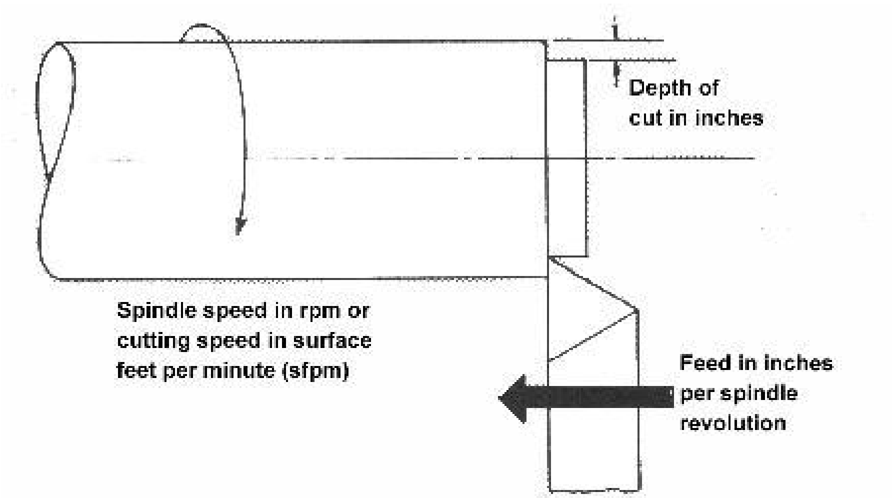
What is turning?

Turning is the machining operation that produces cylindrical parts. In its basic form, it can be defined as the machining of an external surface:

- with the workpiece rotating,
- with a single-point cutting tool, and
- with the cutting tool feeding parallel to the axis of the workpiece and at a distance that will remove the outer surface of the work.

Taper turning is practically the same, except that the cutter path is at an angle to the work axis. Similarly, in contour turning, the distance of the cutter from the work axis is varied to produce the desired shape.

Even though a single-point tool is specified, this does not exclude multiple-tool setups, which are often employed in turning. In such setups, each tool operates independently as a single-point cutter.



Turning and the adjustable parameters

Adjustable cutting factors in turning

The three primary factors in any basic turning operation are speed, feed, and depth of cut. Other factors such as kind of material and type of tool have a large influence, of course, but these three are the ones the operator can change by adjusting the controls, right at the machine.

Speed, always refers to the spindle and the workpiece. When it is stated in revolutions per minute (rpm) it tells their rotating speed. But the important figure for a particular turning operation is the surface speed, or the speed at which the workpiece material is moving past the cutting tool. It is simply the product of the rotating speed times the circumference (in feet) of the workpiece before the cut is started. It is expressed in surface feet per minute (sfpm), and it refers only to the workpiece. Every different diameter on a workpiece will have a different cutting speed, even though the rotating speed remains the same.

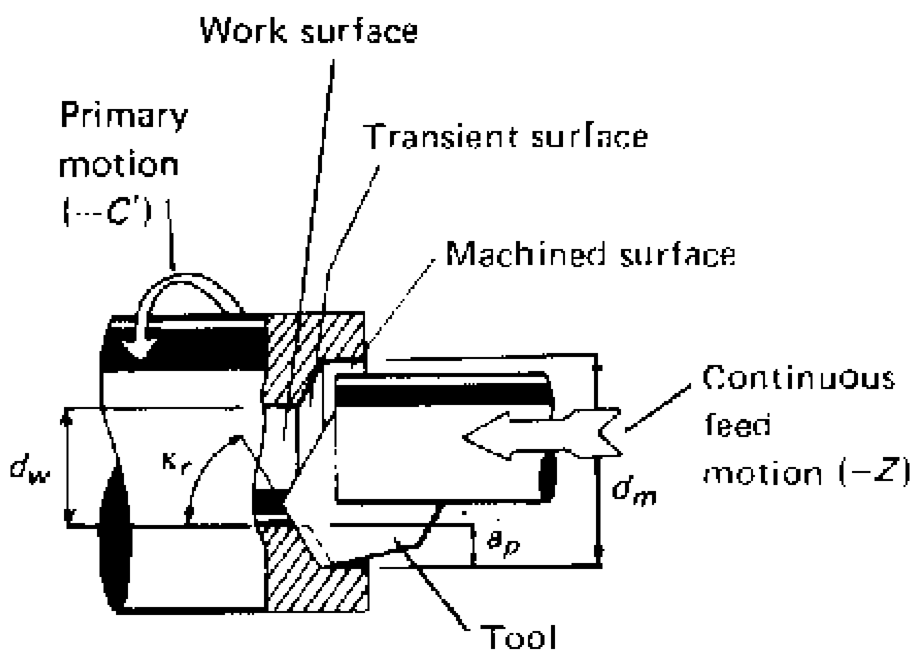
Feed, always refers to the cutting tool, and it is the rate at which the tool advances along its cutting path. On most power-fed lathes, the feed rate is directly related to the spindle speed and is expressed in inches (of tool advance) per revolution (of the spindle), or ipr. The figure, by the way, is usually much less than an inch and is shown as decimal amount.

Depth of Cut, is practically self explanatory. It is the thickness of the layer being removed from the workpiece or the distance from the uncut surface of the work to the cut surface, expressed in inches. It is important to note, though, that the diameter of the workpiece is reduced by two times the depth of cut because this layer is being removed from both sides of the work.

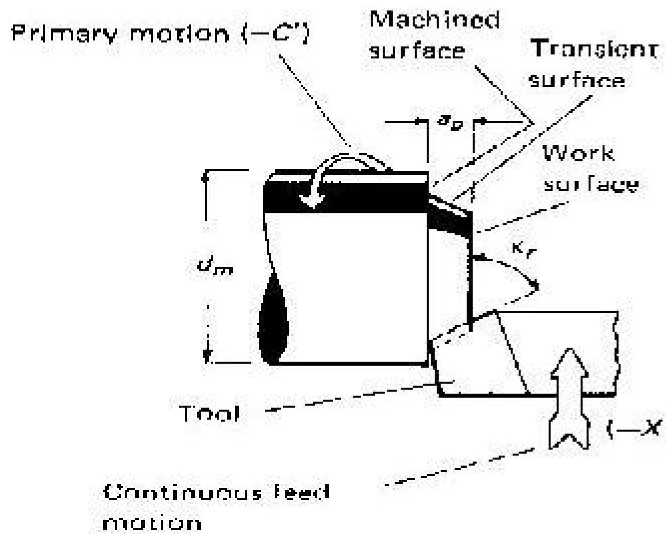
LATHE RELATED OPERATIONS

The lathe, of course, is the basic turning machine. Apart from turning, several other operations can also be performed on a lathe.

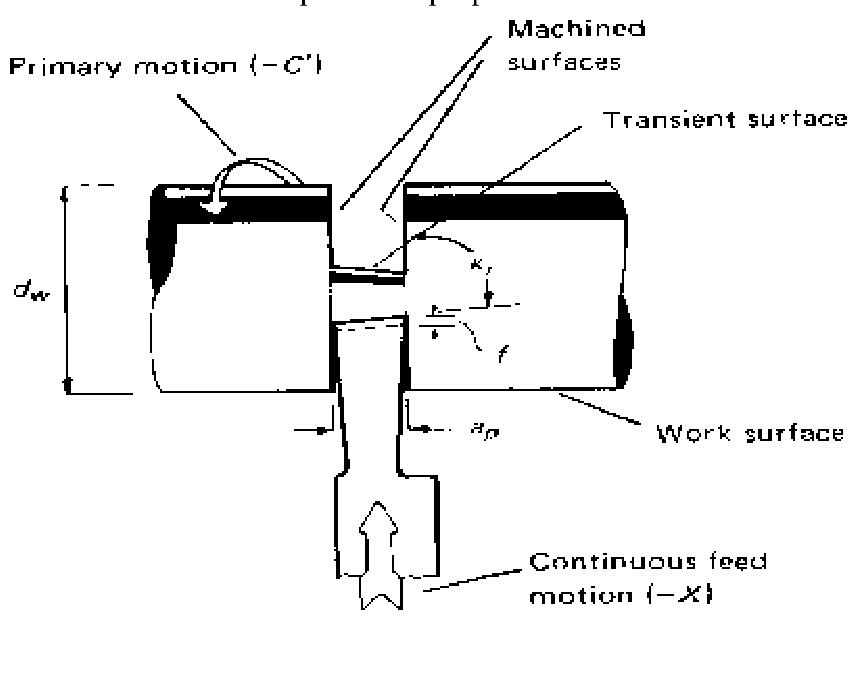
Boring. Boring always involves the enlarging of an existing hole, which may have been made by a drill or may be the result of a core in a casting. An equally important, and concurrent, purpose of boring may be to make the hole concentric with the axis of rotation of the workpiece and thus correct any eccentricity that may have resulted from the drill's having drifted off the center line. Concentricity is an important attribute of bored holes. When boring is done in a lathe, the work usually is held in a chuck or on a face plate. Holes may be bored straight, tapered, or to irregular contours. Boring is essentially internal turning while feeding the tool parallel to the rotation axis of the workpiece.



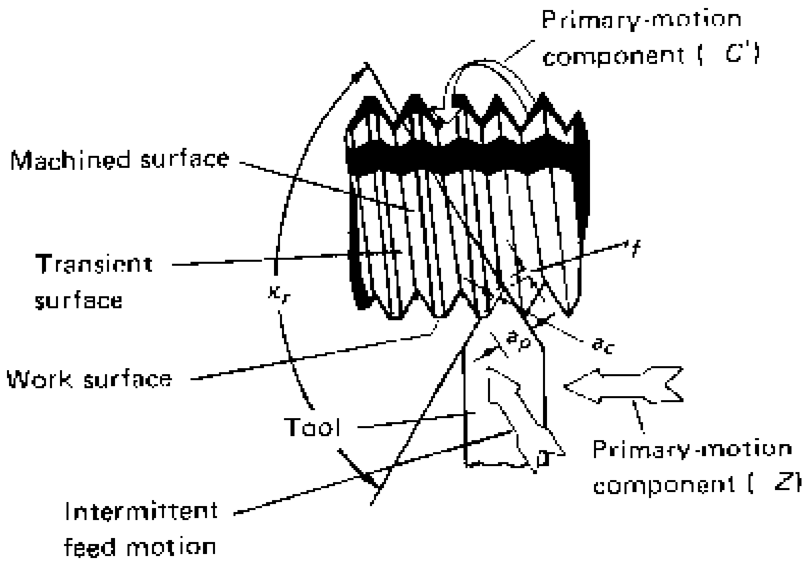
Facing. Facing is the producing of a flat surface as the result of a tool's being fed across the end of the rotating workpiece. Unless the work is held on a mandrel, if both ends of the work are to be faced, it must be turned end for end after the first end is completed and the facing operation repeated. The cutting speed should be determined from the largest diameter of the surface to be faced. Facing may be done either from the outside inward or from the center outward. In either case, the point of the tool must be set exactly at the height of the center of rotation. because the cutting force tends to push the tool away from the work, it is usually desirable to clamp the carriage to the lathe bed during each facing cut to prevent it from moving slightly and thus producing a surface that is not flat. In the facing of casting or other materials that have a hard surface, the depth of the first cut should be sufficient to penetrate the hard material to avoid excessive tool wear.



Parting. Parting is the operation by which one section of a workpiece is severed from the remainder by means of a cutoff tool. Because cutting tools are quite thin and must have considerable overhang, this process is less accurate and more difficult. The tool should be set exactly at the height of the axis of rotation, be kept sharp, have proper clearance angles, and be fed into the workpiece at a proper and uniform feed rate.

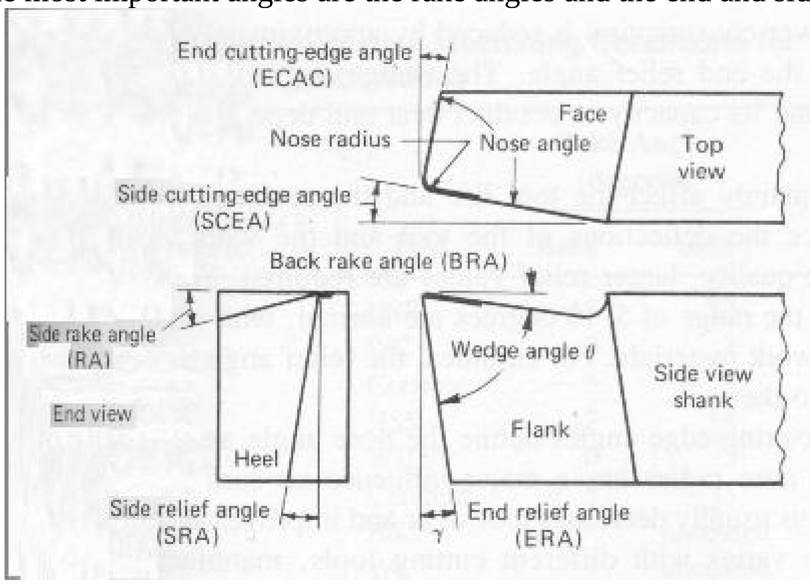


Threading. Lathe provided the first method for cutting threads by machines. Although most threads are now produced by other methods, lathes still provide the most versatile and fundamentally simple method. Consequently, they often are used for cutting threads on special workpieces where the configuration or nonstandard size does not permit them to be made by less costly methods. There are two basic requirements for thread cutting. An accurately shaped and properly mounted tool is needed because thread cutting is a form-cutting operation. The resulting thread profile is determined by the shape of the tool and its position relative to the workpiece. The second by requirement is that the tool must move longitudinally in a specific relationship to the rotation of the workpiece, because this determines the lead of the thread. This requirement is met through the use of the lead screw and the split unit, which provide positive motion of the carriage relative to the rotation of the spindle.

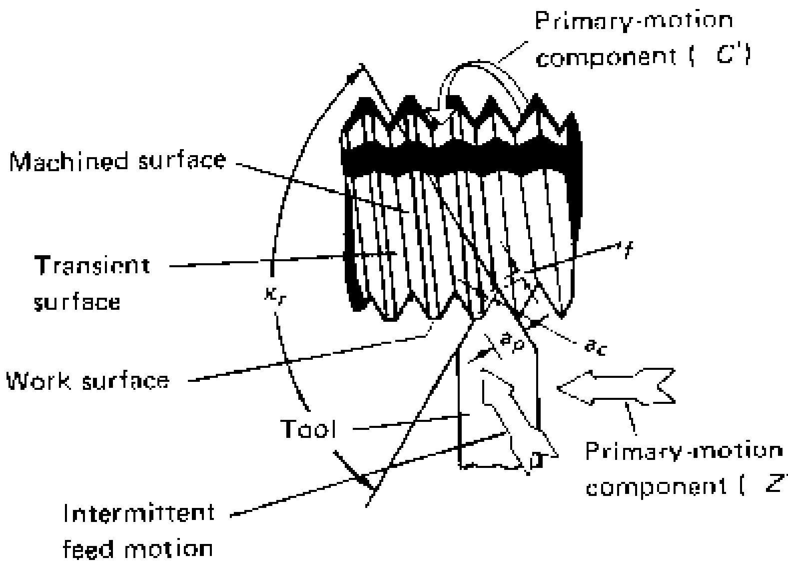


CUTTING TOOLS FOR LATHES

Tool Geometry. For cutting tools, geometry depends mainly on the properties of the tool material and the work material. The standard terminology is shown in the following figure. For single point tools, the most important angles are the rake angles and the end and side relief angles.

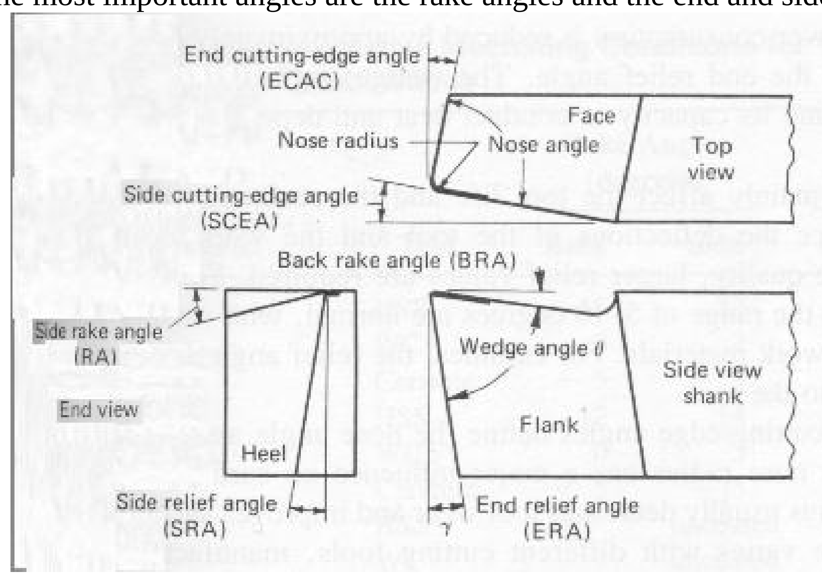


The back rake angle affects the ability of the tool to shear the work material and form the chip. It can be positive or negative. Positive rake angles reduce the cutting forces resulting in smaller deflections of the workpiece, tool holder, and machine. If the back rake angle is too large, the strength 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. There are two basic requirements for thread cutting. An accurately shaped and properly mounted tool is needed because thread cutting is a form-cutting operation. The resulting thread profile is determined by the shape of the tool and its position relative to the workpiece. The second by requirement is that the tool must move longitudinally in a specific relationship to the rotation of the workpiece, because this determines the lead of the thread. This requirement is met through the use of the lead screw and the split unit, which provide positive motion of the carriage relative to the rotation of the spindle.



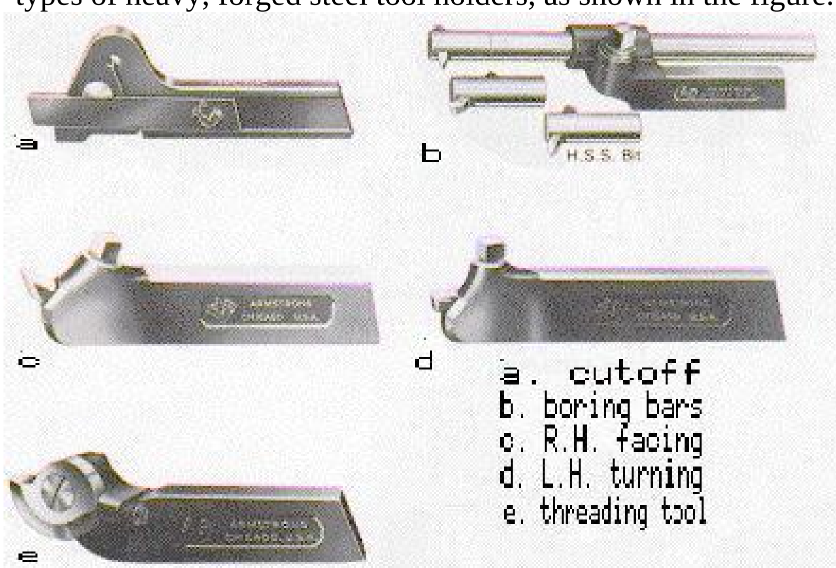
CUTTING TOOLS FOR LATHES

Tool Geometry. For cutting tools, geometry depends mainly on the properties of the tool material and the work material. The standard terminology is shown in the following figure. For single point tools, the most important angles are the rake angles and the end and side relief angles.



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 workpiece, 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. There are two basic requirements for thread cutting. An accurately shaped and properly mounted tool is needed because thread cutting is a form-cutting operation. The resulting thread profile is determined by the shape of the tool and its position relative to the workpiece. The second by requirement is that the tool must move longitudinally in a specific relationship to the rotation of the workpiece, because this determines the lead of the thread. This requirement is met through the use of the lead screw and the split unit, which provide positive motion of the carriage relative to the rotation of the spindle.

Most lathe operations are done with relatively simple, single-point cutting tools. On right-hand and left-hand turning and facing tools, the cutting takes place on the side of the tool; therefore the side rake angle is of primary importance and deep cuts can be made. On the round-nose turning tools, cutoff tools, finishing tools, and some threading tools, cutting takes place on or near the end of the tool, and the back rake is therefore of importance. Such tools are used with relatively light depths of cut. Because tool materials are expensive, it is desirable to use as little as possible. It is essential, at the same, that the cutting tool be supported in a strong, rigid manner to minimize deflection and possible vibration. Consequently, lathe tools are supported in various types of heavy, forged steel tool holders, as shown in the figure.



The tool bit should be clamped in the tool holder with minimum overhang. Otherwise, tool chatter and a poor surface finish may result. In the use of carbide, ceramic, or coated carbides for mass production work, throwaway inserts are used; these can be purchased in great variety of shapes, geometrics (nose radius, tool angle, and groove geometry), and sizes.