Olin College

Training Document for Intermediate Lathe

Last Updated: 10/3/17

TABLE OF CONTENTS

TRAINING GOALS

The goal of this reading and the accompanying training is to ensure your safety in the operation of the lathe. The training consists of this background reading, a quiz about the reading, a training session with an instructor, and a practice piece, which will be begun, and potentially finished, during the training session. The instructor led portion is done in groups of three, in one 2 hour session, and should be scheduled when the background reading and the quiz have been completed. The primary goal of this training is “hands-on” proficiency. If you have been trained prior to Olin at a professional machine shop, school, job, etc., you still need to take this training. This ensures that everybody in the Olin shops understands our safety and operating guidelines.

MACHINE FEATURES

The following operations are taught in this training:

Facing: Removing material from the face of the stock with the side of a cutting tool. This is the first step in making sure that the stock is square with your lathe.

Turning: In this process, you will create round parts by cutting away unwanted material from the diameter of your stock.

Drilling: Creating a hole using a drill bit, creating cylindrical pockets or holes the size of the drill bit.

Tapping and Threading: Removing material from the walls of a piece in the pattern of a bolt’s threads, used to create a bolt hole.

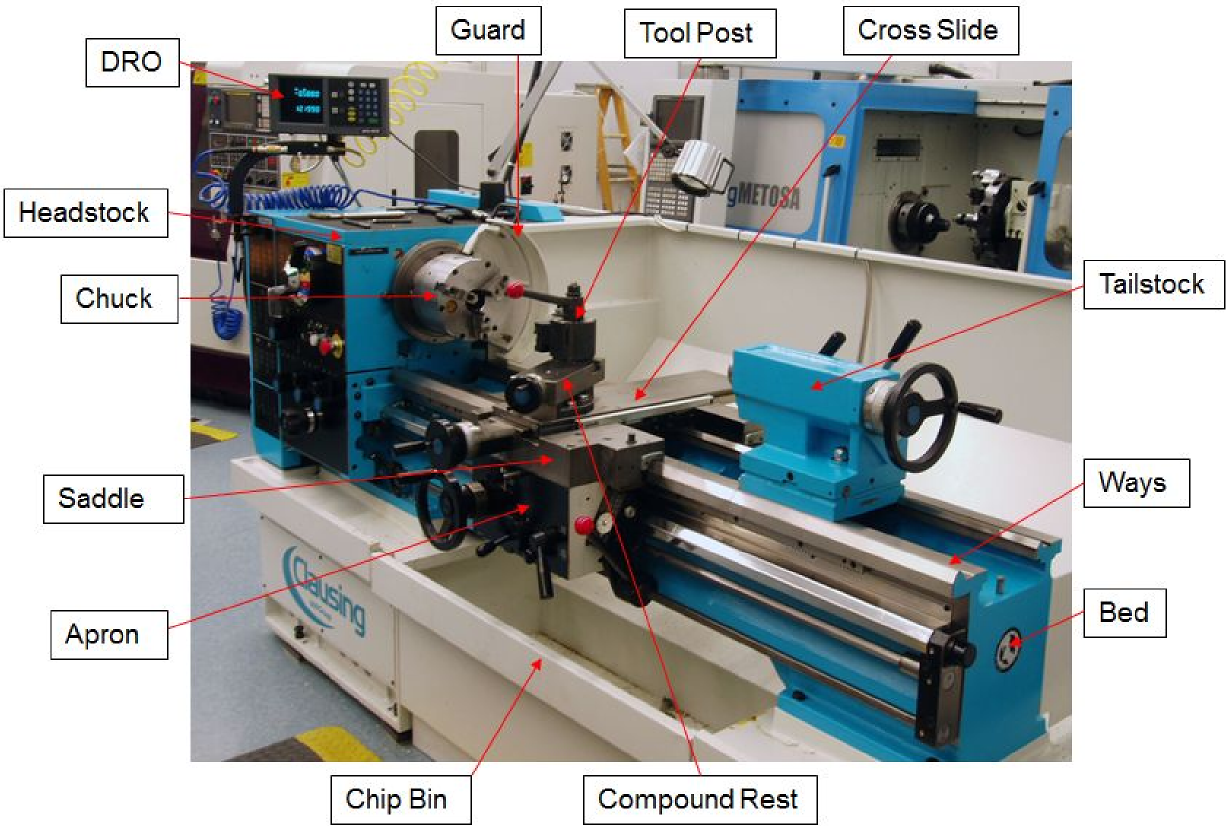
Parting and Grooving: Cutting grooves in the outside diameter of a part, or parting a finished piece from the rest of the stock.

BASIC CAPABILITIES

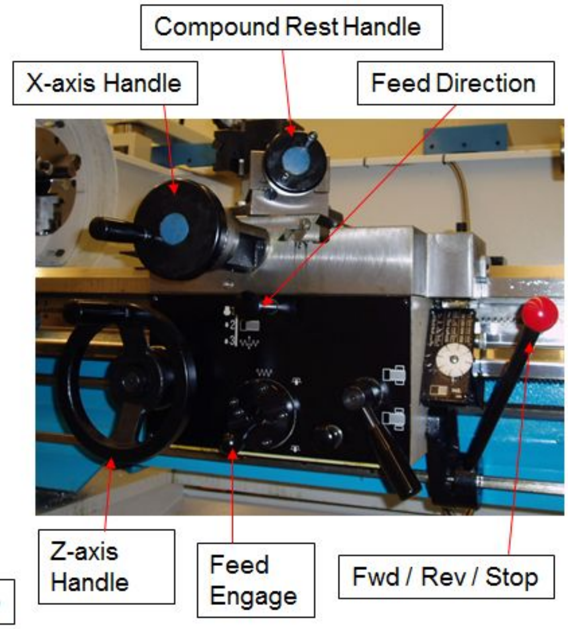
Lathes operate by spinning the work against the edge of a cutting tool. They are usually used to turn--machine--cylindrical parts, but can also produce many unique and irregular shapes. The cutting tool can be moved lengthwise along the lathe bed and across the work at any desired angle. Lathes can also be used for drilling, reaming, turning, boring, knurling, cutting, and shaping. While all operations performed on a lathe will result in features that have a circular or cylindrical nature, the workpiece itself is not always round. In the Olin Machine Shop we use manual lathes (also known as “engine” lathes). Although there are several other types of lathes, this document will focus only on the manual lathe.

In turning operations, the workpiece is held in a collet or chuck and a cutting tool is held in a tool post. The lathe is turned on and the workpiece begins to rotate. The cutting tool is then brought to the rotating workpiece to remove material.

BASIC MACHINE PARTS



CHUCK	For holding work pieces in the spindle of the lathe.
TOOL POST	For holding and quickly changing between different Tool Holders.
TAILSTOCK	For holding drill chucks or lathe centers
APRON	For moving the tool post and cross slide toward or away from the chuck.
TOOL HOLDER	For holding lathe tools
CROSS SLIDE	For cutting at a right angle to the main spindle axis
BED	Connects to the headstock and supports traveling carriage and tailstock
HEADSTOCK	Houses the spindle, spindle speed gears, and tool power feed gears



EMERGENCY STOP SWITCH	For stopping all automated movement of the lathe in case of an emergency
RPM SELECTION	For adjusting the speed of the lathe spindle.
FEED DIRECTION	To determine the direction of the carriage (Z direction) or cross slide power feed (X direction).
FEED ENGAGE	For turning the power feed for the carriage or cross slide on and off.
FEED RATE	To determine the speed of the carriage
FWD/REV/STOP LEVER	For turning the spindle on or off and for setting the rotation direction
COMPOUND REST HANDWHEEL	For manually feeding cutting tools at an angle to the spindle axis.
CROSS SLIDE (X-axis) HANDLE	For manually feeding cutting tools across the spindle (the X axis).
CARRIAGE (Z-axis) HANDLE	For manually feeding cutting tools in line with the spindle (the Z axis).

SAFETY RULES

1. Never wear loose sleeves, gloves, or any jewelry, and always tie your hair back.
2. Keep ALL rags away from the machine while it is in motion.
3. Never use the lathe when tired or rushed for time!
4. ALWAYS remove the chuck key from chuck immediately after using!
5. Make sure your part is securely tightened in the chuck or collet.
6. Always check that the chuck or collet will clear the tool post before you start.
7. Move the tool bit, tool post and/or tailstock a safe distance from the chuck or collet when inserting or removing your workpiece.
8. Don't run the machine faster than the proper cutting speed.
9. In setting up the tool holder, place it to the left most side of the tool post to prevent the compound slide from running into the chuck or spindle attachments.
10. Always make sure that the tool bit is sharp, at the correct height, and has the proper clearance.
11. If work is being turned between centers, make sure that the proper adjustments are made between the centers and that the tailstock is locked in place.
12. Do not grasp or touch chips or turnings with your fingers. Turn off the lathe before clearing chips with a brush or soft air blasts.
13. Set the tool bit on the centerline of the work to prevent the work from climbing over tool or cutting above the center and dragging.
14. Don't cut work completely through when turning between centers.
15. Stop the machine before taking ANY measurements.
16. <u>Leave the entire machine CLEAN when you are done using it!</u>
17. Be safe, ask if unsure, use your common sense, and look out for the safety of others.

OPERATING THE MACHINE

1. Machine Orientation

If you are uncertain and can't remember how the lathe works, please ask one of the shop staff and we will gladly show you. You should be comfortable with any material covered in the fundamental lathe training.

2. Cutting Parameters for the Lathe

Although we perform many operations with the lathe, turning is the most common. Turning refers to rotating a workpiece and reducing its diameter with a cutting tool. The lathe rotates the workpiece at a speed set by the operator. The carriage feeds the cutting tool into the workpiece at a specified rate for each revolution of the spindle. Factors determining spindle speed and feed rate include: the workpiece material, the cutting tool material, and the operation being performed.

Cutting speed is the speed at which a point on the workpiece moves past the cutting tool. If the cutting speed is too fast, the tool loses its edge too quickly. The hardness of the workpiece material and the hardness of the cutting tool both affect the cutting speed. The harder the workpiece material, the lower the cutting speed. The harder the cutting tool, the higher the cutting speed. If the spindle speed remains constant, the cutting speed decreases as the workpiece gets smaller. Therefore, the spindle speed must be set higher for smaller work pieces to maintain the correct cutting speed.

Do not confuse the spindle speed, in rotations per minute, or rpm, with the cutting speed. The spindle speed is set by the operator to achieve the proper cutting speed. During facing, set the spindle speed according to the outside diameter, even though the actual cutting speed will be too slow as you approach the middle of the workpiece. Use a slower cutting speed for roughing cuts and a higher speed for finishing cuts. By beginning at a lower spindle speed, you can speed up as the work allows.

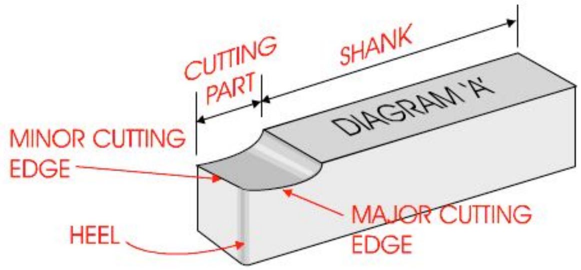
The *feed rate* is the speed at which the tool is fed into the workpiece. It is measured in millimetres or inches per revolution. Feed rates are quite variable-- much depends on the rigidity of the cutting tool and workpiece. Because surface finish does not matter when rough turning, as the main consideration is removing metal as efficiently as possible, higher feed rates are permissible. A high feed rate removes metal quickly, but it also shortens the life of the tool. The most efficient feed rate balances the metal removal rate with optimum tool life. The most efficient feed rate falls somewhere between the two extremes for the work material and tool material.

If you hear a noticeable squeal coming from the part you are turning, there are several ways to remedy this. First, check if you are using dull tool bit. You may also be turning too fast, or your tool may be set too high above center. Improper tool settings can dull tools.

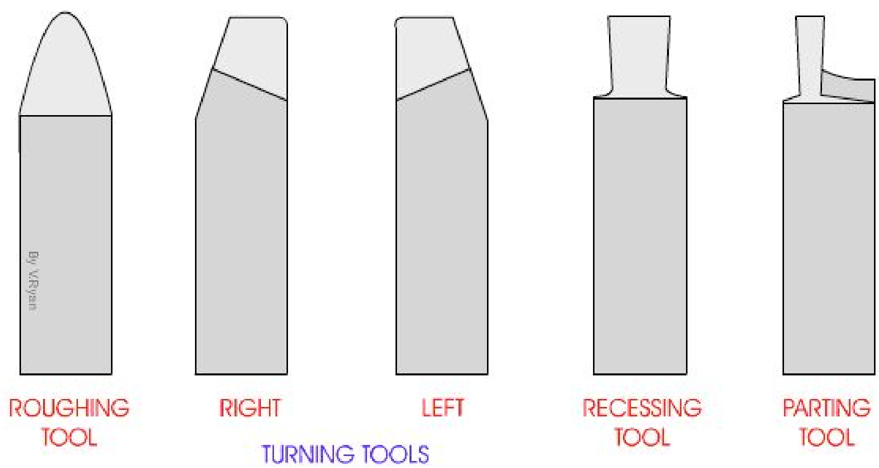
Adapted from: http://www.oldshigh.ca/documents/homework/Lathe%20Operations.pdf

3. Cutting Tools and Toolholders

The cutting tool is held in contact with the rotating work in order to remove material. While most applications used to be performed using a single-point cutting tool, see below, the Olin shops use indexable insert cutting tools. The actual cutter geometry varies with the type of work to be done. The standard cutting tool shape is shown below.



Indexable insert cutting tools are widely used and have inserts in a large variety of shapes and sizes. These inserts attach to special toolholders. As an edge dulls, the next edge is rotated into position until all edges are dulled. The shape of the tool depends on the type of work, roughing or finishing, and on the metal to be machined. Most cutter bits are ground to cut in one direction (left or right). Some cutting tools used for general lathe operations are shown below.



Facing tools are ground to provide clearance with a center. Roughing tools have a small side relief angle. which leaves more material to support the cutting edge during deep cuts. Finishing tools have a more rounded nose to provide a finer finish--round nose tools are generally reserved for lighter turning. They have no back or side rake, which permits cutting in either direction. Left hand cutting tools are designed to cut best when traveling from left to right.

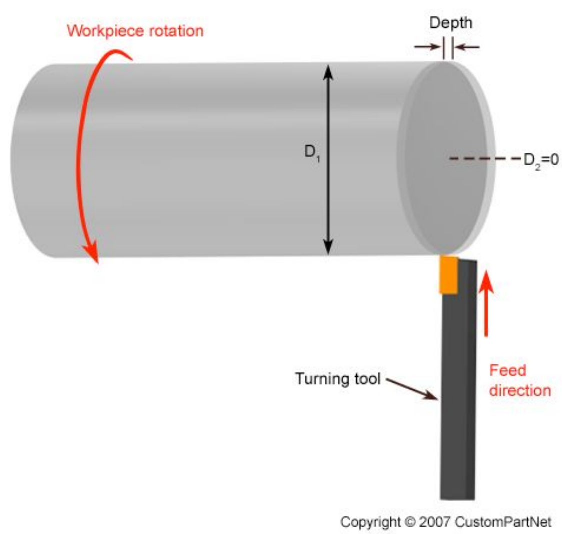
Workholding

Many different devices, such as chucks and collets, are used to hold and drive the work while it is being machined on a lathe. The size and type of work to be machined, and the operation being performed, will determine which work holding device is best for any particular job. A three-jaw chuck can be used for gripping cylindrical workpieces when the operations to be performed are such that the machined surface is concentric with the work surfaces. The advantage of the universal scroll chuck is its ease of operation in centering work for concentric turning.

Collets are used when smooth round stock, or workpieces that have been machined to a given diameter, must be held more accurately than normally can be achieved in a regular three or four jaw chuck. When the collet is pulled inward into the spindle, by means of the draw bar that engages threads on the inside of the collet, the action of the two mating tapers squeezes the collet segments together, causing them to tightly grip the workpiece.

4. Facing Operations

A facing operation consists of machining the end of the work square, and often reducing it to a specific length. Facing is often the first operation performed in order to clean up the face of the workpiece. It is standard machining practice to cut stock slightly longer than needed. The facing tool is ground to prevent interference with the tailstock center. The tool point is set at a slight angle to the work face with the point leading slightly. Avoid excessive tool holder and tool bit overhang when setting up the facing operation.



http://www.custompartnet.com/wu/images/turning/facing-bar.png

Calculating Cutting Speeds

Cutting speed (also called *surface speed* or simply *speed*) is the speed difference between the cutting tool and the surface of the workpiece it is operating on. In other words, the cutting speed is the speed at the periphery of the cutter as the work is rotating. In the imperial system, the cutting speed is notated in terms of feet per minute, fpm, or surface feet per minute, sfpm. In the metric system the cutting speed is notated in terms of meters per minute, or m/min. The cutting speed depends primarily on the work material and its hardness. In general, an increase in the hardness of a material reduces the speed at which it can be cut. The cutting speed is also influenced by the feed rate, and to a lesser extent by the depth of cut. Heavier cuts require a slower cutting speed than lighter cuts. Higher feed rates require higher cutting speeds.

The *spindle speed* is the rotational speed of the spindle of the machine, measured in revolutions per minute (RPM). The preferred spindle speed is determined by working backward from the desired cutting speed (sfm or m/min) and incorporating the diameter of the workpiece. The spindle speed can be calculated:

$$N = \frac{4CS}{D}$$

Where N is in Revolutions per minute, CS is the cutter speed in SFM, and D is work diameter in inches.

5. Turning Operations

Deflection of the workpiece, the cutting tool, and the machine is always present and can never be completely eliminated. This deflection is usually so minimal that it has no influence on an operation, and often goes unnoticed. The deflection only becomes a problem when it results in chatter, vibration, or distortion. The workpiece should not extend more than 2-3 times its diameter from the chuck jaws. The workpiece must be relatively short compared to its diameter so that it is stiff enough that we can safely turn it in the three jaw chuck without supporting the free end of the work.

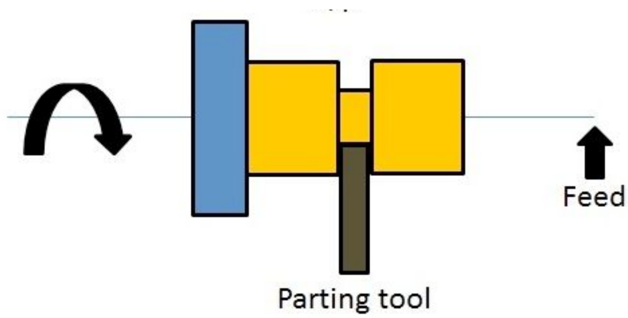
The roughing cut should be as heavy as the machine and tool bit can withstand. The finishing cut should be light and made to cut to the specified dimension in just one pass of the tool bit. When using the power feed to machine to a specific length, always disengage the feed approximately 1/16-inch away from the desired length dimension and finish the cut by hand.

6. Parting and Grooving Operations

In parting operations the workpiece rotates while the tool carries out a radial feed movement. As with face turning, the tool is fed from the periphery of the workpiece toward the center and the cutting speed is reduced to zero. As the diameter of the workpiece is reduced, the radial cutting force will cause the material to break before the insert has cut through it [Ref: American Machinist].

A parting tool is deeper and narrower than a turning tool. It is designed for making narrow grooves and for cutting off parts. When a parting tool is installed, ensure that it hangs over the tool holder enough that the the holder will clear the workpiece (but no more than that). Ensure that the parting tool is perpendicular to the axis of rotation and that the tip is the same height as the center of the part. A good way to do this is to hold the tool against the face of the part. Set the height of the tool, lay it flat against the face of the part, then lock the tool in place. When the cut is deep, the side of the part can rub against sides of the groove, so it's especially important to apply cutting fluid.

Grooving tools are similar to parting tools, but are generally narrower and less deep. Grooving tools come in different sizes, but all operate just like parting tools: the tool is fed, perpendicular to the axis of the work, directly toward the workpiece. Once the desired groove depth is reached, the tool is retracted the same way.



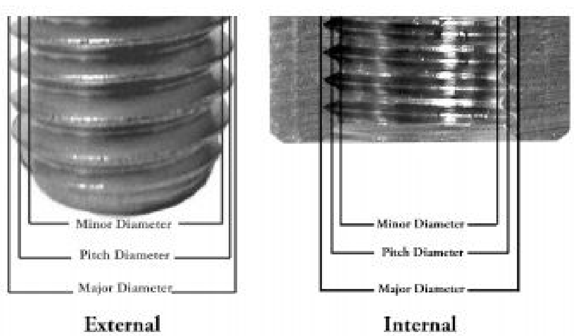
To part off your work, feed the tool bit into the rotating work with the cross slide until the tool completely severs the work. Speeds for parting should be about half that used for straight turning. Feeds should be light but continuous. If chatter occurs, decrease the feed and speed, and check, with the lathe off, for loose lathe parts or a loose setup. Always have the carriage locked down to the bed to reduce vibration and chatter. Never try to catch the cutoff part with your hand; it will be hot and could burn you.

Some important hints for applying grooving and parting tools:

- Always use plenty of cutting fluid.
- Set the center-height of the cutting edge accurately.
- Make sure toolholder/blade is accurately positioned at 90 degrees to the workpiece axis.
- Use toolholder with the shortest possible length of insertion for the operation in question.
- Adapt the cutting speed to avoid vibrations.
- Reduce the feedrate for the final part when parting-off bar material/components.

7. Drilling and Tapping/Threading Operations

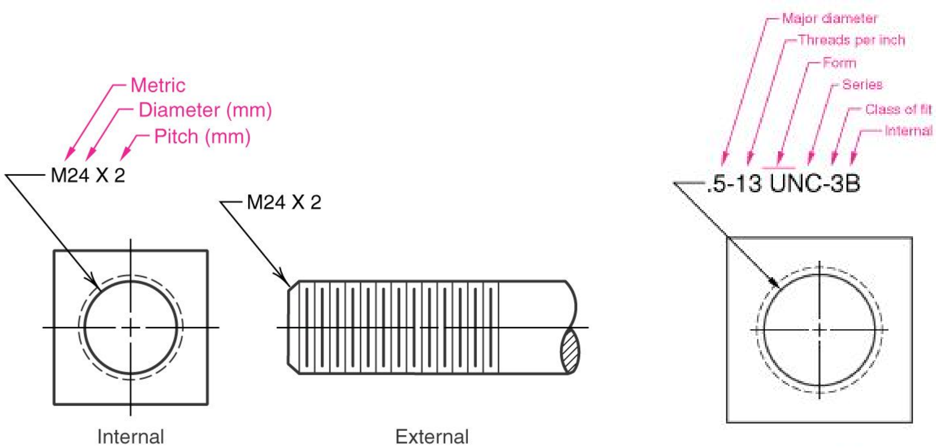
When a hole is to be cut in solid stock, the usual practice is to hold it in a suitable chuck and mount the drill in the tailstock. Drilling as accomplished on a lathe by feeding the stationary drill into the rotating workpiece. The tailstock center keeps the drill aligned and enables it to be fed into the material by the tailstock wheel. When drilling and tapping, it is crucial to use oil. It keeps the bits from squealing, makes the cut smoother, cleans out the chips, and keeps the drill and stock from overheating. Ample clearance must be provided in the back of the work so that the drill will not strike the chuck or headstock spindle when it breaks through. Use a center drill to start the hole; this will ensure that the drill will cut exactly on center. Holes larger than ¼” require drilling a pilot hole. Peck drilling is often recommended.



You can cut internal threads on a lathe using a tap, and external threads using a die or the lead screw of the lathe. The lead screw links the power feed of the carriage to the rotation of the chuck. This allows you to turn a long, consistent section by setting your cutting depth, and then letting the lead screw move the carriage down the length of the work. Because the ratio of RPM between the chuck and lead screw can be adjusted, it also allows you to cut threads of varying pitch. In this training, we will teach you how to use taps and dies to make threads, rather than the power feed.

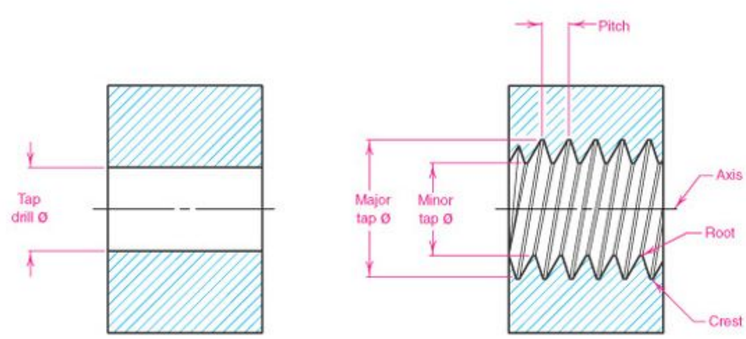
Understanding Thread Type And Sizes

Machine Screw sizes range from No. 0 through No. 14; No. 0 has a .0600" outside diameter; No. 1, .0730"; No. 2, .0860, etc.--all in .0130 increments. Fractional Taps are designated in fractional and integral inch sizes from 1/4" upwards. NC or UNC means coarse thread. NF or UNF means fine thread. NS means special thread.



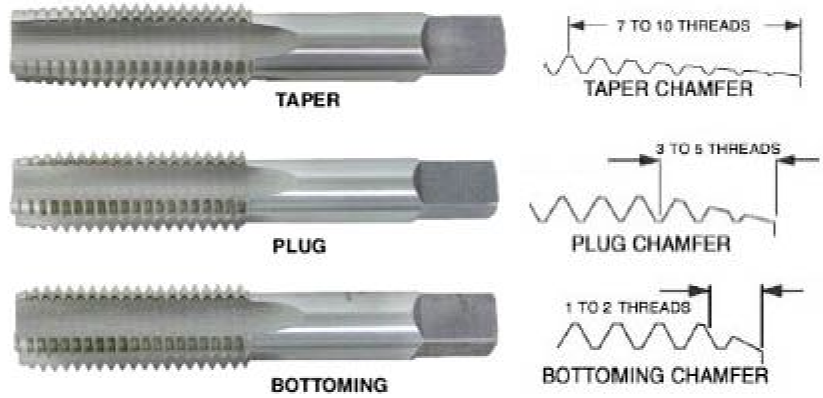
Internal Threads

The size of a tap is the outside diameter of its threads; therefore, theoretically, the hole drilled for tapping could be smaller than the tap by twice the depth of the thread if a full thread is cut. The shape of the thread partly determines the amount to be subtracted from the tap diameter. Do not use a tap drill that is too small as this will cause tapping troubles. A drilled hole should be of sufficient diameter to produce a thread depth of approximately 75%.



Internal threads

The most common hand taps are called taper, plug, and bottoming taps. All three tools are identical except for the bevel angle at the tip. The bevel at the tip serves two purposes: it guides the tap into the hole and ramp cuts the undeveloped first threads.



A bottoming tap is used to cut threads in a hole that has already been partially threaded using one of the more tapered types of tap; the tapered end of a bottoming tap is too short to successfully start into an unthreaded hole. However, the short taper of a bottoming tap enables it to cut threads down to the bottom of a blind hole (a hole that does not go completely through a piece). Taper taps have a very gradual cutting action that is less aggressive than that of the plug tap.

Determining Tap Drill Bit Size

The simplest way of determining the proper drill-tap combination is using a reference chart. You can, however, also calculate proper tap-drill diameters using this formula:

Inch Cutting Taps

$$Drill\ Size\ =\ Basic\ Major\ Diameter\ of\ Thread\ -\ \frac{(0.0120)(\%\ of\ Full\ Thread)}{Threads\ Per\ Inch}$$

Example: To Find the Tap Drill Size for a 1/4-20 Tap with 75% Thread:

$$Drill\ Size\ =\ .250\ -\frac{(0.0130\ x\ 75)}{20}\ =\ .201"\ =\ \#7\ Drill$$

Metric Cutting Taps

$$Tap\ Drill\ Size\ =\ Basic\ Major\ Diameter\ of\ Thread\ -\frac{(\%\ of\ Full\ Thread)\ (Pitch\ in\ Millimeters)}{76.98}$$

Example: To Find the Tap Drill Size for a M6 x 1.0 Tap with 70% Thread:

$$Drill\ Size\ =\ 6.0\ -\ \frac{(70)(\ 1.0\ )}{76.98}\ =\ 5.09067\ =\ 5.1mm\ Drill$$

External Threads

External threads, such as for bolts and studs, are made by using a die. Dies cut threads on the outside of specifically sized rods. Just as you must begin tapping with a hole sized to the tap, so too must you begin threading with a rod sized to the die you wish to use. The rod should be slightly smaller than the dye, to aid in chip removal and eliminate any sharp edges on the threads. Threading follows a similar procedure to that of tapping, but it is more difficult to start the thread cutting process. This difficulty is due to the lack of starting taper in a die. Using a ‘stopped’ spindle on a lathe makes starting the thread significantly easier.

You can use the drill chuck to support the die as you begin to thread. After the thread is begun, the drill chuck is removed and the die handle can then be turned by hand. You will need to use the Machinery’s Handbook to determine the size of the shaft before threading. In the training, we will show you how to read the charts.

In order for taps and dies to cut, they must be harder than the materials they are cutting. This additional hardness also makes them brittle, meaning, they can be easily broken. Lubricating the cutting threads reduces friction binding and aids in chip removal. You may also find that you must retract from the hole or shaft often in order to clear out the chips.

Table 3. *(Continued)* **Standard Series and Selected Combinations — Unified Screw Threads**

Nominal Size, Threads per Inch, and Series Designation[a]	External[b]								Internal[b]					
	Class	Allowance	Major Diameter			Pitch Diameter		UNR Minor Dia.,[c] Max (Ref.)	Class	Minor Diameter		Pitch Diameter		Major Diameter
			Max[d]	Min	Min[e]	Max[d]	Min			Min	Max	Min	Max	Min
10–24 UNC	2A	0.0010	0.1890	0.1818	—	0.1619	0.1586	0.1394	2B	0.145	0.156	0.1629	0.1672	0.1900
	3A	0.0000	0.1900	0.1828	—	0.1629	0.1604	0.1404	3B	0.1450	0.1555	0.1629	0.1661	0.1900
10–28 UNS	2A	0.0010	0.1890	0.1825	—	0.1658	0.1625	0.1464	2B	0.151	0.160	0.1668	0.1711	0.1900
10–32 UNF	2A	0.0009	0.1891	0.1831	—	0.1688	0.1658	0.1519	2B	0.156	0.164	0.1697	0.1736	0.1900
	3A	0.0000	0.1900	0.1840	—	0.1697	0.1674	0.1528	3B	0.1560	0.1641	0.1697	0.1726	0.1900
10–36 UNS	2A	0.0009	0.1891	0.1836	—	0.1711	0.1681	0.1560	2B	0.160	0.166	0.1720	0.1759	0.1900
10–40 UNS	2A	0.0009	0.1891	0.1840	—	0.1729	0.1700	0.1592	2B	0.163	0.169	0.1738	0.1775	0.1900
10–48 UNS	2A	0.0008	0.1892	0.1847	—	0.1757	0.1731	0.1644	2B	0.167	0.172	0.1765	0.1799	0.1900
10–56 UNS	2A	0.0007	0.1893	0.1852	—	0.1777	0.1752	0.1681	2B	0.171	0.175	0.1784	0.1816	0.1900
12–24 UNC	2A	0.0010	0.2150	0.2078	—	0.1879	0.1845	0.1654	2B	0.171	0.181	0.1889	0.1933	0.2160
	3A	0.0000	0.2160	0.2088	—	0.1889	0.1863	0.1664	3B	0.1710	0.1807	0.1889	0.1922	0.2160
12–28 UNF	2A	0.0010	0.2150	0.2085	—	0.1918	0.1886	0.1724	2B	0.177	0.186	0.1928	0.1970	0.2160
	3A	0.0000	0.2160	0.2095	—	0.1928	0.1904	0.1734	3B	0.1770	0.1857	0.1928	0.1959	0.2160
12–32 UNEF	2A	0.0009	0.2151	0.2091	—	0.1948	0.1917	0.1779	2B	0.182	0.190	0.1957	0.1998	0.2160
	3A	0.0000	0.2160	0.2100	—	0.1957	0.1933	0.1788	3B	0.1820	0.1895	0.1957	0.1988	0.2160
12–36 UNS	2A	0.0009	0.2151	0.2096	—	0.1971	0.1941	0.1821	2B	0.186	0.192	0.1980	0.2019	0.2160
12–40 UNS	2A	0.0009	0.2151	0.2100	—	0.1989	0.1960	0.1835	2B	0.189	0.195	0.1998	0.2035	0.2160
12–48 UNS	2A	0.0008	0.2152	0.2107	—	0.2017	0.1991	0.1904	2B	0.193	0.198	0.2025	0.2059	0.2160
12–56 UNS	2A	0.0007	0.2153	0.2112	—	0.2037	0.2012	0.1941	2B	0.197	0.201	0.2044	0.2076	0.2160
1/4–20 UNC	1A	0.0011	0.2489	0.2367	—	0.2164	0.2108	0.1894	1B	0.196	0.207	0.2175	0.2248	0.2500
	2A	0.0011	0.2489	0.2408	0.2367	0.2164	0.2127	0.1894	2B	0.196	0.207	0.2175	0.2224	0.2500
	3A	0.0000	0.2500	0.2419	—	0.2175	0.2147	0.1905	3B	0.1960	0.2067	0.2175	0.2211	0.2500
1/4–24 UNS	2A	0.0011	0.2489	0.2417	—	0.2218	0.2181	0.1993	2B	0.205	0.215	0.2229	0.2277	0.2500
1/4–27 UNS	2A	0.0010	0.2490	0.2423	—	0.2249	0.2214	0.2049	2B	0.210	0.219	0.2259	0.2304	0.2500
1/4–28 UNF	1A	0.0010	0.2490	0.2392	—	0.2258	0.2208	0.2064	1B	0.211	0.220	0.2268	0.2333	0.2500
	2A	0.0010	0.2490	0.2425	—	0.2258	0.2225	0.2064	2B	0.211	0.220	0.2268	0.2311	0.2500
	3A	0.0000	0.2500	0.2435	—	0.2268	0.2243	0.2074	3B	0.2110	0.2190	0.2268	0.2300	0.2500
1/4–32 UNEF	2A	0.0010	0.2490	0.2430	—	0.2287	0.2255	0.2118	2B	0.216	0.224	0.2297	0.2339	0.2500
	3A	0.0000	0.2500	0.2440	—	0.2297	0.2273	0.2128	3B	0.2160	0.2229	0.2297	0.2328	0.2500

UNIFIED SCREW THREADS

1737

8. Cleaning the Machine/Proper Care Procedures

The lathe ways are precision ground surfaces and must not be used as tables for other tools. The ways must be kept clean of grit and dirt. The lead screw and gears should be checked frequently for any metal chips that could become lodged in the gearing mechanisms. Don't leave measuring or other precision tools on the table of the machine. They will get damaged or contaminated.

The entire machine must be cleaned after every use. If another user needs the machine immediately after you, you should still clean the machine. The process is simple and should not take more than 15 minutes. Make sure you are aware of the clock and leave enough time to finish clean-up. The procedure is:

A. Turn off the machine and remove the cutter and collet from the spindle.
B. Put away all your hand, set-up and cutting tools. If not sure where they go, ask the staff.
C. Use a brush to remove the chips from the vise, table, and ways.
D. Brush or vacuum the difficult to reach spots. Wipe the spindle, table, ways, covers, vise, etc…
E. Wipe off ALL cutting fluids and oils from the ENTIRE machine. Top to bottom, machine must be dry.
F. Sweep the floor and surrounding areas. Metal chips are to be placed in metal trash bin.
G. There should be NO visible chips of any size on the machine. Leave the machine clean for the next person.

9. Other Considerations to be Aware Of

A machine may be left set-up with your job ONLY if you will be returning at the next available opportunity. You should always clean the machine and all tools/cutters/supplies must be put away. Machine set-ups will be broken down after 4 hours or if there is an urgent need, unless other arrangements are made with a supervisor or shop manager. Make sure you leave a sign with your name, phone number and a time/date when you will be returning to use the machine.

- Never be afraid to ask for help and guidance, that is why the shop supervisor is there! When in doubt, always ask!
- These are very capable machines, but must be used correctly to avoid damage and accidents. Learn how to use them correctly, there are no shortcuts in quality and safety!
- Come prepared. Have your material/parts, complete (accurate) drawing(s), a plan of action, and a list of tools you will need.
- If you break any bits, end mills, or other tools, please let us know, so that we can replace it.
- Double check your setup before starting any operation. Check for tightness/rigidity, correct speeds/feeds, obstructions, clamps, etc.
- Re-read the safety rules, your life and health depends on it!

NOTES

APPENDIX

PRACTICE PIECE

Machine the following piece to complete your training.

Adapted from:
-UC Riverside Manual Toolroom Lathes SOP