

4. CUTTING

Topics:

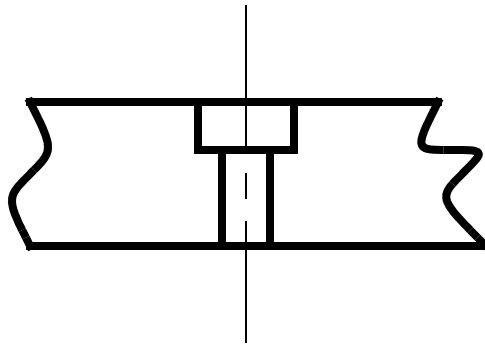
-

Objectives:

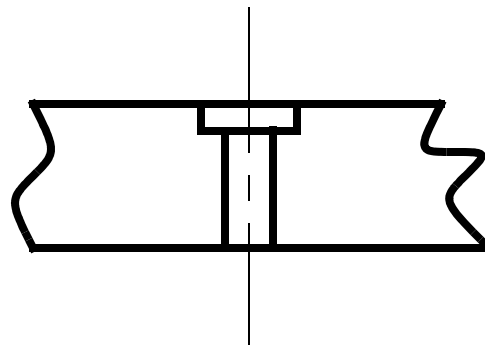
-

4.1 Drilling

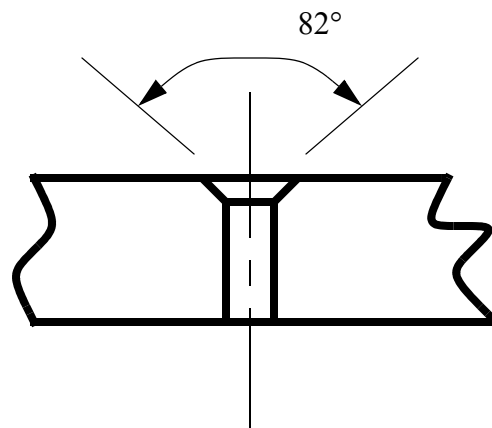
- A very common operation that cuts cylindrical holes.
- General type of drill presses in use are,
 - Sensitive - typically belt driven, and the bit is fed by hand. There are a limited choice of speeds. A bench top machine
 - Vertical or Pillar - has a heavy frame to support a wider range of work. The table height is adjustable, and power speeds and feeds are available.
 - Radial Arm - For very large and heavy work. The arm is power driven for the height location. The drilling head traverses the swinging arm. The workpiece remains stationary on the machine base, or work table. The machine spindle is moved to the location required.
- More specialized drill presses are,
 - Gang Type - several spindles/or stations are mounted on one long table
 - Multi Spindle - There are many spindles mounted on one head to allow many holes to be drilled simultaneously (e.g., up to 24)
 - Numerical Control Type - The machine can automatically change tooling with a turret or automatic tool changer. Speeds, feeds and table position are controlled using a computer program.
- Counter Bores - Allows the head of cap screws to be sunk beneath a surface



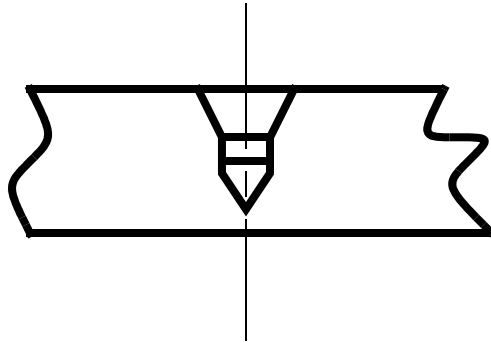
- Spot Face - Allows the head of a bolt to be sunk beneath the surface. This is basically a shallow counter bore.



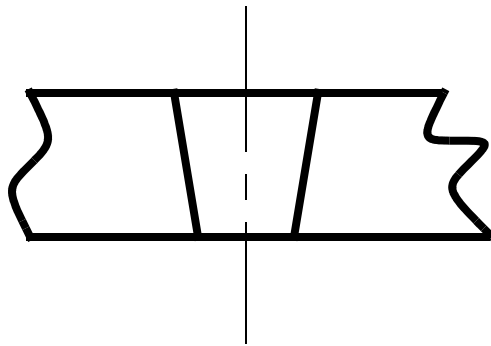
- Counter Sink - Allows counter sunk head screws to be sunk beneath a surface.



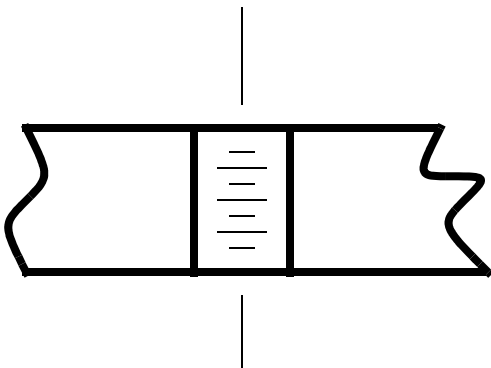
- Center Drilling - Allows parts to be mounted between centers, on lathes typically.



- Tapered Holes - these holes can be cut using reamers.



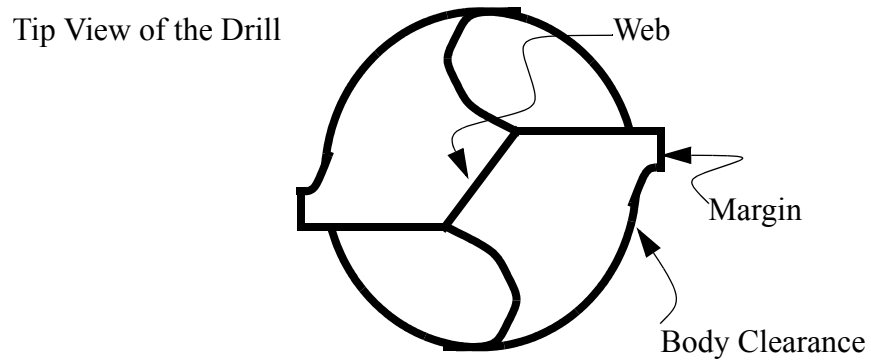
- Threaded Holes - Taps can be used to add threads to holes



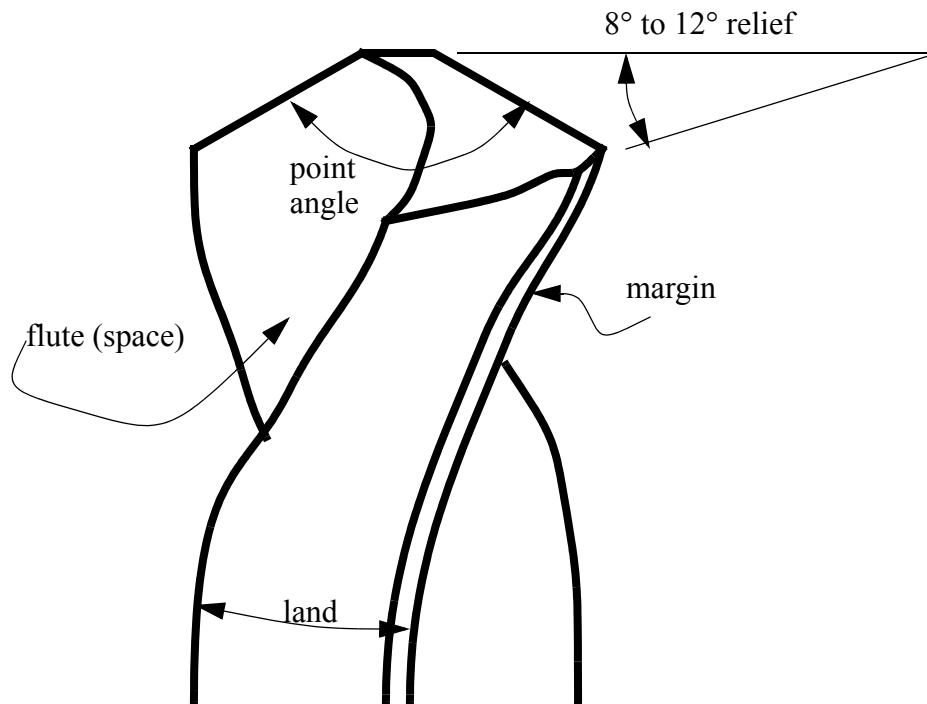
- High tolerance finishes for holes can be made with boring or reaming.

4.1.1 Drill Bits

- The twist drill does most of the cutting with the tip of the bit.
- There are flutes to carry the chips up from the cutting edges to the top of the hole where they are cast off.
- Some of the parts of a drill bit are diagramed below as viewed from the cutting tip of the drill,



- Some other features of the drill bit are shown below for a side view of the drill bit,



- Typical parameters for drill bits are,
 - Material is High Speed Steel

- Standard Point Angle is 118°

- Harder materials have higher point angles, soft materials have lower point angles.
- The helix results in a positive cutting rake.
- Drill bits are typically ground (by hand) until they are the desired shape. When done grinding, the lips should be the same length and at the same angle, otherwise an oversized hole may be produced.
- Drill sizes are typically measured across the drill points with a micrometer
- Typical drill sizes are,
 - FRACTIONAL - 1/64" to 3 1/4" dia. in 1/64" steps
 - NUMBER - #1 = 0.228" dia. to #80 = 0.0135" dia.
 - LETTER - A = 0.234" dia. to Z = 0.413" dia.
 - METRIC - 0.4mm dia. to 50mm dia.

DRILL #	dia. (in.)	DRILL #	dia. (in.)	DRILL #	dia. (in.)	DRILL #	dia. (in.)
1	0.2280	25	0.1495	49	0.0730	73	0.0240
2	0.2210	26	0.1470	50	0.0700	74	0.0225
3	0.2130	27	0.1440	51	0.0670	75	0.0210
4	0.2090	28	0.1405	52	0.0635	76	0.0200
5	0.2055	29	0.1360	53	0.0595	77	0.0180
6	0.2040	30	0.1285	54	0.0550	78	0.0160
7	0.2010	31	0.1200	55	0.0520	79	0.0145
8	0.1990	32	0.1160	56	0.0465	80	0.0135
9	0.1960	33	0.1130	57	0.0430	81	0.0130
10	0.1935	34	0.1110	58	0.0420	82	0.0125
11	0.1910	35	0.1100	59	0.0410	83	0.0120
12	0.1890	36	0.1065	60	0.0400	84	0.0115
13	0.1850	37	0.1040	61	0.0390	85	0.0110
14	0.1820	38	0.1015	62	0.0380	86	0.0105
15	0.1800	39	0.0995	63	0.0370	87	0.0100
16	0.1770	40	0.0980	64	0.0360	88	0.0095
17	0.1730	41	0.0960	65	0.0350	89	0.0091
18	0.1695	42	0.0935	66	0.0330	90	0.0087
19	0.1660	43	0.0900	67	0.0320	91	0.0083
20	0.1610	44	0.0860	68	0.0310	92	0.0079
21	0.1590	45	0.0820	69	0.0292	93	0.0075
22	0.1570	46	0.0810	70	0.0280	94	0.0071
23	0.1540	47	0.0810	71	0.0260	95	0.0067
24	0.1520	48	0.0785	72	0.0250	96	0.0063
						97	0.0059

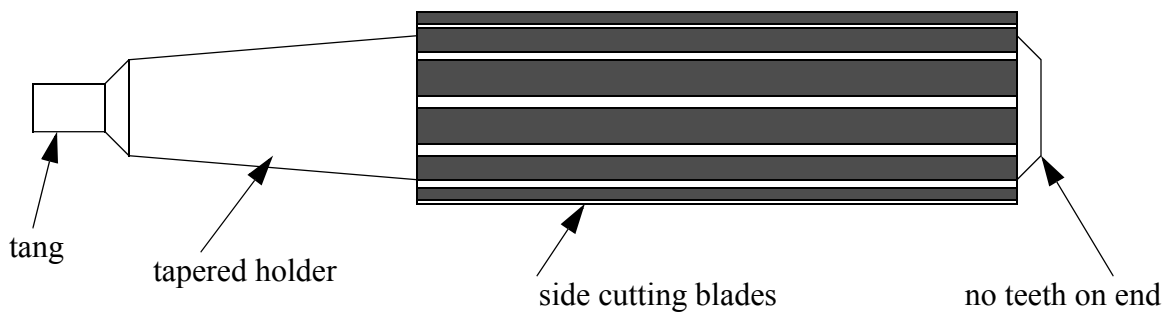
Letter size	dia. (in.)	Letter size	dia. (in.)
A	0.234	N	0.302
B	0.238	O	0.316
C	0.242	P	0.323
D	0.246	Q	0.332
E	0.250	R	0.339
F	0.257	S	0.348
G	0.261	T	0.358
H	0.266	U	0.368
I	0.272	V	0.377
J	0.277	W	0.386
K	0.281	X	0.397
L	0.290	Y	0.404
M	0.295	Z	0.413

- Some standard drill types are,
 - Straight Shank - this type is held in a chuck
 - Taper shank - this type is held in a sleeve, and a machine spindle. A drift may also be used.
- Some other types of drills used are,
 - Core drills - a drill with a small helix, and 3 or 4 flutes. This is used for light drilling, such as opening holes in castings.
 - High helix - When drilling a deep hole in a soft material these drills are used to help remove chips
 - Straight fluted - Used to drill soft metals and plastics. The straight flutes prevent the bit from digging in.
 - Centre drills - A drill with a small entry tip, and a widening profile. The result is a hole that has a conical shape on the outside, that may be used to mount the part between centres, or to act as a guide for a larger drill.
- Typically an allowance of a third of the drill bit diameter is given for the tip of the drill.
- Center Drill Sizes [Krar],

Regular Size	Work Dia. (in.)	Countersink dia. (in.)	Drill point dia. (in.)	Body Size (in.)
1	3/16-5/16	3/32	3/64	1/8
2	3/8-1/2	9/64	5/64	3/16
3	5/8-3/4	3/16	7/64	1/4
4	1-1.5	15/64	1/8	5/16
5	2-3	21/64	3/16	7/16
6	3-4	3/8	7/32	1/2
7	4-5	15/32	1/4	5/8
8	over 6	9/16	5/16	3/4

4.1.2 Reamers

- Reamers are a special class of drill. They are used after a hole has been drilled near to final size. The reamers is then used to remove a small quantity of material, and finish the hole with a good surface texture, roundness, and alignment.



- These are often used to provide holes for bearings, parallel and taper dowels, and various fits with a shaft.
- These are typically made of High Speed Steel, or with carbide tips.
- The main body contains many straight and helical flutes. The tip does not contain any cutting edges.
- Various types are,
 - Parallel Reamer - Straight fluted reamer held in a drill press spindle with a tapered shank.

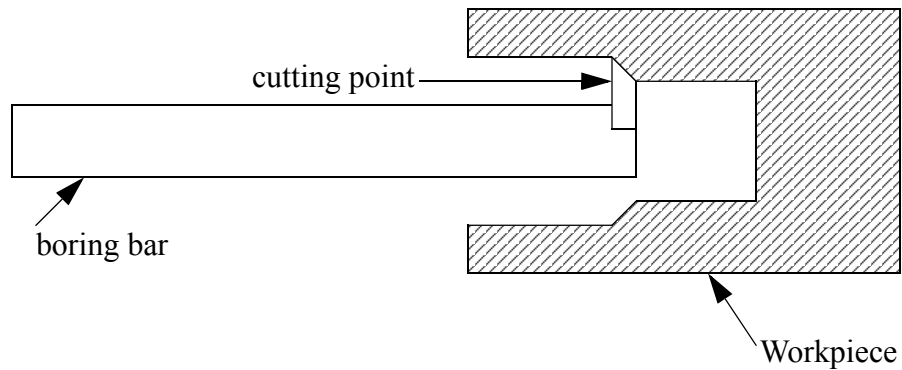
- Parallel Hand - Straight flutes, but held in a hand tap wrench.
- Taper Reamers - has a taper from one end to the other. These can be used in a spindle (tapered shank), or by hand (for a taper wrench).
- Adjustable Reamer - This uses inserted blades.

- Recommended allowances and speeds for reaming [Krar],

Hole Size (in.)	Allowance (in.)	Material	Speed (ft/min)
1/4	0.010	Aluminum	130-200
1/2	0.015	Brass	130-180
3/4	0.018	Bronze	50-100
1	0.020	Cast Iron	50-80
1.25	0.022	Machine Steel	50-70
1.5	0.025	Steel Alloys	30-40
2	0.030	Stainless Steel	40-50
3	0.045	Magnesium	170-270

4.1.3 Boring

- Boring is used for high quality finished.



- In boring the tool can be rotated, or the work can be rotated.

4.1.4 Taps

- Taps can use for both internal and external threads.
- A typical set of hand taps consists of
 - #1 Taper
 - #2 Plug
 - #3 Bottoming
- There are flutes in the taps to help remove chips, to provide cutting edges, and channels for lubrication.
- There are a number of sets of threads available,
 - UNC (Unified National Course)
 - UNF (Unified National Fine)
 - ACME
 - Metric
- To create one of these holes, we must first drill a hole that is slightly smaller. For example,

5/8 - 11 - UNC



outside diameter = $5/8 = 0.625''$
11 threads per inch (T.P.I.)
Unified National Coarse is the tooth profile

The tap drill size is Outside Diameter - $1/T.P.I.$ for UNC, UNF, Metric threads.

Therefore, the Tap Drill Size (TDS) is,

$$T.D.S. = 0.625'' - 1/11'' = 17/32''$$

- Some setups associated with taps are,
 - alignment of the tap in a drill press
 - use of taping attachments
- NF/NC Thread Tap Drill Sizes [Krar],

National Coarse (NC)			National Fine (NF)		
Tap Size	TPI	Tap Drill Size	Tap Size	TPI	Tap Drill Size
#5	40	#38	#5	44	#37
#6	32	#36	#6	40	#33
#8	32	#29	#8	36	#29
#10	24	#25	#10	32	#21
#12	24	#16	#12	28	#14
1/4	20	#7	1/4	28	#3
5/16	18	F	5/16	24	1
3/8	16	5/16	3/8	24	Q
7/16	14	U	7/16	20	25/64
1/2	13	27/64	1/2	20	29/64
9/16	12	31/64	9/16	18	33/64
5/8	11	17/32	5/8	18	37/64
3/4	10	21/32	3/4	16	11/16
7/8	9	49/64	7/8	14	13/16
1	8	7/8	1	14	15/16
1-1/8	7	63/64	1-1/8	12	1-3/64
1-1/4	7	1-7/64	1-1/4	12	1-11/64
1-3/8	6	1-7/32	1-3/8	12	1-19/64
1-1/2	6	1-11/32	1-1/2	12	1-27/64
1-3/4	5	1-9/16			
2	4-1/2	1-25/32			

4.1.5 Process Parameters

- The parameters for drilling are found in almost the same way as for lathes,

$$CS = rpm \times C$$

$$C = \frac{\pi D}{12} = \frac{\pi D}{1000}$$

$$rpm = \frac{12 \times CS}{\pi \times D} = \frac{1000 \times CS}{\pi \times D}$$

imperial

metric

where,

CS = cutting speed (fpm or m/s) - can be selected from tables

rpm = revolutions per minute of the machine spindle

C = circumference of the drill bit (ft. or m)

D = diameter of drill bit (in. or mm)

$$T = \frac{L}{F} = \frac{L}{rpm \times F}$$

$$C = T \times R$$

where,

L = length of cut (in. or mm)

F = feed rate (in./rev. or mm/rev.) - found in tables

R = Machine cost (\$/min.)

- Typical high speed drill speeds are, [Krar]

Drill dia. (in.)	steel casting 40 fpm	tool steel 60 fpm	cast iron 80 fpm	machine steel 100 fpm	brass/aluminum 200 fpm
1/16	2445	3665	4890	6110	12225
1/8	1220	1835	2445	3055	6110
3/16	815	1220	1630	2035	4075
1/4	610	915	1220	1530	3055
5/16	490	735	980	1220	2445
3/8	405	610	815	1020	2035
7/16	350	525	700	875	1745
1/2	305	460	610	765	1530
5/8	245	365	490	610	1220
3/4	205	305	405	510	1020
7/8	175	260	350	435	875
1	155	230	305	380	765

- Consider also the typical feeds for drilling, [Krar]

Drill dia. (in.)	Feed per Rev. (in.)
1/8 or less	0.001 to 0.002
1/8 to 1/4	0.002 to 0.004
1/4 to 1/3	0.004 to 0.007
1/2 to 1	0.007 to 0.015
1 to 1.5	0.015 to 0.025

4.1.6 The mrr For Drilling

- considering the parameters defined in the discussion of speeds and feeds, etc, the mrr is given below,

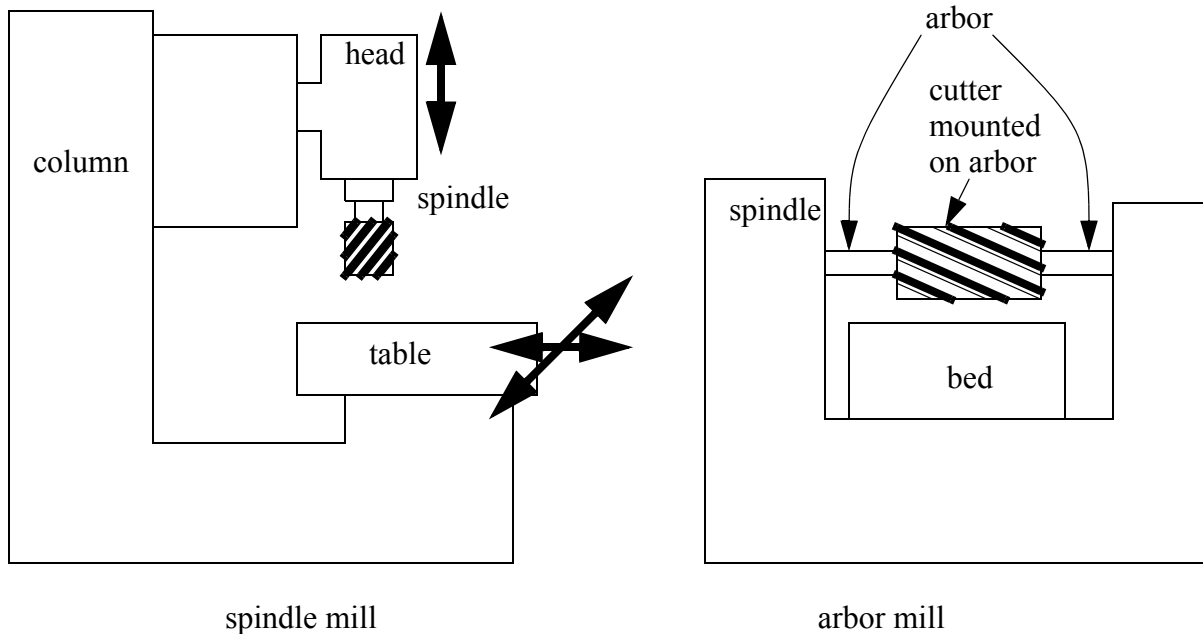
$$mrr = A \times F \times rpm = \frac{\pi D^2}{4} \times F \times rpm$$

where,

A = cutting area of the drill bit (a cross section)

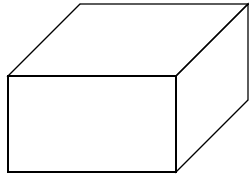
4.2 Milling

- Milling machines typically have a rotating cutting tool mounted in a spindle. The work is mounted on a bed, and then either the spindle, or bed is moved. Cutting is done with different parts of the milling tool, as will be described later.
- Some basic types of milling machines include,
 - Knee and Column
 - vertical
 - horizontal
 - horizontal with vertical head attachment
 - universal (table rotates in plan view) and is used for helical milling
 - Ram & Turret - Light weight machine tool with slotter on one end of turret. No power feeds.
 - Special Purpose - For production usage. Usually more rigid construction.

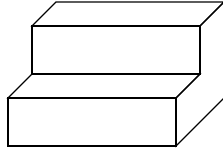


4.2.1 Types of Milling Operations

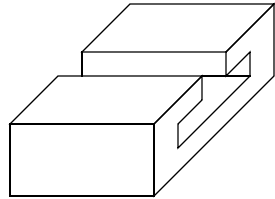
- Typical operations re pictured below



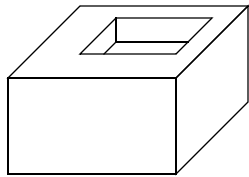
Face - cut a face flat



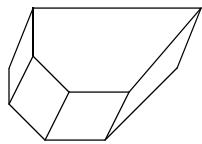
Step



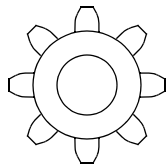
Slots



Pockets/contours



Angles



Gear Teeth

4.2.1.1 - Arbor Milling

- The advantages of arbor milling are,
 - The cutter is held more rigidly on the spindle nose
 - There is less variation in the arbor torque
 - The teeth responsible for surface finish do not encounter the hard mill scale
 - Lower power requirements
 - Flatter work surface finish.

- For straddle milling
 - Two similar side and face cutters are mounted on the same arbor, with spacers to separate them.
 - This allows two sides of a part to be cut in a single pass.

- For Gang milling
 - Many dissimilar cutters are mounted on the same arbor at the same time.
 - When the work is passed under the cutter, multiple cuts are made in a single pass, reducing alignment problems, and decreasing operation time.

4.2.2 Milling Cutters

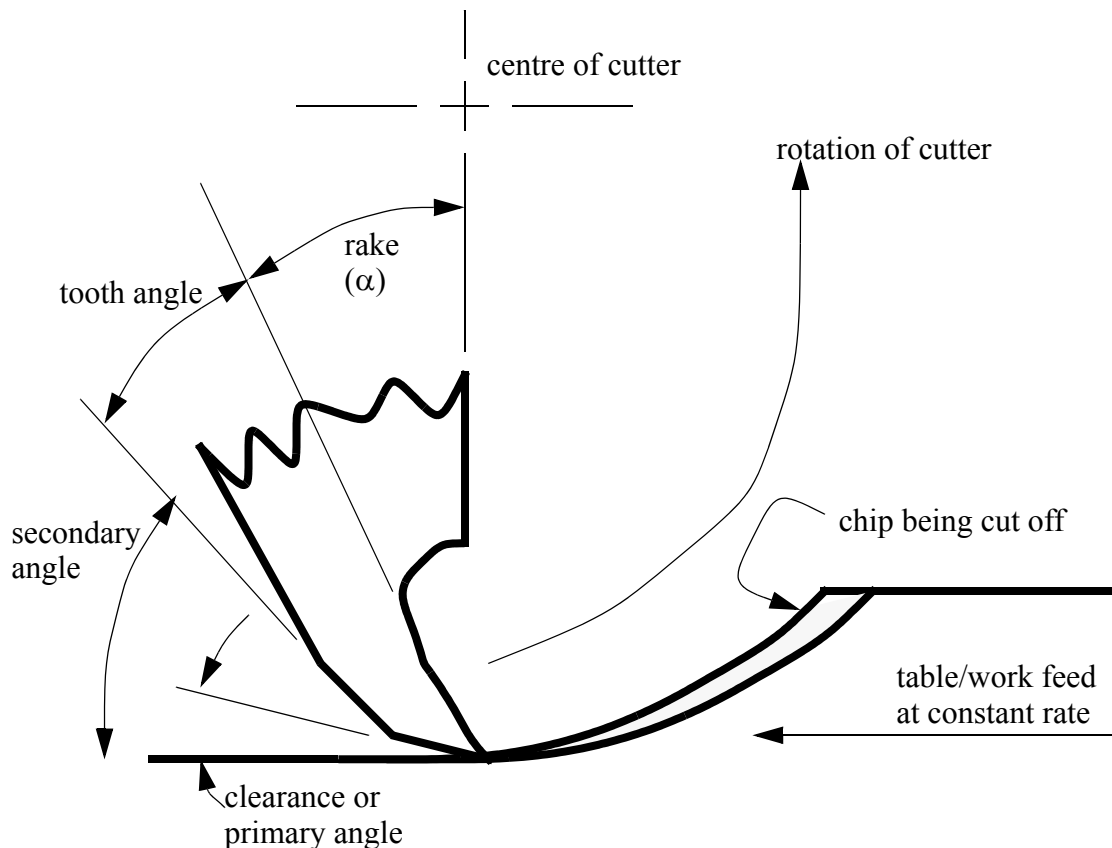
- The family milling cutters include a number of basic operations, but in general they will cut with some combination of the end and/or the sides.

- The basic types include,
 - End Mills - The face and sides at the bottom end of this tool are used for plunge cutting (two flutes) and side and end cuts (four flute).
 - Plain - These mills are used to cut with the sides only. They are generally mounted on an arbor.
 - Side or Side & Face -
 - Face - This cutter is held on a spindle nose.
 - Shell and adapter -
 - Form -
 - T-Slot, Dovetail, Woodruff -
 - Slitting Saws -

4.2.3 Milling Cutting Mechanism

- In milling each tooth on a tool removes part of the stock in the form of a chip.

- There are two types of cutting actions,
 Peripheral - The teeth at the periphery do the cutting
 Face - The teeth on the face of the cutter remove metal.
- The basic interface between tool and work is pictured below. This shows a peripheral milling tooth.



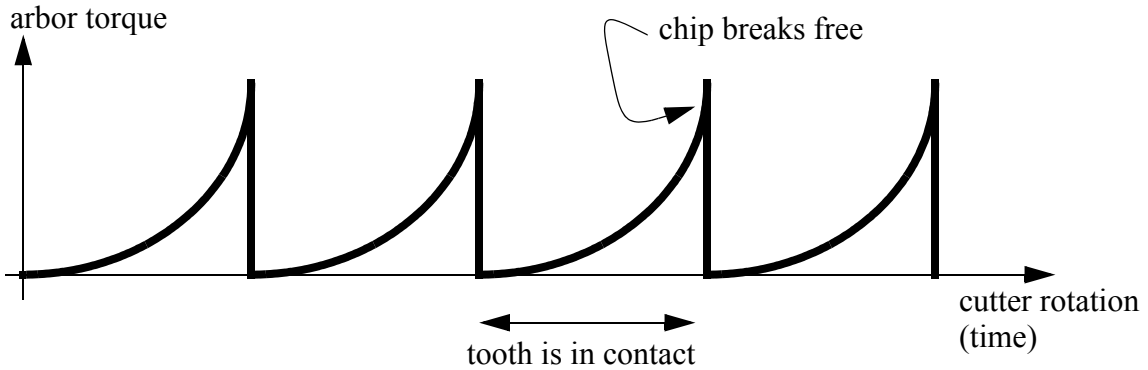
4.2.3.1 - Up-Cut Milling

- The milling method shown above is called up-cut (or conventional) milling. In this case the table is moving towards the cutter, opposing the cutter direction. The basic steps of chip cutting here are,
 1. As the tooth makes contact with the surface, the tooth begins to push down. As the tooth continues to turn, it reaches a point at which the pressure has built up to a high level, and the tooth begin to dig in.
 2. As the tooth starts to dig, it cuts down, and the metal chip begins to shear off.
 3. The tooth continues to cut the chip off, until it reaches the surface of the material. At this point the chip breaks free, and the cutting forces drop to zero.
- Because the cutter does not start to cut when it makes contact, and because the advance moves

high points past the cutter contact, the surface has a natural waviness.



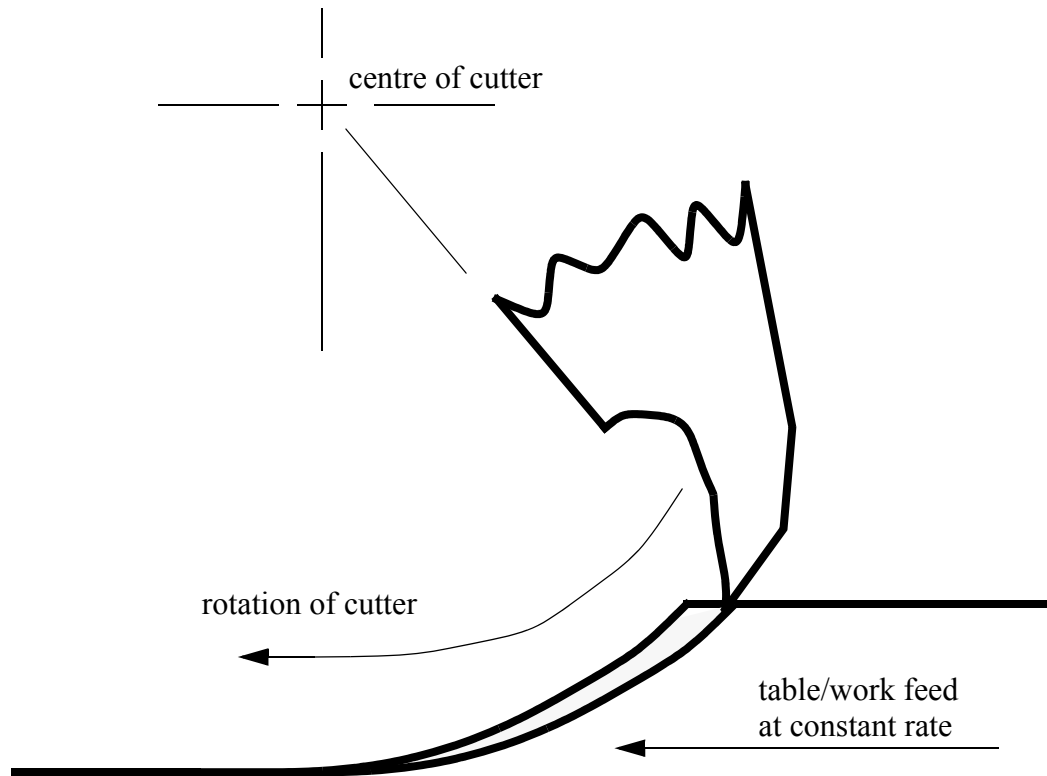
- If a cutter has straight flutes, then a torque profile for it might look like,



- The peak arbor torque can be smoothed out by using helical cutting blades, so that there is always a cutter in contact at any one time.

4.2.3.2 - Down-Cut Milling

- When the cutter rotation is in the same direction as the motion of the work being fed, it is referred to a Down-cut, or climb milling.



- When this cutter makes contact with the work, it must begin cutting at the maximum torque. As a result, a back-lash eliminator must be used to take play out of the system.
- This method has advantages,
 - The cutter forces are directed into the table, which reduces fixture forces, and allows thinner workpieces
 - There is less radial pressure on the arbor
 - Better surface finishes obtained because there is no “dig-in”

4.3 Feeds and Speeds

- Milling is somewhat different than drilling and turning,

$$CS = rpm \times C$$

$$C = \frac{\pi D}{12} = \frac{\pi D}{1000}$$

$$rpm = \frac{12 \times CS}{\pi \times D} = \frac{1000 \times CS}{\pi \times D}$$

imperial

metric

where,

CS = cutting speed (fpm or m/s) - can be selected from tables

rpm = revolutions per minute of the machine spindle

C = circumference of the cutter (ft. or m)

D = diameter of the cutter (in. or mm)

$$F = fpt \times \#t \times rpm$$

where,

F = feed rate (in./min.) - this is independent of the spindle rpm

fpt = feed per tooth - found in tables

#t = number of teeth on a particular tool

$$T = \frac{L}{F} = \frac{L}{rpm \times F}$$

$$C = T \times R$$

where,

L = length of cut (in. or mm)

R = Machine cost (\$/min.)

- Typical speeds are, [Krar]

Work Material	HSS tool (fpm)	carbide tool (fpm)
machine steel	70-100	150-250
tool steel	60-70	125-200
cast iron	50-80	125-200
bronze	65-120	200-400
aluminum	500-1000	1000-2000

- Typical feed per tooth values for HSS cutters, [Krar]

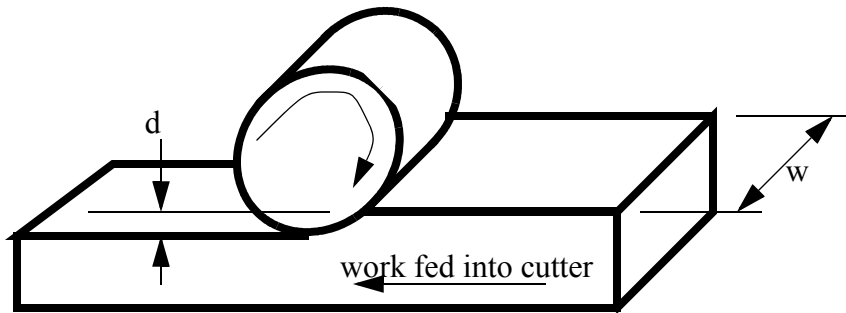
Material	face mill (in.)	helical mill (in.)	slot/side mill (in.)	end mill (in.)	form cut (in.)	circular saws (in.)
aluminum	0.022	0.018	0.013	0.011	0.007	0.005
brass/bronze (medium)	0.014	0.011	0.008	0.007	0.004	0.003
cast iron (medium)	0.013	0.010	0.007	0.007	0.004	0.003
machine steel	0.012	0.010	0.007	0.006	0.004	0.003
tool steel (medium)	0.010	0.008	0.006	0.005	0.003	0.003
stainless steel	0.006	0.005	0.004	0.003	0.002	0.002

- Typical feed per tooth values for cemented carbide (tipped) cutters, [Krar]

Material	face mill (in.)	helical mill (in.)	slot/side mill (in.)	end mill (in.)	form cut (in.)	circular saws (in.)
aluminum	0.020	0.016	0.012	0.010	0.006	0.005
brass/bronze (medium)	0.012	0.010	0.007	0.006	0.004	0.003
cast iron (medium)	0.016	0.013	0.010	0.008	0.005	0.004
machine steel	0.016	0.013	0.009	0.008	0.005	0.004
tool steel (medium)	0.014	0.011	0.008	0.007	0.004	0.004
stainless steel	0.010	0.008	0.006	0.005	0.003	0.003

4.3.1 The mrr for Milling

- considering the parameters defined in the discussion of speeds and feeds, etc, the mrr is given below,



$$mrr = w \times d \times F$$

where,

w = width of cut

d = depth of cut

4.3.2 Process Planning for Prismatic Parts

- The basic steps are,
 1. Cut off the stock slightly larger than required.
 2. Cut the basic outside diameter to size using a milling machine.
 3. Lay out the basic features of the parts (in manual setups, this involves coating the surface with a blue stain, this is then cut and marked).
 4. Use a bandsaw to rough cut the work.
 5. On the mill, cut steps, radii, angles, grooves, etc.
 6. Lay out the holes to be drilled, and then drill them.
 7. Ream holes as required
 8. Grind any surfaces that require it. Ground surfaces should generally have 0.010"

4.3.3 Indexing

- It may sometimes become necessary to rotate parts on a milling machine, beyond the rotation

offered in some beds (e.g. Universal Milling Machine).

- Some of the applications that require this capability are milling of,
 - polygons,
 - splines
 - gears,
 - cams
 - spirals
- This method can be done with a dividing head. This is basically a worm gear unit. As the crank is turned, the cylindrical gear will drive the round gear. This will result in an apparatus that takes large motions in the crank, and results in small rotations of the work. When coupled with a scale of some description this becomes very accurate.
- If a worm wheel has 40 teeth, each rotation of the crank will result in a rotation of $40/360$ degrees, or $1/40$ th of a rotation. This means the rotation is 40:1.

***** INCLUDE FIGURES OF INDEXING HEAD

- There are two methods of indexing,
 - Direct Indexing - A notched plate is located so that the crank shaft can be fixed at set positions (notches).
 - Simple Indexing - Work is rotated by turning a crank. The crank is finally positioned using a plate with holes, and a sector arm. (The sector arm is used to count off the divisions on the plates)
- An example of the calculations involved is,

Say that we want to mill a polygon on 11 sides (i.e., 1/11th of a circle).

First, we will assume that we have a worm ratio of 40:1, and that we are using a Brown and Sharp #2 plate.

Next, we will calculate the fraction of the indexed plate to be covered,

$$INDEX = \left(\frac{40}{1}\right)\left(\frac{1}{11}\right) = \frac{40}{11} = 3\frac{7}{11}$$

So, we must turn the crank handle 3 times, plus a bit more. Next we must determine which ring of index holes to use, and how many to count ahead by.

We can do this by looking at the remainder (7/11) and taking the denominator (11). Next we look at the counts available for the Brown and Sharp #2 plate (i.e., 21, 23, 27, 29, 31, 33), and from this we will notice that 33 is a multiple of 11. Therefore we can compute the number of divisions required with,

$$holes = 33\left(\frac{7}{11}\right) = 21$$

Therefore in total, we must advance the crank 3 full rotations, and 21 holes (in the ring of 33) to rotate 1/11th of a circle.

- Another example of indexing considers a rotation of 50 degrees,

First we will calculate the total indexing required,

$$INDEX = \left(\frac{40}{1}\right)\left(\frac{50}{360}\right) = \frac{2000}{360} = \frac{50}{9} = 5\frac{5}{9}$$

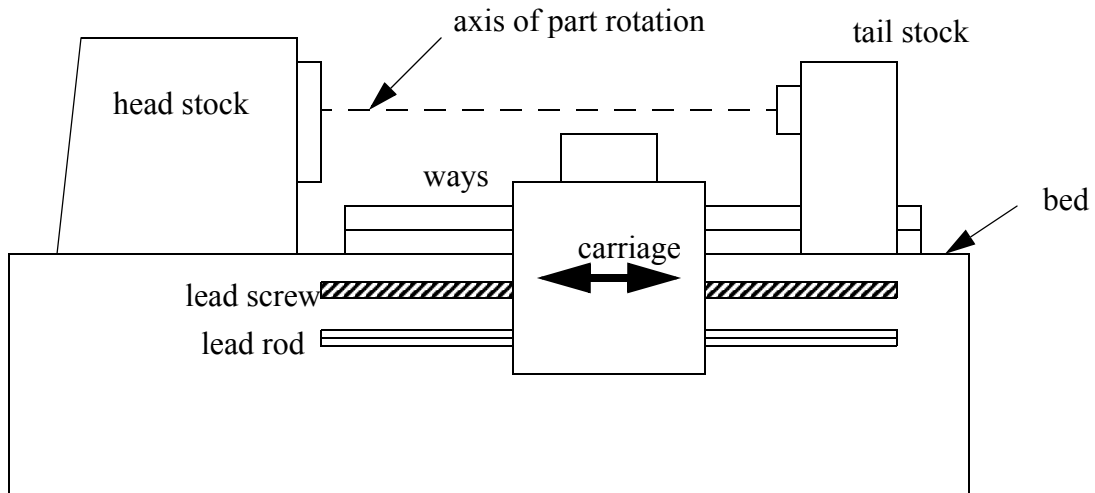
Therefore there are 5 full rotations of the indexing wheel required. Next we look at the list of indexing plates. Assume we are using the Cincinnati Standards Plates, we should look for the ring that has lowest number of index holes and is a multiple of 9. This would be 54 on the other side. Therefore we would advance the sector arm by,

$$holes = 54\left(\frac{5}{9}\right) = 30$$

- Differential indexing - is sometimes required to move plates both forward and backward part of a turn to obtain correct spacing. i.e., output shaft through gear train drives the index plate.
XXXXXXXXXXXXXXXXXXXX
- Helical milling - the machine table is rotated through a helix angle. The machine lead screw drives the dividing head. Work is rotated while the machine table feeds.
XXXXXXXXXXXXXXXXXXXX
- CAM Milling - requires a milling machine with a rotating vertical head. The dividing head is driven by the machine lead screw.

4.4 Lathes

- Cutting is performed in lathes by rotating the workpiece, and then holding a relatively stationary tool against it. Where the tool touches, the work is cut down in round patterns.
- A lathe is a large machine that rotates the work, and cutting is done with a non-rotating cutting tool. The shapes cut are generally round, or helical. The tool is typically moved parallel to the axis of rotation during cutting.
- Manual lathes have the following major components,



head stock - this end of the lathe contains the driving motor and gears. Power to rotate the part is delivered from here. This typically has levers that let the speeds and feeds be set.

ways - these are hardened rails that the carriage rides on.

tail stock - this can be used to hold the other end of the part.

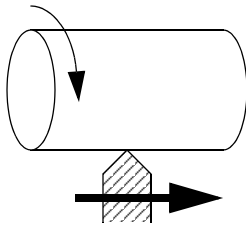
bed - this is a bottom pan on the lathe that catches chips, cutting fluids, etc.

carriage - this part of the lathe carries the cutting tool and moves based on the rotation of the lead screw or rod.

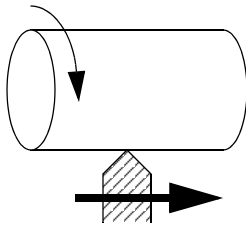
lead screw - a large screw with a few threads per inch used for cutting threads

lead rod - a rod with a shaft down the side used for driving normal cutting feeds.

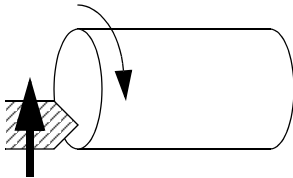
- General classifications used when describing lathes are,
 - Swing - the largest diameter of work that can be rotated.
 - Distance Between Centres - the longest length of workpiece
 - Length of Bed - Related to the Distance Between Centres
 - Power - The range of speeds and feeds, and the horsepower available
- The critical parameters on the lathe are speed of rotation (speed in RPM) and how far the tool moves across the work for each rotation (feed in IPR).
- Operations on a lathe include,



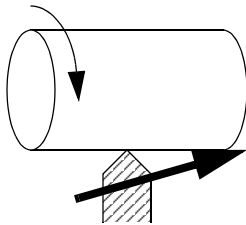
Turning - produces a smooth and straight outside radius on a part.



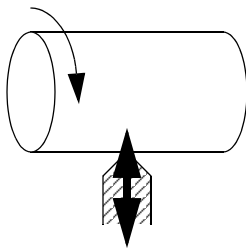
Threading - The cutting tool is moved quickly cutting threads.



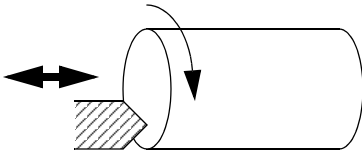
Facing - The end of the part is turned to be square.



Tapering - the tool is moves so as to cut a taper (cone shape).



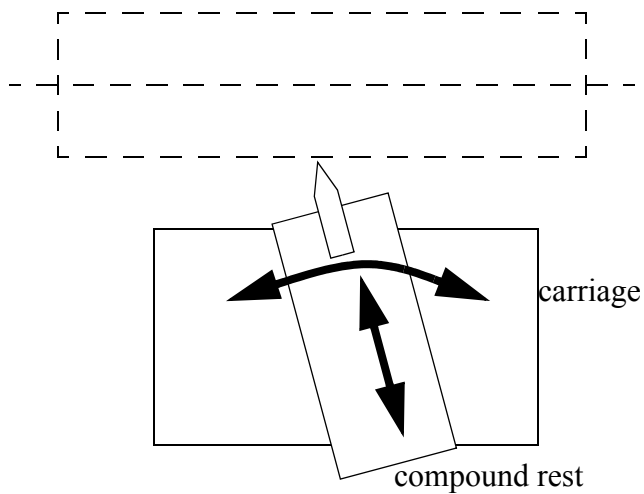
Parting/Slotting/Grooving - A tool is moved in/out of the work. shallow cut will leave a formed cut, a deep cut will cut off the unsupported part.



Drilling/Boring - a cutter or drill bit is pushed into the end to create an internal feature.

4.4.1 Machine tools

- There are two tool feed mechanism on most lathes. These cause the cutting tool to move when engaged.
 - The larger screw (the lead screw) will cause the lathe cutter to advance quickly. This is used for cutting screws, and for moving the tool quickly. Typical feed rates range from about 0.05" to 0.5" per revolution.
 - The small screw (the feed rod) will move the cutter slowly forward. This is largely used when doing rough cuts, or finishing operations. Typical feeds with this screw range from 0.001" to 0.010" per revolution.
- On a lathe the axial distance of the tool on the part is set by the carriage. A compound rest is used on a lathe that allows the radial tool position and orientation of the cutting edges.

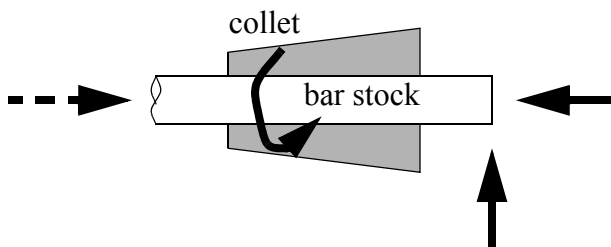


- Work is held in the lathe with a number of methods.

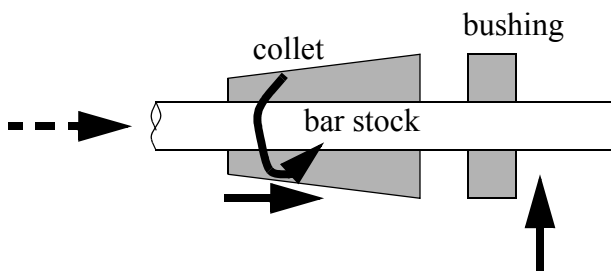
- 3 jaw self centering chuck
- 4 jaw independently adjusted chuck
- Between centres
- Face Plates
- Mandrels
- Collets
- Soft Jaws

4.4.1.1 - Production Machines

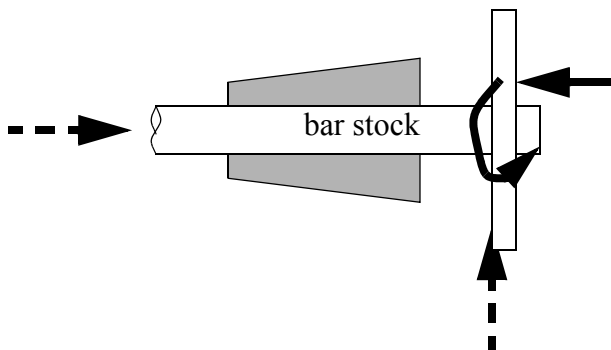
- In production there are a variety of cutting machines used to increase throughput by automatically feeding stock (through the headstock).



Collet - Stock is fed through from the back of the machine and clamped by the collet. The collet is then driven to turn the part and cutting tools cut the exposed stock and then the part is cut off, and the stock is advanced for the next part. This is the most basic process.



Sliding Headstock - In these machines the collet still grips the part, but it slowly moves forward. The cutting tools only move in a radial direction and are positioned near the bushing (it may have bearings also). Keeping the tools near the bushing reduced bending moments and allows slender parts to be cut.



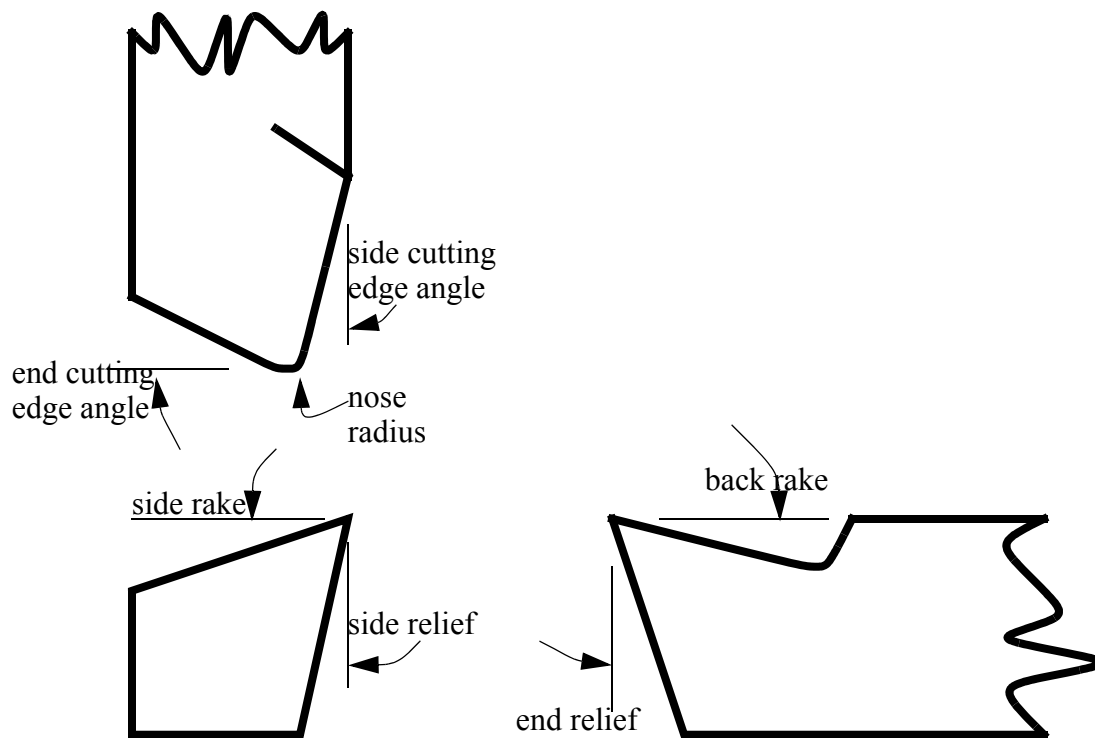
Esco - In this type of machine the bar stock is still held and advanced through a collet, but the tools rotate on a mounting assembly. The tools on the assembly can be moved in radial distances to change the profile of the part. This machine allows coiled stock to be turned and is suited to simpler parts.

- Other types of turning centers provide multiple operations on a single machine,
 - Multispindle - Multiple spindles in a single machine allows parallel operations in a sin-

- Rotary Transfer - Large machines where parts are moved to different stations, typically over ten stations. These may have other tools such as drills mounted.
- CNC machines - These computer controlled machines are typically flexible, but a bit slower. Flexibility is enhanced by a wider variety of operations and multiple tools in the same machine.
- Cam - For high production rates, cams can be made to drive the cutting heads. Their geometry will move the tool in complex patterns.

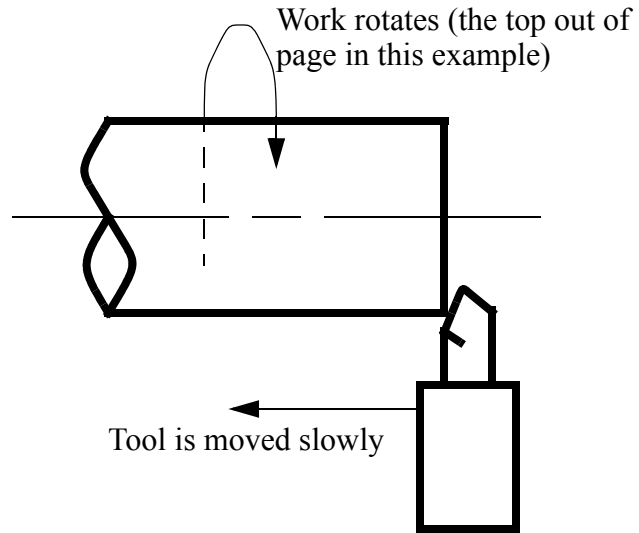
4.4.2 Toolbits

- A lathe toolbit is shown in the figure below, with a few terms defined.

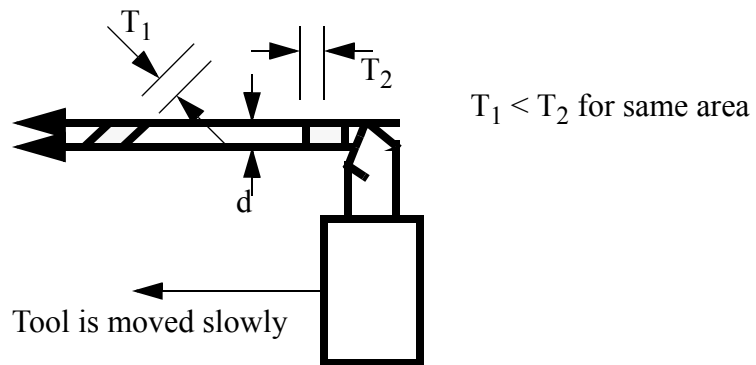


- In general, as the rake angle increases (positive), the cutting forces are reduced, the surface finish improves, and tool life increases.
- The side edge cutting angle has two effects outlined below,

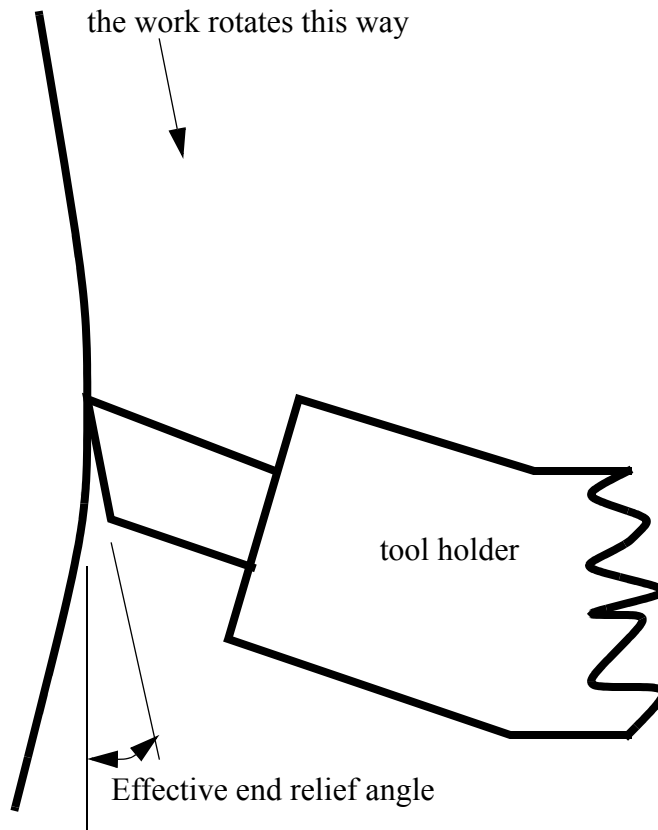
1. The angles edge allows a slow build up of cutting forces



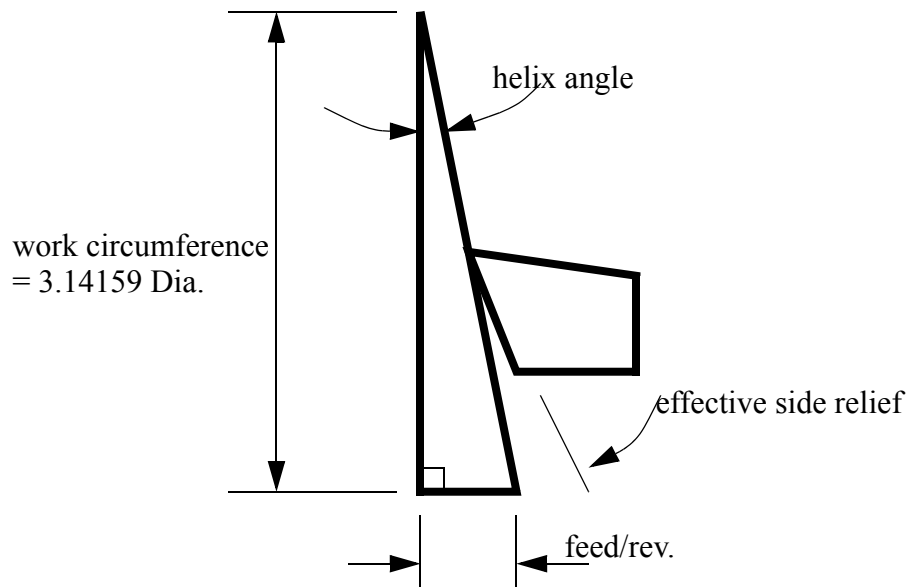
2. Increase in the side rake angle reduces the chip thickness



- The End Relief Angle prevents friction on the flank of the tool. The holder for the bit is often angled, and the end relief angle must be larger than the tool holder angle to prevent rubbing.



- The side relief angle has a function similar to the end relief, This angle must exceed the feed helix angle.



- Increasing the nose radius improves the surface finish. But this reaches a limit.

4.4.3 Thread Cutting On A Lathe

- Threads are cut using lathes by advancing the cutting tool quickly so that it cuts in a helical band. This helical band 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.
- The basic procedure is,
 1. The tool point must be ground so that it has the same angle as the thread to be cut. Typical angles are 60° for Vee threads, and 29° for ACME threads. A thread gauge can be used to measure thread angles. (also called Centre Gauge or Fish Tail Gauge).
 2. The correct gear ratio is required between the machine spindle to the lead screw. This can be determined with the equation,

$$ratio = \frac{driver}{driven} = \frac{TPI_{LEADSCREW}}{TPI_{WORKPIECE}}$$

where,

$TPI_{LEADSCREW}$ = the threads per inch on the lead screw (typically 4)

$TPI_{WORKPIECE}$ = the TPI to be cut on the workpiece

For example, to cut 20 TPI we calculate,

$$ratio = \frac{4}{20} = 5\left(\frac{4}{20}\right) = \frac{20}{100}$$

The increase is made to match the number of teeth available in our lathe (these figures depend on specific machine tools).

3. The compound slide is set at half the thread angle. This is so that as multiple passes are made to cut the thread (most threads require a few passes to cut), the tool will be advanced in by the compound slide in such a way that only one face cuts. If both faces were used for cutting there would be a good chance of vibrations and chatter. For example, if a 60° thread is being cut, the compound rest is often set at 29° .
4. The cutting tool is set in the holder perpendicular to the work, and the fishtail gauge is used to check the angle of the point.
5. The In-feed is set to the surface of the part for the first pass (quite often the first pass just scratches the surface to allow visual checking of the settings). On each subsequent pass the infeed will be set closer.
6. The cross slide is set at the same location for each cutting pass. i.e., the dial setting is zero.

7. The In-feed is adjusted on the compound slide for each pass by moving it in a distance. A simple measure of this distance is,

$$\Delta_{INFEED} = \frac{0.75}{TPI}$$

***** INCLUDE CHASING DIAL FIG 31-13

8. The chasing dial is used to restart the thread cutting in synchronization with what has been cut before. (If this step is not done properly, the notches in a thread might be cut over existing ridges - effectively cutting the entire thread flat to the bottom). The carriage of the lathe is driven across by a split nut. When the split nut is closed over the lead screw, it begins to move. It must be clamped over the lead screw when it is at the right angle. The method for doing this is with the chasing dial. The chasing dial has 16 different locations to engage at. In some cases you can engage the nut at any time, in other cases there are only a few positions to engage at. The basic rules are,

Calculate the following ratio (the previous example is used for illustration), and reduce the denominator to the smallest integer value.

$$R = \frac{TPI_{WORKPIECE}}{TPI_{LEADSCREW}} = \frac{20}{4} = \frac{5}{1}$$

Other examples could be,

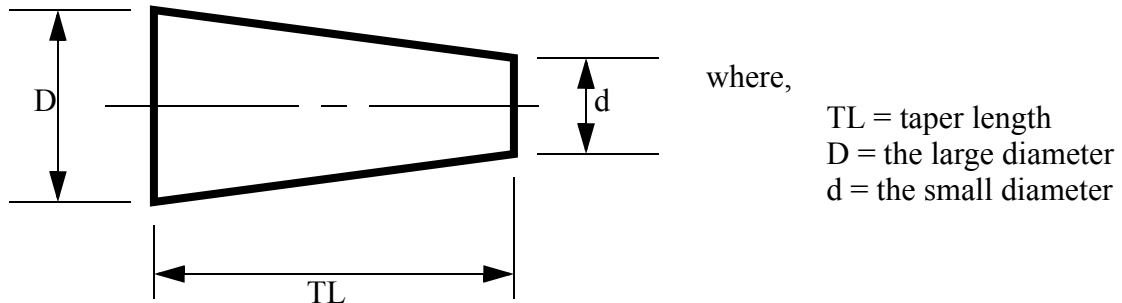
$$\frac{18}{4} = \frac{9}{2}, \frac{19}{4}, \frac{7\frac{1}{2}}{4} = \frac{15}{8}, \frac{3\frac{1}{4}}{4} = \frac{13}{16}$$

Then looking at the denominator only, select the positions of the chasing dial that the carriage can be engaged at,

DENOMINATOR	WHEN TO ENGAGE CARRIAGE
1	close nut at any position
2	every 1/8 of dial (e.g., at any line)
4	every 1/4 of dial (e.g., at any line with number)
8	every 1/2 of dial (e.g., 1 and 3, or 2 and 4)
16	every revolution at the same place (e.g., 1)

4.4.4 Cutting Tapers

- A taper is a conical shape.
- Tapers can be cut with lathes quite easily.
- The typical measures for tapers are shown below,



In Imperial:

$$tpf = \frac{D-d}{TL} \times 12$$

where,

D = large diameter (in.)
 d = small diameter (in.)
 TL = the taper length (in.)
 tpf = taper per foot (in./ft.)

In Metric:

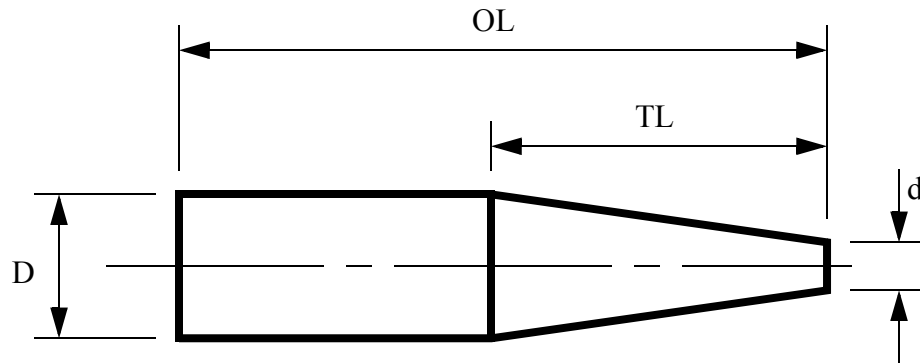
Specified as a ratio of mm change in diameter to length in mm
 For example, a 20cm long bar that changes in diameter from 3cm to 2.2cm
 would result in,

$$\Delta D : TL = (30 - 22) : 200 = 8 : 200 = 1 : 25$$

- Standard tapers include,
 - Lathe-Spindle Nose - Used for alignment of hole/shaft pairs
 - type D-1 (tpf = 3")
 - type L (tps = 3.5")
 - Self Holding Tapers - Used for stability
 - Taper shank drills, reamers, sleeves, etc.
 - Use "Morse Tapers" numbered 1 to 7

4.4.5 Turning Tapers on Lathes

- There are some common methods for turning tapers on a lathe,
 - Off-setting the tail stock
 - Using the compound slide
 - using a taper turning attachment
 - using a form tool
- 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. The method for determining the offset distance is described below.



$$OFFSET = \frac{OL}{TL} \times \frac{(D-d)}{2} = \frac{tpf \times OL}{24}$$

where,

OL = overall length

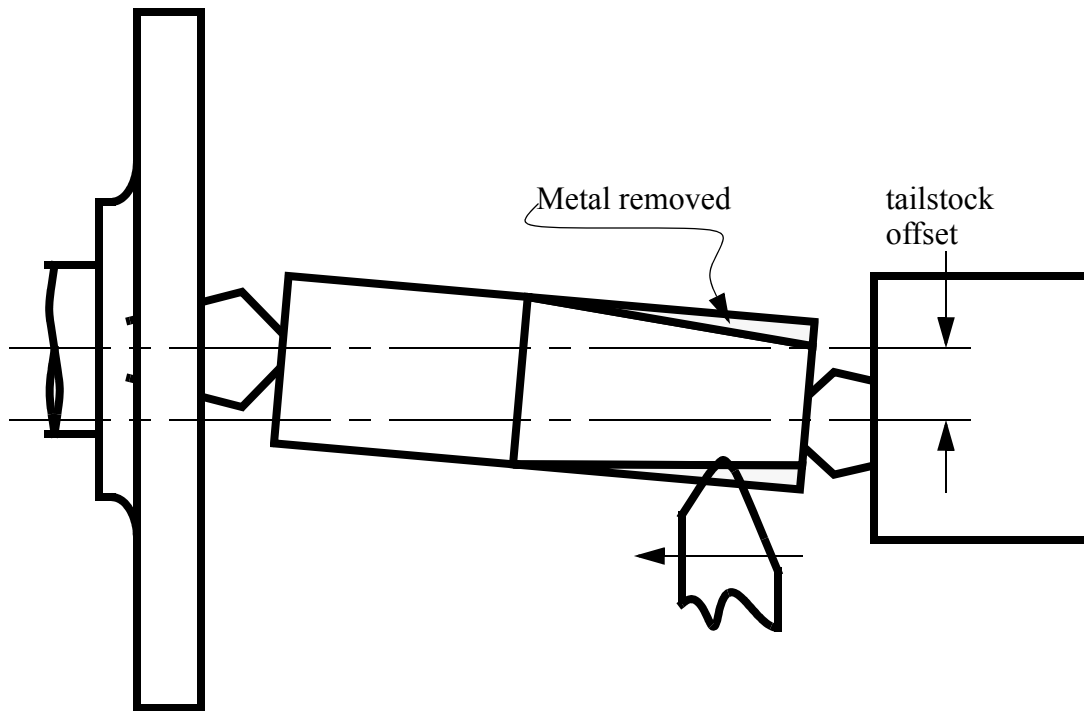
TL = taper length

D = the large taper diameter

d = the small taper diameter

tpf = taper per foot (in.)

OFFSET = the distance to move the tailstock from the zero setting



It is necessary to measure the tailstock offset when using this method. This can be done with,

1. A scale
2. A dial indicator

This method is limited to small tapers over long lengths.

The misalignment of the centres used in this method can cause damage to the work, and to the centres.

- The 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.
- Taper Turning Attachment - Additional equipment is attached at the rear of the lathe. The cross slide is disconnected from the cross feed nut. The cross slide is then connected to the attachment. As the carriage is engaged, and travels along the bed, the attachment will cause the cutter to move in/out to cut the taper.
- 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.

4.4.6 Feeds and Speeds

- If we consider the speed and feed of a lathe,
 - Spindle Speed is in revolutions per minute
 - Feed is in inches per revolution
- The Feed Chart is used to select the speeds and feeds of the lathe, and is often attached to the lathe near the setting levers.
- There are some simple (geometric) equations that can be listed,

$$CS = rpm \times C$$

$$C = \frac{\pi D}{12} \quad \text{imperial} \quad \frac{\pi D}{1000} \quad \text{metric}$$

$$rpm = \frac{12 \times CS}{\pi \times D} \quad \text{imperial} \quad \frac{1000 \times CS}{\pi \times D} \quad \text{metric}$$

where,

CS = cutting speed (fpm or m/s) - can be selected from tables

rpm = revolutions per minute of the machine spindle

C = circumference of the workpiece (ft. or m)

D = diameter of workpiece (in. or mm)

$$T = \frac{L}{rpm \times F}$$

$$C = T \times R$$

where,

L = length of cut (in. or mm)

F = feed rate (in./rev. or mm/rev.) - found in tables

R = Machine cost (\$/min.)

- Typical cutting speeds for a high speed steel tool are, [Krar]

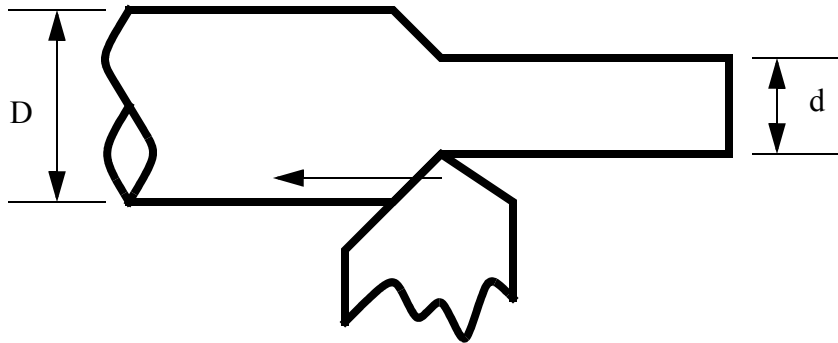
Material	Rough Cut (fpm)	Finish Cut (fpm)	Thread cut (fpm)
machine steel	90	100	35
tool steel	70	90	30
cast iron	60	80	25
bronze	90	100	25
aluminum	200	300	60

- Typical feeds when using a high speed steel tool are, [Krar]

Material	Rough Cut (in./rev.)	Finish Cut (in./rev.)
machine steel	0.010-0.020	0.003-0.010
tool steel	0.010-0.020	0.003-0.010
cast iron	0.015-0.025	0.005-0.012
bronze	0.015-0.025	0.003-0.010
aluminum	0.015-0.030	0.005-0.010

4.4.7 The mrr for Turning

- considering the parameters defined in the discussion of speeds and feeds, etc, the mrr is given below,



$$mrr = \left(\frac{\pi D^2}{4} - \frac{\pi d^2}{4} \right) \times F \times rpm$$

where,

D = diameter of workpiece before cutting

d = diameter of workpiece after cutting

4.4.8 Process Planning for Turning

- The general steps when process planning for turning external parts are,
 1. Rough cuts all diameters to within 1/32" starting with the largest diameters first.
 2. Rough cut all shoulders and steps to within 1/32"
 3. Do special operations such as knurling and grooving
 4. Cool the workpiece to get it close to the final dimension.
 5. Finish turn the diameters, then the shoulders and steps
 6. Deburr if necessary

- If the part is to be mounted between centres, plan should precede by,
 1. cut stock that is 1/8" larger than required.
 2. Put the work in the lathe, in a chuck, and face and centre drill the end.
 3. reverse the piece in the chuck and face the piece to size, and centre drill.
 4. Mount the work between centres

- For work to be mounted in a chuck, (implies internal features),
 1. cut the stock 1/8" wider in diameter, and 1/2" longer.
 2. Mount the work in the chuck with 5/16" to 3/8" inside.
 3. Use a facing operation (lightly) to square the end.
 4. Rough cut the external diameters, from the largest to the smallest.
 5. Drill out the centre of the work using a drill chuck mounted in the tailstocks spindle. Start with a centre drill, and increase drill sizes to increase the hole.
 6. Mount a boring tool to cut the internal diameter to close to the final diameter.

7. Cut any special feature now.
8. Do finish cuts on outside and inside.
9. Reverse the part in the chuck and face off the material to size. Protect the work by placing a piece of soft metal between it and the chuck.

4.5 Cutting Speeds, Feeds, Tools, and Times

- Cutting is a balance between a number of factors,
 - cutting slowly will add costly time to manufacturing operations.
 - cutting faster will lead to decreased tool life, and extra time will be required to repair tools.
- Some reasonable speeds and feeds for a single cutting point tool are given below [Krar],

MATERIAL	DEPTH (in.)	FEED PER REV. (ipr)	CUTTING SPEED (fpm)
Aluminum	0.005-0.015	0.002-0.005	700-1000
	0.020-0.090	0.005-0.015	450-700
	0.100-0.200	0.015-0.030	300-450
	0.300-0.700	0.030-0.090	100-200
Brass, Bronze	0.005-0.015	0.002-0.005	700-800
	0.020-0.090	0.005-0.015	600-700
	0.100-0.200	0.015-0.030	500-600
	0.300-0.700	0.030-0.090	200-400
cast iron (medium)	0.005-0.015	0.002-0.005	350-450
	0.020-0.090	0.005-0.015	250-350
	0.100-0.200	0.015-0.030	200-250
	0.300-0.700	0.030-0.090	75-150
machine steel	0.005-0.015	0.002-0.005	700-1000
	0.020-0.090	0.005-0.015	550-700
	0.100-0.200	0.015-0.030	400-550
	0.300-0.700	0.030-0.090	150-300
tool steel	0.005-0.015	0.002-0.005	500-750
	0.020-0.090	0.005-0.015	400-500
	0.100-0.200	0.015-0.030	300-400
	0.300-0.700	0.030-0.090	100-300
stainless steel	0.005-0.015	0.002-0.005	375-500
	0.020-0.090	0.005-0.015	300-375
	0.100-0.200	0.015-0.030	250-300
	0.300-0.700	0.030-0.090	75-175
titanium alloys	0.005-0.015	0.002-0.005	300-400
	0.020-0.090	0.005-0.015	200-300
	0.100-0.200	0.015-0.030	175-200
	0.300-0.700	0.030-0.090	50-125

4.6 Cutting Power

- There are a number of reasons for wanting to calculate the power consumed in cutting. These numbers can tell us how fast we can cut, or how large the motor on a machine must be.
- Having both the forces and velocities found with the Merchant for Circle, we are able to calculate the power,

$$\begin{array}{l}
 P_c = \frac{F_c V_c}{33000} \\
 P_s = \frac{F_s V_s}{33000} \\
 P_f = \frac{F \times V_f}{33000}
 \end{array}
 \quad \text{All have units of Horsepower (i.e., 1/33000)}$$

where,

$$\begin{array}{l}
 P_c = \text{the total cutting power} \\
 P_s = \text{the shearing power required} \\
 P_f = \text{the friction losses}
 \end{array}$$

- We can relate the energy used in cutting to the mrr.

$$\text{Energy Consumed} \quad P_c = F_c \times V_c$$

$$\text{Metal Removal Rate} \quad Q = A_0 \times V_c$$

where,

$$A_0 = \text{Area of Cut}$$

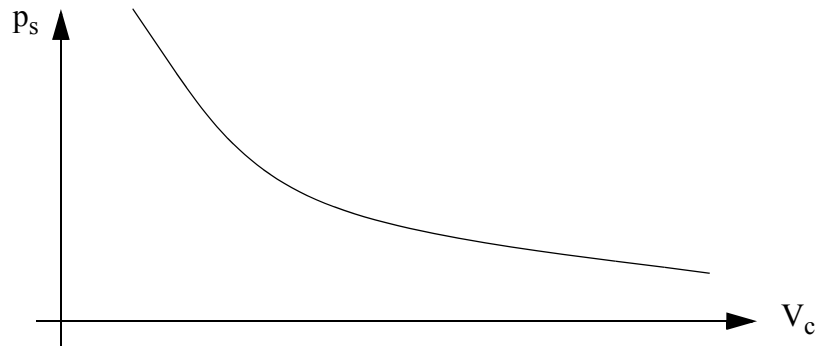
***Note: both W_c and Q are proportional to V_c

From these basic relationships we can a simple relationship that is the ratio between the energy consumed, and the volume of metal removed,

$$p_s = \frac{P_c}{Q} = \frac{F_c \times V_c}{A_0 \times V_c} = \frac{F_c}{A_0}$$

You will notice that the result is a force over an area, which is a pressure. As a result P_s will be called the Specific Cutting Pressure.

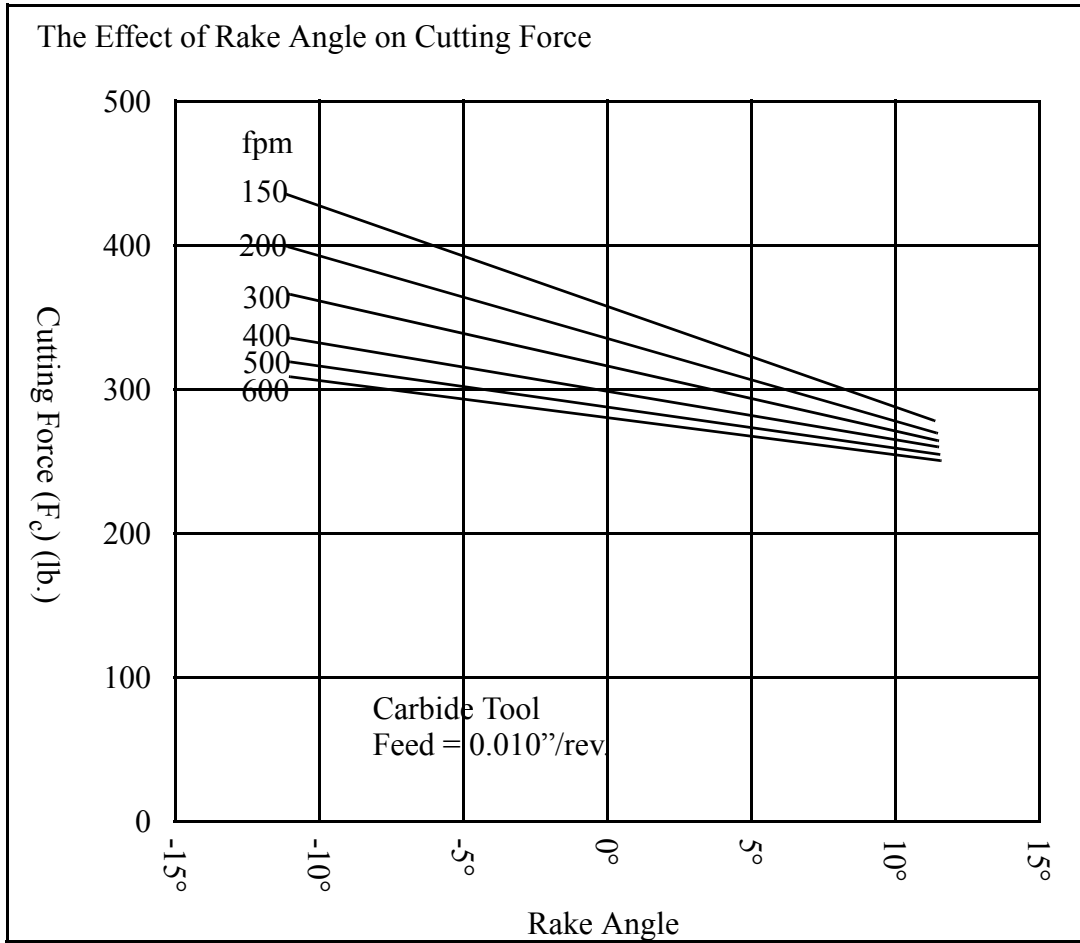
- The cutting force will vary, thus changing P_s , as the cutting velocities are changed.



This curve turns downward for two reasons,

1. The tool experiences edge forces that are more significant at lower cutting speeds.
2. As the velocity increases, the temperature increases, and less energy is required to shear the metal.

- The effects of rake angle on cutting are shown in the graph below, [REF *****]



- The horsepower required for cutting can be found using empirical methods,

Unit horse power (HP_u) is the amount of power to remove a volume of metal in a period of time.

HP_u = power to cut 1 cubic inch per minute - found in tables

$HP_g = Q \times HP_u =$ Gross Horsepower

Average Unit Horsepower Values of Energy Per Unit Volume [REF]		
Material	BHN	HP_u (HP/(in ³ /min.))
Carbon steels	150-200	1.0
	200-250	1.4
	250-350	1.6
Leaded steels	150-175	0.7
Cast irons	125-190	0.5
	190-250	1.6
Stainless steels	135-275	1.5
Aluminum alloys	50-100	0.3
Magnesium alloys	40-90	0.2
Copper	125-140	0.7
Copper alloys	100-150	0.7

- If we consider the implications these formulas have when cutting on a lathe, we would be able to develop the following equations,

$$Q = f \times d \times V \times 12$$

where,

f = feed

d = depth or cut

V = velocity

$$HP_c = \frac{F_c \times V_c}{33000} = HP_u \times Q \times c$$

where,

c = a feed factor from tables

Feed		Factor
(ips or ipr)	(mm/rev or mm/stroke)	
0.002	0.05	1.4
0.005	0.12	1.2
0.008	0.20	1.05
0.012	0.30	1.0
0.020	0.50	0.9
0.030	0.75	0.80
0.040	1.00	0.80
0.050	1.25	0.75

We can also consider the efficiency of the machine tool,

$$HP_g = \frac{HP_c}{e_m}$$

where,

$$e_m = \text{the machine tool efficiency factor } [0, 1]$$

from this we can determine the minimum machine tool horsepower required,

$$HP_M = HP_I + HP_g = HP_I + \frac{HP_c}{e_m}$$

where,

$$HP_M = \text{The minimum machine tool horse power required}$$

$$HP_I = \text{The idle horsepower consumed by the machine tool}$$

4.7 Examples

4.8 Summary

4.9 References and Bibliography

4.10 Problems

4.11 Challenge Problems

4.12 Practice Problems

1. What would happen if a drill without flutes was used?
2. If we want a hole with a 1/2-14-UNC thread, what size of tap drill should be used?
3. What type of drill press is suitable for drilling holes in car engine blocks? Justify your answer.
4. Which of these statements is not correct?
 - a) work is not moved on a radial arm drill press.
 - b) automatic feeds are available on sensitive drill presses.
 - c) multispindle drill presses always drill multiple holes at once.
 - d) all of the above.
5. Which of these statements is correct?
 - a) a margin of a drill bit does most of the cutting.
 - b) the relief angle on the tip of the drill bit makes it not a conical shape.
 - c) a large drill bit point angle is useful for cutting soft materials.
 - d) none of the above.
6. Which of the following statements is not correct?
 - a) core drills have 3 or 4 flutes.
 - b) high helix drills help in chip removal.
 - c) straight fluted drills are used for sheet metal.
 - d) centre drills are for long holes, such as gun barrels.
7. Which of the following is not a typical drill press operation?
 - a) counter boring.
 - b) spot facing.
 - c) counter sinking.
 - d) none of the above.
8. Which of the statements is most correct?
 - a) reamers are used to finish holes with accuracies not possible when a normal drill is used.
 - b) adjustable taps will cut a wide variety of threaded holes.
 - c) taps and reamers can both be used without a machine tool.
 - d) none of the above.
9. Given a hole that is to be drilled then reamed to 3.000", develop a process plan including speeds and feeds.
10. We want to drill a hole that is 2.369" in diameter. If we know that the accuracy the shop can provide for drilling is +0.030" to -0.010",
 - a) what is an appropriate fractional drill size to use?
 - b) what operation might follow?

11. Calculate the machine tool spindle speeds for the following,
- drilling with a $19/32$ " high speed steel bit in mild steel. The CS is 70 ft./min.
12. We are to drill 6 holes in a 2" thick mild steel plate. The plate is held in a jig. We are using a $63/64$ " high speed steel drill, and the suggested parameters are CS = 80 ft./min. with a feed of 0.004"/rev. After drilling each hole is to be finished with a 1.0" diameter reamer. If the suggested parameters for the reamer are CS = 80 ft./min. with a feed of 0.010"/rev.,
- calculate the time to do all of the operations (and make allowances for drill point travel)
 - find the cost to produce 500 parts when each part needs 3 minutes for setup (no operation), labor rates are \$25/hr., and overhead is \$25/hr.
13. Which of these statements is not correct?
- work is not moved on a radial arm drill press.
 - automatic feeds are available on sensitive drill presses.
 - multispindle drill presses must always drill multiple holes at once.
 - all of the above.

ans. B

14. Which of these statements is correct?
- a margin of a drill bit does most of the cutting.
 - the relief angle on the tip of the drill bit makes it a conical shape.
 - a large drill bit point angle is useful for cutting soft materials.
 - none of the above.

ans. D

15. Which of the following statements is not correct?
- core drills have a hollow center to remove chips.
 - high helix drills help in chip removal.
 - straight fluted drills are used for sheet metal.
 - centre drills are for long holes, such as gun barrels.

ans. A or D

16. Which of the following is not a typical drill press operation?
- counter boring.
 - spot facing.
 - counter sinking.
 - none of the above.

ans. D

17. Which of the statements is most correct?
- reamers are used to finish holes with accuracies not possible when a normal drill is used.

- b) adjustable taps will cut a wide variety of threaded holes.
- c) taps and reamers can both be used without a machine tool.
- d) none of the above.

ans. A

18. What are functions of the following parts of a drill bit. a) body, b) web, c) point, d) tang, e) margin, f) flutes, g) body clearance.
 19. What are the purposes of the following drill points. a) conventional, b) flat, c) long angle.
 20. What applications are the following drill bits well suited to? a) high helix, b) straight flute, c) gun, d) hard steel, e) core, f) oil hole.
 21. What will happen if a drill bit has unequal angles on the cutting edges/lips? What if the edges are not of equal length?
 22. Why should most holes be started with a center drill?
 23. What are the disadvantages of a thick web found on some drills?
 24. What is the purpose of pilot holes?
 25. What is the main difference between a) threading operations and tapping operations? b) boring and reaming?
 26. List 5 ways work can be held in a lathe.
 27. Can peripheral and face milling be done with the same cutter? How common is this?
 28. Describe the steps in cutting a 3/8-12-UNC tapped hole.
(ans. center drill, drill 1/4", drill .292", starting tap, finishing tap)
 29. a) Explain the cutting mechanism of a drill bit, and b) suggest the features of a drill bit for cutting a thin piece of sheet metal.
-
1. A 2" diameter milling cutter with 8 teeth has been selected. What is the table feed if we are milling at 80 ft./min. with a tooth load of 0.004"/tooth?
 2. Calculate the machine tool spindle speeds for the following,
 - a) milling with a 3/4" high speed steel cutter in tool steel work. The CS is 60 ft./min.
 - b) milling with a 150mm diameter tungsten carbide tipped face cutter in stainless steel work. The CS is 65 m/min.

3. You are given a block of aluminum (5" by 5" by 5") and you must mill off a 1/16" layer. Using the tables for speeds and feeds, and using the other details provided below, determine a cost for the operation.

- Milling cutter- high speed steel
- diameter 2"
- 10 teeth with a tooth load of 0.004" per tooth
- cost for the machine is \$20.00 per hours

ans. This cut is more than a finishing cut.

We can assume this is a light rough cut or heavy finishing cut. The cutter type will be assumed to be a face mill. Because the part width is 5" and the cutter is 2" we will need three passes to cut the part.

$$D = 2in \quad \#t = 10 \quad f_{pt} = 0.004 \quad CS = \frac{500 + 1000}{2} = 750\text{fpm}$$

$$R = 20\$/\text{hr}$$

$$rpm = \frac{12 \times CS}{\pi \times D} = \frac{12(750)}{\pi 2} = 1432$$

$$F = f_{pt} \times \#t \times rpm = (0.004)(10)(1432) = 57\text{ipm}$$

$$L_{pass} = 5in + D \quad L = 3L_{pass} = 3(5 + 2) = 21in$$

$$T = \frac{L}{F} = \frac{21in}{57ipm} = 0.37min$$

$$C = T \times R = \left(\frac{0.37}{60}\right)(20) = 0.12\$$$

4. Which of the following statements is true for milling?
- a) milling cutters can cut with the face and peripheral teeth.
 - b) the cutting edge moves opposite to the direction used in lathes.
 - c) indexing is used to cut rounded surfaces.
 - d) none of the above.
5. What are the advantages of upcut and downcut milling?

ans. Upcut - lower tool impact forces
 Upcut - loose work is safer
 Downcut - pushes work into table
 Downcut - better surface finish

6. Which of the following statements is true for milling?
- a) milling cutters can cut with the face and not the peripheral teeth.
 - b) the cutting edge moves opposite to the direction used in lathes.
 - c) indexing is only used to cut rounded surfaces.
 - d) none of the above.

ans. D

7. Given a 3" dia. 8 tooth fly cutter, with carbide cutting points, and a steel work piece, recommend, a) RPM, b) feed.
8. Given a 6" dia. high speed steel arbor mill with 10 teeth, that will be cutting cast iron work, recommend, a) RPM, feed.
9. Calculate the indexing required when would be cutting a gear with 36 teeth? Use one of the Brown and Sharp indexing plates.
10. Determine the angular indexing required (on Brown and Sharp, and Cincinnati Standard plates) if we want an angle of $23^{\circ}30'$.

1. Given that a tapered piece is to be made with the tailstock offset method, determine the taper per foot, and offset required if, you are starting with a bar of stock that is 8" long, and 1.125" in diameter, and the final taper is to be 6" long and 1" at the small end.
(ans. tpf = 0.25", offset = 0.0833")

2. Given the 1/2-12 UNC thread that is to be cut on the lathe,
- a) What should the gear ratio between the machine spindle and the lead screw be if the lead screw is 5 t.p.i.?
 - b) What should the in-feed be for each pass?

3. Which of the following statements about lathe toolbits is correct?
- a) a small nose radius will result in a smoother surface.
 - b) small relief angles will always increase friction.
 - c) large rake angles will decrease cutting forces.
 - d) none of the above.

(ans. c)

4. Which of the statements about lathes below is most correct?
- a) jawed chucks hold only standard sizes of pieces.
 - b) mandrels hold work pieces from the outside.
 - c) the chasing dial is used for measuring fine cuts.
 - d) lead screws and feed rods are lathe parts

(ans. d)

5. When turning between centres a dog is required; what is a dog in this context?

(ans. it holds the work piece so that it can be driven with a face plate mounted on the lathe spindle)

6. A centre gauge (fish tail gauge) is employed in thread cutting. Suggest two uses for the gauge.

(ans. aligning a cutting tool for threads,)

7. Given an external 9/16-12-UNC thread, determined which tools would be used.

(ans. a turning tool to turn the outside diameter of 9/16" + 1/12", UNC tool to turn thread)

8. If we are rough cutting a 5" diameter bar of bronze on a lathe with a HSS tool,

a) what speed and feed should be used?

b) if the cut is 12" long, and will be made in two passes, how long will the operation take?

c) if the setup time is 5 minutes, and the machine rate is \$50/hr., what will the cost of the operation be?

Using the lookup tables in the notes we pick a surface cutting speed and feed.

$$D = 5in \quad L = 12in \quad R = 50\$/hr$$

Rough

Finish

$$CS = 90 \frac{ft}{min}$$

$$CS = 100 \frac{ft}{min}$$

$$f = \frac{0.015 + 0.025}{2} \frac{in}{rev} = 0.02 \frac{in}{rev}$$

$$f = \frac{0.003 + 0.010}{2} \frac{in}{rev} = 0.0065 \frac{in}{rev}$$

$$rpm = \frac{12CS}{\pi D} = 68$$

$$rpm = \frac{12CS}{\pi D} = 76$$

$$T = \frac{L}{rpm \times f} = 17.39min$$

$$T = \frac{L}{rpm \times f} = 24.29min$$

$$C = T \times R = \left(\frac{2(8.82) + 5}{60} \right) 50 = 18.71\$$$

$$C = T \times R = \left(\frac{2(24.29) + 5}{60} \right) 50 = 44.65\$$$

9. List the basic steps for setting up a lathe to cut a thread on a bar of stock, assume the stock is mounted between centres already.

(ans. see thread cutting section)

10. If a taper of 1mm in 10mm is to be cut, what will the offset distance be for a 10cm part?

11. Calculate the machine tool spindle speeds for the following,
a) turning on a lathe with a high speed steel tool in mild steel work with a diameter of 2.75". The cutting speeds is 100 ft./min.
12. We have been given a mild steel bar that is to be turned on a lathe. It has a diameter of 14" and a length of 28". We have been asked to make two rough passes, and one finishing pass. The tool we have selected is Carbide. When doing rough cuts we use a feed of 0.007"/rev., and for finishing cuts we use a feed of 0.004"/rev. How long will this operation take?
13. Which of the following statements about lathe toolbits is correct?
a) a small nose radius will result in a smoother surface.
b) small relief angles will always increase friction.
c) large rake angles will decrease cutting forces.
d) none of the above.

ans. C or D

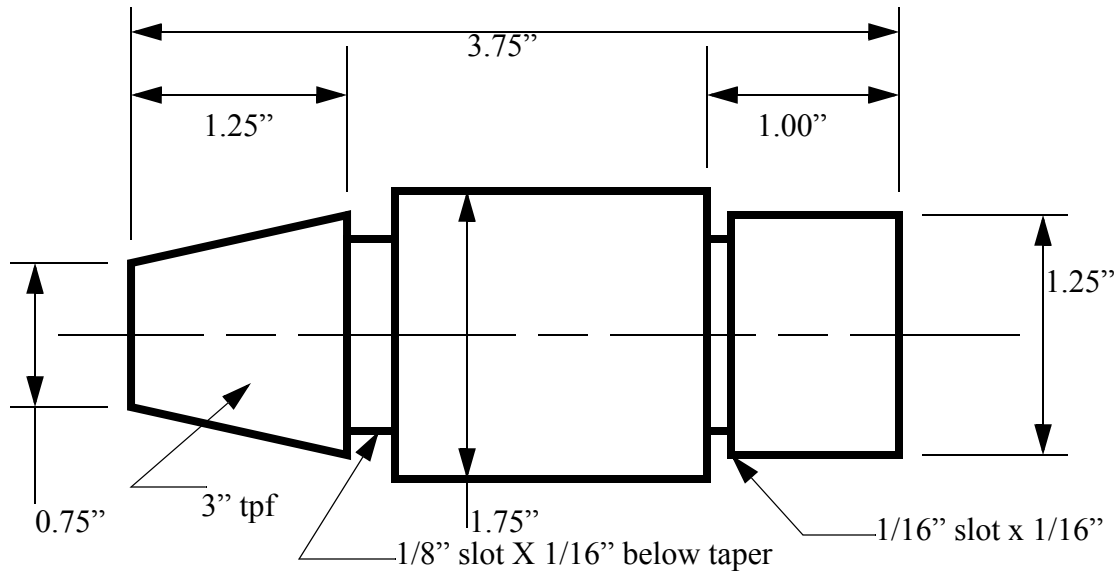
14. Which of the statements about lathes below is most correct?
a) jawed chucks hold only standard sizes of pieces.
b) collets hold work pieces from the outside.
c) the chasing dial is used for measuring fine cuts.
d) lead screws and indexers are lathe parts

ans. B

15. Given the non-standard 3/8-19 UNC thread that is to be cut on the lathe,
a) What should the gear ratio between the machine spindle and the lead screw be if the lead screw is 4 t.p.i.?
b) What should the in-feed be for each pass?

ans. a) 4/19, b) 0.039

16. Develop a rough process plan for the part below by clearly listing operation steps in the correct sequence. Feeds, speeds, times and costs are not needed at this time.



Operation Number	Operation Description
0010	Cut off 2" dia. Stock to 4"
0020	Mount in lathe chuck, face and centre drill

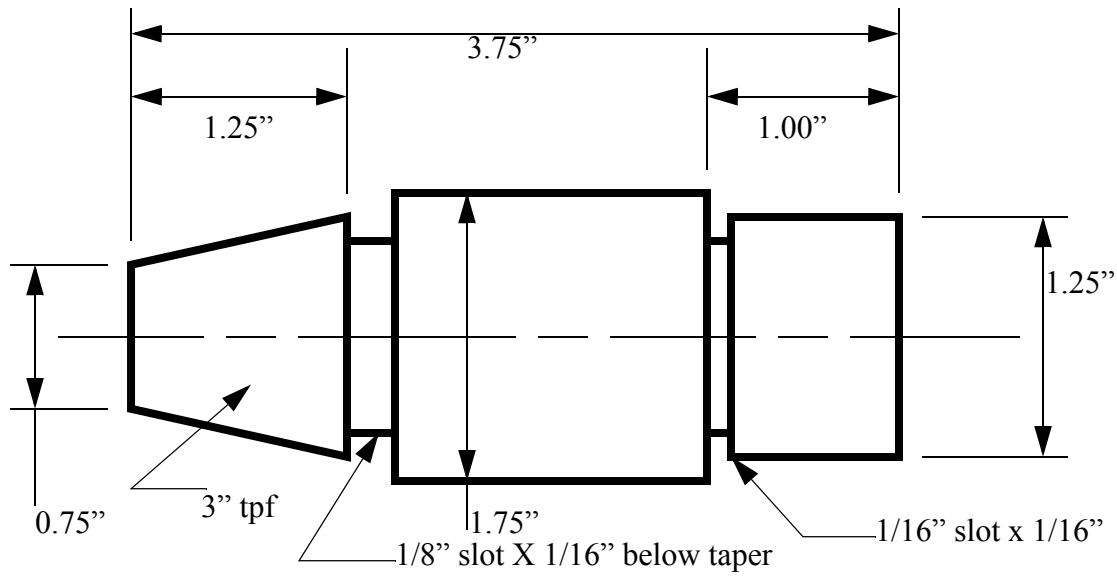
ans.

Operation Number	Operation Description (Note excess details given for beginners)
0010	Cut off 2" dia. Stock to 4"
0020	Mount in lathe chuck, face and center drill to 3.75" length
0030	Mount between centers
0040	Turn entire length to 1.75" dia.
0050	Cut slot with form tool 1" from end to 1/16" depth
0060	Turn one end down to 1.25" dia. for 15/16"
0070	Reverse part in centers (cover finished end with soft metal)
0080	Cut 1/8" by 1/16" slot
0090	Turn taper with taper turning attachment
0100	Return tailstock to normal position
0110	Deburr and inspect

*Note: the implied tolerances +/- 0.005 would not require cooling

17. The aluminum component below is to be turned on a lathe using a HSS tool. Develop a process plan, including offset for the taper, speeds, feeds, etc. Put the process plan in a list similar to the format shown. Assume a cost of \$45.00/hr. for the lathe, and \$25.00/hr. for all other

pieces of equipment. State all assumption clearly, and justify numbers in the process plan with calculations or references.

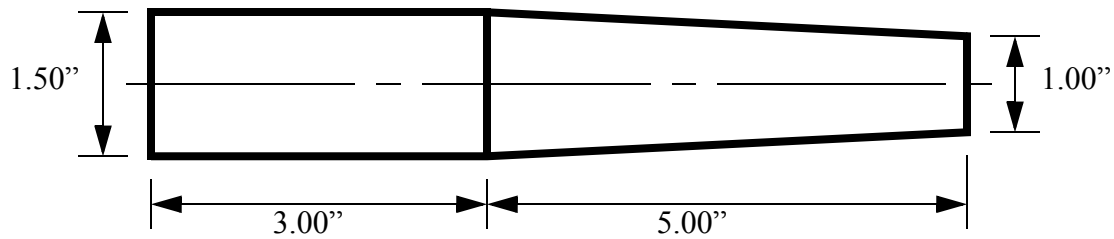


Operation Number	Operation Description	Time	Cost
0010	Cut off Stock to 4"	6 min.	\$5.00
0020	Mount in lathe chuck, face and centre drill	12 min.	\$9.00

18. On a lathe toolbit what are the functions of, a) the side relief angle, b) end relief angle, c) back rake, d) side rake angle, nose radius.
19. What applications are large positive rake angles for? negative rake angles?
20. What is the difference between end and face milling?
21. What RPM should be used to rough cut a cast iron piece with a 3" dia. with a high speed steel tool. What RPM should be used for a similar workpiece of plain carbon steel? What RPM should be used for the two materials if finishing cuts are being made?
22. Calculate the time required to machine a 2" dia. copper rod that is to be turned for a length of 10".
23. What are rough and finish turning operations used for?
24. What are two methods for cutting stepped shoulders on a lathe?
25. Explain the difference between self holding and steep tapers using the coefficient of friction.
26. Find the tpf and tailstock offset for tapers on the following work.

a) $D=1.5''$, $d=1.25''$, $TL=4''$, $OL=8''$

27. convert a metric taper of 1:50 to a tpf. Convert a 1"tpf to metric.
28. Define the terms, fit, tolerance, allowance, limits, clearance, press fit, precision.
29. For a 1"-8-NC thread find the minimum and maximum diameters and minimum width of the toolbit point.
30. Describe the differences in speeds, feeds and depths of cuts for roughing and finishing cuts.
31. What types of chips are desirable when setting up automated cutting processes?
32. Compare the time to cut a work piece using a high speed steel tool and a carbide tool. The 4" dia. aluminum work is to be rough turned over a length of 14".
33. What operations can be performed on a lathe?
34. How are the parameters different for a lathe when turning, as opposed to finishing?
35. A taper is to be cut on the aluminum part below. Indicate how far the tailstock should be offset and the speed and feed settings for the lathe.



(ans. offset=0.4", feed 0.005-0.010", speed 760RPM)

1. An orthogonal cut is made with a carbide tool having a 15° positive rake angle. The various parameters were noted,
 - the cut width was 0.25"
 - the feed was set at 0.0125"
 - the chip thickness was measured to be 0.0375"
 - the cutting speed was 250 ft./min.
 - the forces measured were $F_c = 375$ lb. and $F_t = 125$ lb.

- a) Use Merchant's Circle to scale, and the velocity diagram
 - b) From the Merchant Circle diagram find the shear angle (ϕ), friction force (F), friction normal force (N), and shear force (F_s).
 - c) From the Velocity diagram find the friction velocity (V_f).
 - d) Calculate values for the coefficient of friction (μ) and the metal removal rate.
 - e) Calculate values, and compare the results for the results found in a), b) and c).
- (ans: $F = 218\text{lb.}$, $N = 330\text{lb.}$, $\phi = 19.37^\circ$, $F_s = 312\text{ lb.}$, $\mu = 0.948$, $V_c = 250\text{ ft./min.}$, $V_f = 83.5\text{ ft./min.}$ $Q = 9.375\text{ in}^3/\text{min.}$)

2. The cutting forces for a lathe are listed below,

- work RPM = 125
- feed/rev = 0.005"
- chip thickness = 0.0123"
- rake angle of tool = 14°
- $F_t = 150\text{ lb}$, $F_c = 245\text{ lb}$
- work diameter = 8"

- a) Find the horsepower consumed in cutting, shearing and friction.
- b) Find a maximum lathe horsepower, assuming the machine efficiency is 95% and it requires 1/8 idle horsepower.
- c) Based on the cutting horsepower, what material(s) might we be cutting?

3. What roles do rake and relief angles play in cutting tools?

ans. the rake angle will change the basic cutting parameters. A positive rake (sharp tool) will give lower cutting forces, but less edge strength. A negative or neutral rake will give higher cutting forces, but more strength. The relief angle provide a gap behind the cutting edge so that the tool does not rub the work.

4. Which of these statement is the most correct?

- a) a continuous chip with built up edge may result when we try to cut too much metal.
- b) a continuous chip will result when cutting very brittle work materials.
- c) a discontinuous chip will result when we use fine feeds and speeds.
- d) none of the above.

ans. a

5. One of the assumptions behind orthogonal cutting is,

- a) that the rake angle is positive.
- b) that the tool is only cutting with one edge and one point.
- c) the shear plane is a function of before and after chip thicknesses.
- d) none of the above.

ans. b

6. Which of these statements is correct?

- a) the cutting pressure drops as cutting velocity increases.
- b) power required drops as metal temperature and cutting velocity increase.
- c) we can use the quantity of metal removed by itself to estimate the required horsepower of a machine tool.
- d) all of the above.

ans. a

7. A lathe toolbit with a rake angle of 20° is cutting a section of pipe with an inner diameter of 6" and an outer diameter of 6.25". The cut has a depth of 0.010" and the chip has a thickness of 0.020". If the lathe is turning at 200 rpm, and the measured cutting forces are $F_c = 300$ lb, and $F_t = 125$ lb,

- a) what assumption must you make.
- b) find the following values using a graphical or numerical solution: (Marks are only awarded for correct answers) F_s , F_N , F , N , τ , ϕ , μ , V_c , V_f , V_s .
- c) what is the minimum horsepower required for the machine?
- d) given that the tube is aluminum, use another method to find the required horsepower.

ans.

$$\alpha = 20\text{deg} \quad F_c = 300\text{lbs} \quad F_t = 125\text{lbs} \quad t_1 = d = 0.010\text{in}$$

$$RPM = 200 \quad t_2 = 0.020\text{in} \quad D = \left(\frac{6 + 6.25}{2}\right) = 6.125\text{in}$$

a) reasonable assumptions are that we are performing orthogonal cutting. This means that we are cutting fully through the wall of the tube. We also want to assume that the effects of the different cutting speeds from the inside to the outside of the tube are negligible.

$$b) \quad r_c = \frac{t_1}{t_2} = 0.5 \quad \phi = \text{atan}\left(\frac{r_c \cos \alpha}{1 - r_c \sin \alpha}\right) = \text{atan}(0.56676719) = 29.5\text{deg}$$

$$F = F_t \cos \alpha + F_c \sin \alpha = 220\text{lbs} \quad N = F_c \cos \alpha - F_t \sin \alpha = 239\text{lbs}$$

$$F_s = F_c \cos \phi - F_t \sin \phi = 200\text{lbs} \quad F_n = F_t \cos \phi + F_c \sin \phi = 257\text{lbs}$$

$$\mu = \frac{F}{N} = 0.92 \quad \tau = \text{atan}(0.92) = 42.6\text{deg}$$

$$V_c = \frac{RPM \pi D}{12} = \frac{200 \pi (6.125)}{12} = 321\text{fpm}$$

$$V_s = \frac{321 \cos 20}{\cos(29.5 - 20)} = 306\text{fpm} \quad V_f = \frac{321 \sin 29.5}{\cos(29.5 - 20)} = 160\text{fpm}$$

$$c) \quad HP_c = \frac{F_c V_c}{33000} = \frac{300(321)}{33000} = 2.9\text{HP} \quad \text{For an efficient machine with no idle horsepower.}$$

$$d) \quad Q = d(6.25 - 6)V_c 12 = 0.010(0.25)321(12) = 9.63 \frac{\text{in}^3}{\text{min}}$$

$$HP_c = HP_u Q = 0.3(9.63) = 2.9\text{HP}$$

8. Calculate the machine tool spindle speeds for the following:

a) Milling with a tungsten carbide tipped face cutter on a stainless steel work piece. C.S. = 65 m/min., cutter dia. = 150mm.

b) Drilling with a High Speed Steel drill in Machine Steel work, with C.S. = 70 ft./min., and a drill diameter of 19/32"

c) Turning on a lathe with a High Speed Steel tool in a mild steel work piece. Surface cutting speed = 100 ft./min., and a workpiece diameter of 2.75"

d) Milling with a High Speed Steel cutter in tool steel work with a cutter speed of 60 ft./min., and a cutter diameter of 3/4".

9. Short answer,

- a) Why are ceramics normally provided as inserts for tools, and not as entire tools?
 b) List the important properties of cutting tool materials and explain why each is important.

- ans. a) Ceramics are brittle materials and cannot provide the structural strength required for a tool.
- b) hardness at high temperatures - this provides longer life of the cutting tool and allows higher cutting speeds.
 toughness - to provide the structural strength needed to resist impacts and cutting forces
 wear resistance - to prolong usage before replacement
 doesn't chemically react - another wear factor
 formable/manufacturable - can be manufactured in a useful geometry

10. A turning cut was made in a magnesium workpiece with a feed of 0.050ipr. The cutting speed was 300 fpm, and the cutting force was measured as 200lbs. The lathe is 95% efficient and has an idle horsepower of 0.1HP. Using all of the provided information estimate the horsepower required for the cut.

ans.

$$HP_c = \frac{V_c F_c}{33000} = \frac{300 \frac{ft}{min} (200lbs)}{33000 \frac{ftlbs}{minHP}} = 1.82HP$$

$$HP_M = HP_I + \frac{HP_c}{e} = 0.1HP + \frac{(1.82HP)(0.75)}{0.95} = 1.54HP$$

11. Develop an expression that is the ratio friction power over cutting power using the equations for orthogonal cutting power. Simplify the expression to be in terms of measured values (rake angle, F_c , F_t , and chip thicknesses).

ans.

$$R = \frac{W_f}{W_c} = \frac{\left(\frac{F V_f}{33000}\right)}{\left(\frac{F_c V_c}{33000}\right)} = \frac{F V_f}{F_c V_c} = \frac{(F_t \cos \alpha + F_c \sin \alpha) \left(\frac{V_c \sin \phi}{\cos(\phi - \alpha)}\right)}{F_c V_c}$$

$$R = \frac{(F_t \cos \alpha + F_c \sin \alpha) (\sin \phi)}{F_c \cos(\phi - \alpha)} = \frac{(F_t \cos \alpha + F_c \sin \alpha)}{F_c} \left(\frac{\sin \phi}{\cos(\phi - \alpha)}\right)$$

$$R = \frac{(F_t \cos \alpha + F_c \sin \alpha) \left(\frac{t_1}{t_2}\right)}{F_c}$$

12. A new lathe tool is to be used on cast iron work with a 6" diameter to make a 5" long rough cut in 3 passes. The operation conditions listed below were provided by the supplier or

assumed. Calculate the parameters a) to e) as requested.

Cutting Speed = 300 fpm
Feed Rate = 0.008 ipr
Depth of Cut = 0.125"
Idle Horse Power = 0.25
Machine Efficiency = 0.90

- a) Spindle RPM
- b) Time to make the cut (min.)
- c) Metal Removal Rate Q (in.³/min.)
- d) Cutting Horse Power (HP_c)
- e) Minimum Machine Tool Motor HP.

13. Which of these statement is most correct?

- a) a continuous chip with built up edge may result when we try to cut brittle metals.
- b) a continuous chip will result when cutting very strong work materials.
- c) a discontinuous chip will result when we use heavy feeds and speeds.
- d) all of the above.

ans. C

14. One of the assumptions behind calculating orthogonal cutting forces is,

- a) that the rake angle is positive.
- b) that the tool is only cutting with one edge and one point.
- c) the shear plane is a function of before and after chip thicknesses.
- d) none of the above.

ans. C

15. Which of these statements is most correct?

- a) the cutting pressure drops as cutting velocity decreases.
- b) power required to cut each cubic inch drops as cutting velocity increases.
- c) we can use the quantity of metal removed by itself to estimate the required horsepower of a machine tool.
- d) all of the above.

ans. B

16. A new lathe tool is to be used on cast iron work with a 6" diameter to make a 36" long rough cut in 4 passes. The operation conditions listed below were provided by the supplier or assumed. Calculate the parameters a) to e) as requested.

Cutting Speed = 200 fpm
Feed Rate = 0.010 ipr
Depth of Cut = 0.100"
Idle Horse Power = 0.25
Machine Efficiency = 0.90

- a) Spindle RPM
 b) Time to make the cut (min.)
 c) Metal Removal Rate Q (in.³/min.)
 d) Cutting Horse Power (HP_c)
 e) Minimum Machine Tool Motor Horse Power.

ans. a) 127rpm, b) 113min., c) 2.4 ipm, d) 1.23 or 3.94HP, e) 1.62 or 4.63HP

$$D = 6\text{in} \quad CS = 200 \frac{\text{ft}}{\text{min}} \quad f = 0.01\text{ipr} \quad d = 0.1\text{in} \quad HP_I = 0.25 \quad e = 0.9$$

$$\text{a) } rpm = \frac{CS}{\pi d} = \frac{200 \frac{\text{ft}}{\text{min}}}{\pi(6\text{in})} = \frac{200(12)\text{in}}{\pi(6)\text{in min}} = 127rpm$$

$$\text{b) } T = \frac{L}{f(\text{rpm})} = \frac{36\text{in}}{0.01\text{ipr}(127rpm)} = 28.35\text{min} \quad (\text{for one pass})$$

$$\text{c) } Q = 12fdCS = 12(0.01)(0.1)(200) = 2.4 \frac{\text{in}^3}{\text{min}}$$

$$\text{d) } HP_c = HP_u Q = \left(\frac{0.5 + 1.6}{2} \right) 2.4 = 2.5\text{HP}$$

$$\text{e) } HP_M = HP_I + \frac{HP_C}{e} = 0.25 + \frac{2.5}{0.9} = 3.0\text{HP}$$

17. a) Define machinability. b) What determines the machinability of a metal?

20. What factors will affect surface finish?

21. Sketch a single edge cutting tool and label the a) face, b) flank, c) nose, d) cutting edge, e) relief, f) shank.

22. Why is the cutting speed important? What will happen at different cutting speeds, from very slow to very fast?

23. We have set up a lathe and are doing an orthogonal cut. The feed rate of the lathe is 0.1mm, and the chip thickness after the cut is 0.2mm. The depth of the chip being cut is 5mm. The surface cutting speed of the tool is 2m/s. The tool has a rake angle of 10deg. The tangential force is measured as 200N, and the cutting force is 500N. a) Calculate the shear force and velocity. b) Calculate the total energy produced in the cut, c) Calculate the energy used to shear d) Explain the difference between the total and the shear energy. [based on Kalpakjian]

ans.

Given,

$$t_1 = 0.1 \text{ mm} \quad \alpha = 10^\circ \quad V_c = 2 \frac{\text{m}}{\text{s}} \quad F_c = 500 \text{ N}$$

$$t_2 = 0.2 \text{ mm} \quad \text{depth} = 5 \text{ mm} \quad F_t = 200 \text{ N}$$

Find the total power and shear power.

$$W_c = F_c V_c = (500 \text{ N}) \left(2 \frac{\text{m}}{\text{s}} \right) = 1000 \text{ W} \left(\frac{1 \text{ HP}}{746 \text{ W}} \right) = 1.34 \text{ HP}$$

$$r_c = \frac{t_1}{t_2} = \frac{0.1}{0.2} = 0.5$$

$$\phi = \text{atan} \left(\frac{r_c \cos \alpha}{1 - r_c \sin \alpha} \right) = 28.3^\circ$$

$$F_S = F_c \cos \phi - F_t \sin \phi = 345 \text{ N}$$

$$V_S = \frac{V_c \cos \alpha}{\cos(\phi - \alpha)} = 2.07 \frac{\text{m}}{\text{s}}$$

$$W_S = F_S V_S = 714 \text{ W} \left(\frac{1 \text{ HP}}{746 \text{ W}} \right) = 0.96 \text{ HP}$$

Finally the ratio between the cutting power and the shear power

$$\frac{W_S}{W_c} = \frac{0.96}{1.34} = 0.71$$

24. How is machining different than other processes?
25. What is the difference between a roughing and finishing operation? How does this affect the workpiece and the power consumed?
26. What type of chip is expected at higher cutting speeds?
27. Does the friction power in cutting increase more with a feed or speed increase?
28. Why does cost typically increase for finishing operations.
29. Explain the correction factor 'c' used with the HPU values.
(ans. the HPU values are not linear, and 'c' corrects for these non-linear values)