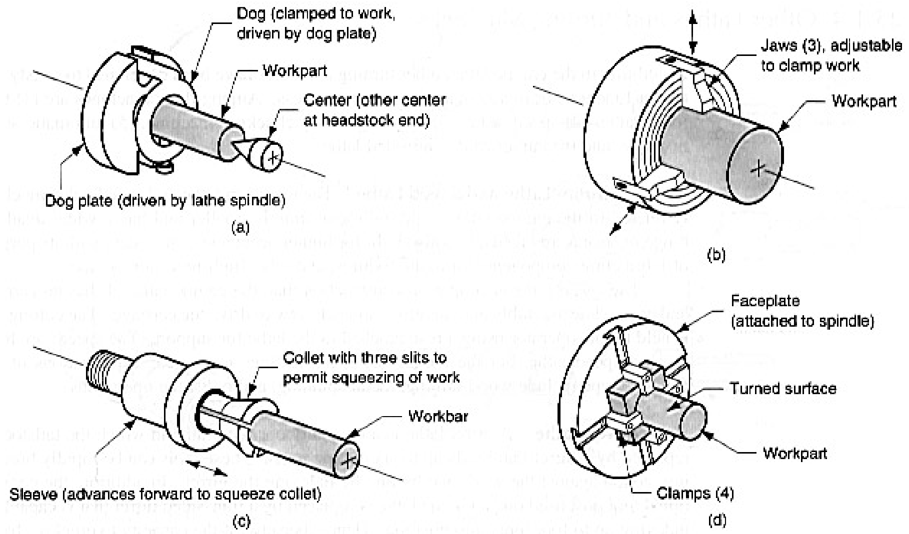
There are two tool feed mechanism in the engine lathes. These cause the cutting tool to move when engaged.

❶ The *lead screw* will cause the apron and cutting tool to advance quickly. This is used for cutting threads, and for moving the tool quickly.

❷ The *feed rod* will move the apron and cutting tool slowly forward. This is largely used for most of the turning operations.

Work is held in the lathe with a number of methods,

- ❖ Between two *centres*. The workpiece is driven by a device called a *dog*; The method is suitable for parts with high *length-to-diameter ratio*.
- ❖ A *3 jaw self-centering chuck* is used for most operations on cylindrical workparts. For parts with high *length-to-diameter ratio* the part is supported by center on the other end.
- ❖ *Collet* consists of tubular bushing with longitudinal slits. Collets are used to grasp and hold barstock. A collet of exact diameter is required to match any barstock diameter.
- ❖ A *face plate* is a device used to grasp parts with irregular shapes:



Four work holding methods used in lathes: (a) mounting the work between centers using a dog, (b) three-jaw chuck, (c) collet, and (d) face plate for noncylindrical workparts.

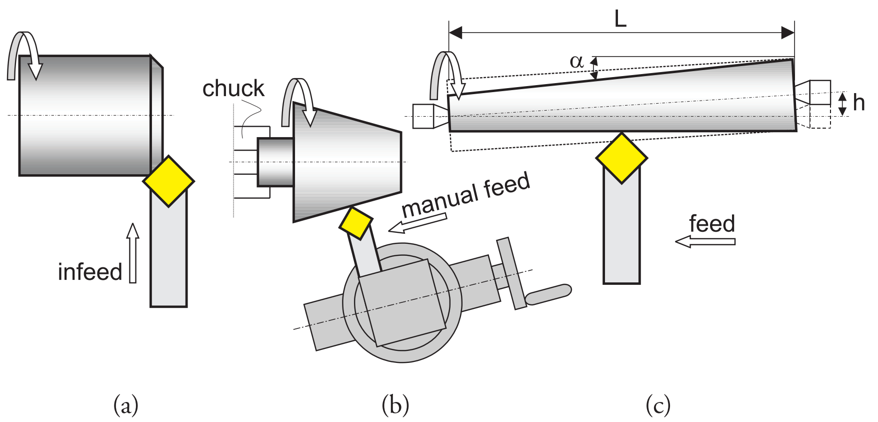
Turning tapers on engine lathes

A taper is a conical shape. Tapers can be cut with lathes quite easily. There are some common methods for turning tapers on an engine lathe,

- Using a *form tool*: This type of tool is specifically designed for one cut, at a certain taper angle. The tool is plunged at one location, and never moved along the lathe slides.
- *Compound Slide Method*: The compound slide is set to travel at half of the taper angle. The tool is then fed across the work by hand, cutting the taper as it goes.
- *Off-Set Tail Stock*: In this method the normal rotating part of the lathe still drives the workpiece (mounted between centres), but the centre at the tailstock is offset towards/away from the cutting tool. Then, as the cutting tool passes over, the part is cut in a conical shape. This method is limited to small tapers over long lengths. The tailstock offset h is defined by

$$h = L\sin\alpha$$

where L is the length of workpiece, and α is the half of the taper angle.



Three methods for turning tapers on an engine lathe: (a) using a form tool, (b) the compound slide method, and (c) offsetting tailstock.

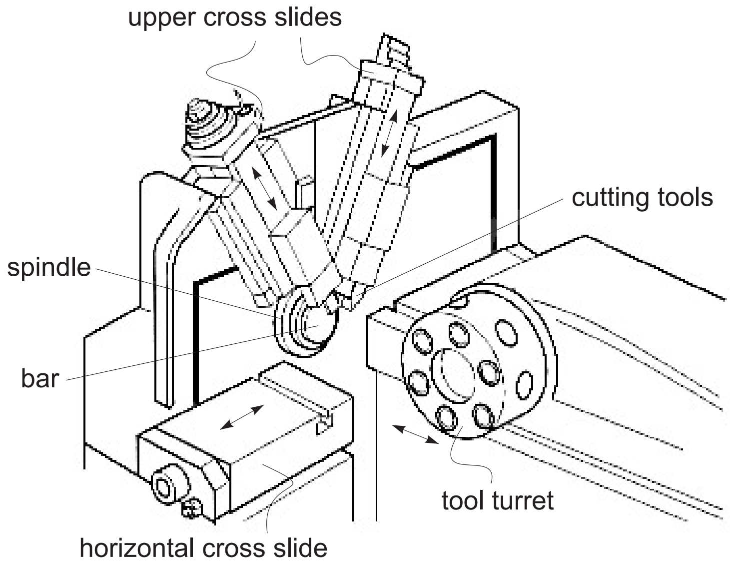
Turret lathes

These machines are capable of carrying out multiple cutting operations on the same workpiece. Several cutting tools are mounted on a *tetra-*, *penta-*, or *hexagonal turret*, which replaces the tailstock. These tools can be rapidly brought into action against the workpiece one by one by indexing the turret. In some machines four additional tools are mounted in a square turret on the cross slide, or two or three more tools are mounted in tool posts on several cross slides. Turret lathes are used for high-production work. The up-to-date lathes are numerically controlled as discussed later.

(*Left*) Turret lathe; (*Right*) Close-up view of a turret lathe showing a set of three octagonal turrets with a total number of 24 different cutting tools, and the bar workpiece held in a collet.

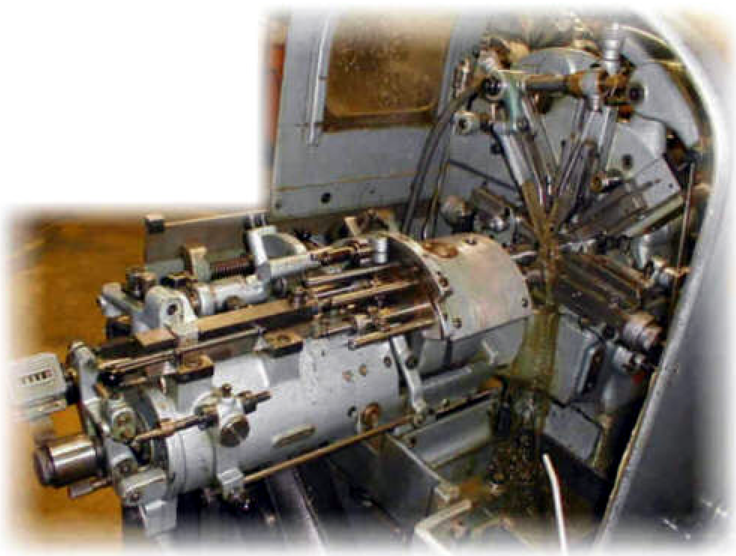
Single-spindle and multi-spindle bar machines

In these machines, instead of a chuck, a collet is used, which permits long bar stock to be fed through the headstock into position. At the end of each machining cycle, a cutoff operation separates the new part. Owing to the high level of automation, the term *automatic bar machine* is often used for these machines. Bar machines can be classified as single spindle or multiple spindle. The *single-spindle bar machine* is sometimes referred to as *swiss automatics*.



Schematics showing the principal components of a single-spindle bar machine with two cross slides, one horizontal cross slide and a hexagonal turret (cutting tools are only shown on the upper cross slides)

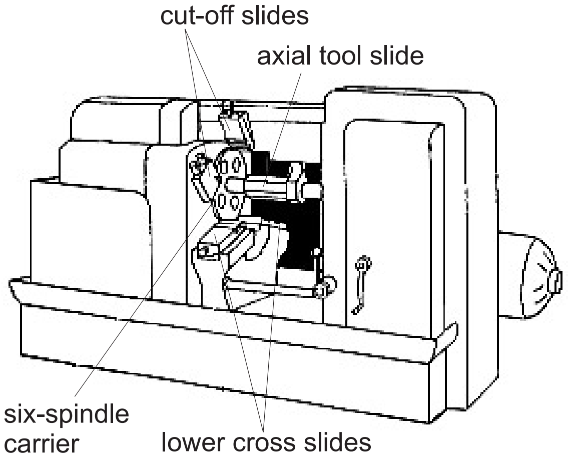
The *single-spindle bar machine* has up to six upper cross slides and two horizontal cross slides with cutting tools, which move radially inwards. All operations on the machine are controlled by appropriately shaped cams. The machine is usually equipped with three-spindle drilling/threading turret, or with a multi-position turret. More recent machines are numerically but not cam controlled.



A close-up view of a single spindle bar machine.

Typical parts produced on a single spindle bar machine.

To increase production rate, *multiple-spindle bar machines* are available. A spindle carrier in which four to eight spindles feed and rotate as many bars replaces the headstock of the lathe. A tetra-, hexa-, or octagonal axial tool slide on which tool holders are mounted replaces the tailstock. Additional tools are engaged radially, mounted on lower cross slides. So, multiple parts are machined simultaneously by multiple tools. At the end of each machining cycle, the spindles are indexed to the next set of cutting tools. A single part is completed at each indexing of the spindle carrier.

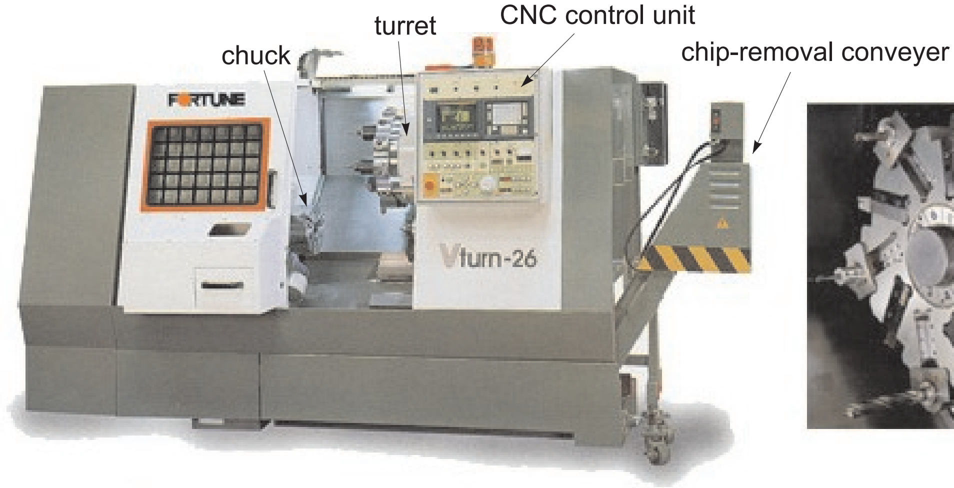


The principle components of a six-spindle bar machine

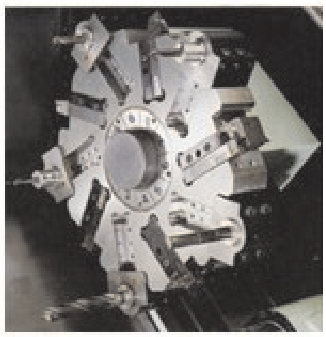
CNC controlled multiple-spindle bar machine

Computer-controlled lathes (CNC lathes)

Computer-controlled (*numerically controlled*, *NC*, *CNC*) lathes incorporate a computer system to control the movements of machine components by directly inserted coded instructions in the form of numerical data. A CNC lathe is especially useful in contour turning operations and precise machining. There are also not chuck but bar modifications. A CNC lathe is essentially a turret lathe. The major advantage of these machines is in their versatility - to adjust the CNC lathe for a different part to be machined requires a simple change in the computer program and, in some cases, a new set of cutting tools.



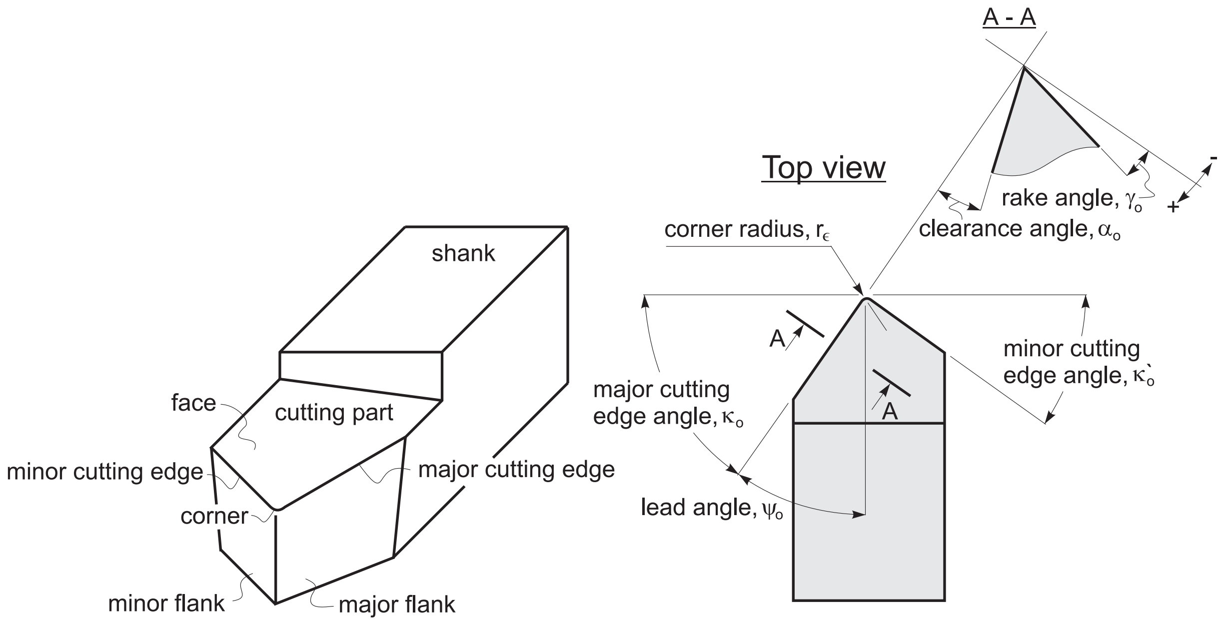
CNC chuck lathe



Ten-position turret of a CNC lathe

Cutting tools

The geometry and nomenclature of cutting tools used in turning is standardized by ISO 3002/1-1982:

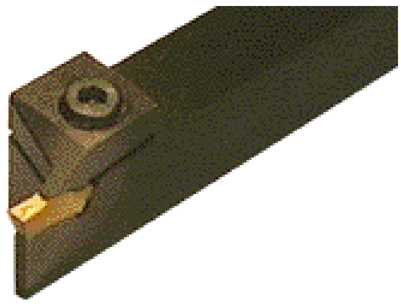


Cutting edges, surfaces and angles on the cutting part of a turning tool

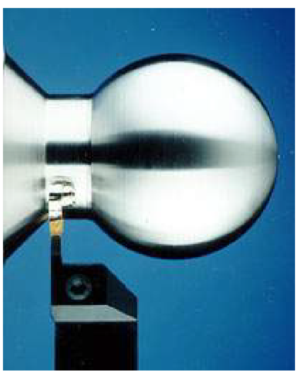
The figure shows only the most important geometrical features of a turning cutting tool. Recommendations for proper selection of the cutting tool geometry are available in the reference materials.

Cutting tool are available in different brazed or clamped designs for different operations. Some of the clamped tools are shown in the figures:

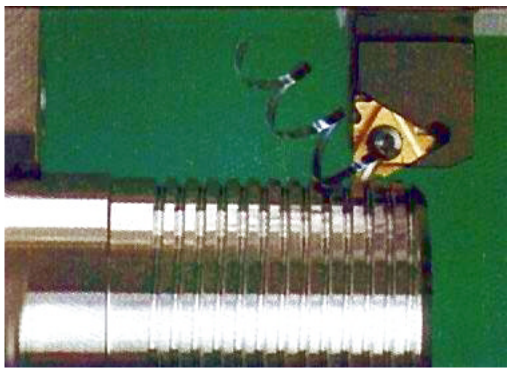
Cutting tool for straigth turning



Cutting tool for grooving



Cutting tool for profiling



Cutting tool for threading

Process capabilities and process planning in turning

The general steps when turning external workpart hold in a chuck should follow the next sequence,

❶ First rough cuts are applied on all surfaces, starting with the cylindrical surfaces (largest diameters first) and then proceeding with all faces;
❷ Special operations such as knurling and grooving (if any) are applied;
❸ Diameters are finished first, then the faces. The maximum surface finish if turning steel is $R_a \sim 1.6$ μm. If higher surface finish is required, grinding should follow machining. Grinding and other finishing operations are discussed in *Chapter 7*;
❹ External threads (if any) are cut;
❺ Deburring is applied, if necessary.

If the part is to be mounted between centres, plan should precede by,

❶ The workpiece is hold in a chuck, and the face is squired;
❷ A centre hole is drilled using a center drill (*Section 6.3*);
❸ The workpiece is reversed in the chuck. Steps ❶ and ❷ are repeated for the other face;
❹ The workpiece is mounted between centres and the general plan is followed.

If the workpart has a central hole, the hole is drilled starting with a centre drill, and increasing drill diameters gradually. Finally, boring is applied (*Section 6.5*) to achieve the final diameter of the hole. Machining of the internal features is scheduled after rough cuts and before special operations (after step ❶ in the general plan).