UNIVERSITY OF WYOMING | College of Engineering and Applied Science Engineering Shop

EN Bldg Room # L65 - Dept. 3295 – (307) 766-4243 - www.uwyo.edu/ceas/shop/
1000 E. University Ave.
Laramie, WY 82071-2000

College of Engineering Machine Shop Facilities and Best Practices

1. OBJECTIVE

To provide an overview and basic knowledge of the University of Wyoming, College of Engineering, Machine Shop equipment, tools, terminology, and practices.

2. MEASURING TOOLS

The University of Wyoming, College of Engineering, Machine Shop uses the English measurement system, i.e. the machine tools, the measuring tools, etc... are in inches. Conversions must be made when working with metric units.

a. **RULERS AND SCALES** are typically made of steel and are used to make rough measurements. Rulers and scales are typically graduated in 1/8, 1/16, 1/32, and/or 1/64 inch divisions.

b. **SQUARES** are used for laying out lines on work pieces, for checking two surfaces for squareness.

c. **GAGES** are made for a large variety of purposes: screw pitch, radius, center, wire and sheet metal, etc...

d. **VERNIER CALIPERS AND MICROMETERS** are precision measuring tools generally used to measure to the thousandths or ten-thousandths of an inch.

   i. **VERNIER CALIPERS** are precision measuring tools (3 decimal places). Two types are available; vernier scale or dial scale. Six-inch range dial calipers are the most common, but 12, 20, 36, and 48-inch vernier calipers are also available. Vernier depth gages are another type of measuring instrument that are used to measure hole depths, slots, recesses, etc. They are available in 6 and 12-inch ranges.

   ii. **MICROMETERS** are another precision measuring tool. They commonly have 3 decimal place accuracy. Some 1-inch vernier micrometers will give 4 place readings. Micrometers are either inside or an outside variety and in a variety of sizes up to 6-inches. The measuring range for any particular sized micrometer is limited to 1 inch.

e. **DIAL INDICATORS** are used to check correct alignment of one machine component with another and to correctly position work pieces in machine tools prior to machining. A dial indicator shows the amount of error, in alignment for a part being positioned in a machine tool. Most inch dial indicators are calibrated in either thousandths of an inch (0.001 inch) or ten-thousandths of an inch (0.0001 inch).

3. OPERATIONS

a. DRILLING, COUNTER SINKING, REAMING, AND COUNTER BORING

i. **DRILLS** are used to cut holes through or into a material. The most common type of drill is the twist drill, which with either a straight shank or a tapered shank (Morse taper), the straight shank is for fitting into a drill chuck, the tapered shank fits into a tapered spindle. Drill sizes are designated one of three ways, fractional, number, and letter. The accuracy of drilled holes is expected to fall within the range given by the following formulas1 provided the drilling equipment is in good repair and the drill sharpening is reasonable:

Maximum Oversize = 0.005 + 0.005D
Minimum Oversize = 0.001 + 0.003D

Where D is the nominal drill diameter in inches. Expect drilled holes to be out of round.

ii. **CENTER DRILLS** are used for two main purposes: for providing a starting hole for drills or
for providing a center hole for stock to be held in a lathe.

iii. **COUNTER SINKS** are used to machine a coned shaped enlargement at the end of a hole. Many shapes and sizes are available, but most commonly used are 60 degree for lathe centers, and 82 degree for flat head machine screws. Countersinks are also used for deburring operations.

iv. **REAMERS** are multiple edged cutting tools used to enlarge and finish a drilled hole to exact dimensions and smoothness. Reamers are typically made of H.S.S. There are many types of reamers, hand, machine, taper, etc. Machine reamers are designed for use on a drill press, lathe, or milling machine. Commonly used straight reamers are available in three sizes; standard, over and under. e. g. a ½–inch standard reamer is 0.500-inch diameter, the over reamer is 0.501-inch diameter, and the under reamer is 0.499-inch diameter.

v. **COUNTER BORES** are used to spot face or counter bore to enlarge a hole to receive the head of hex head cap screws or fillister head machine screws.

vi. **DRILLING SPEEDS AND FEEDS** The use of proper cutting speeds and feeds promotes good tool life and machining efficiency. A cutting speed that is too high will cause overheating of the cutting tool and premature cutting edge failure. Too low of cutting speed will reduce productivity and increase tool wear.

EN Bldg Room # L65 - Dept. 3295 – (307) 766-4243 - www.uwyo.edu/ceas/shop/
1000 E. University Ave.
Laramie, WY 82071-2000

Cutting speed (cs), spindle speed, revolutions per minute (rpm) are interrelated terms. Cutting speeds are expressed in terms of feet per minute (fpm). Feed rate is expressed in terms inches per minute. Cutting speed is the distance a point on the circumference will travel in one minute.

Example: a ¼ inch diameter drill with a cutting speed of 80 fpm has a spindle speed of 1222 rpm. A 1-inch diameter drill with the same cutting speed of 80 fpm has a spindle speed of 305 rpm. The spindle speed rpm can be calculated with the following formula:

$$RPM=(CS)(12)/(DIA)(3.14).$$

Generally there is no one correct speed or feed for all materials or conditions. Speeds and feeds are adjusted up and down depending upon the conditions of the particular job setup and materials. Conditions may include such factors as kind of material, the cutter material, cutting fluids, size and rigidity of the machine, setup, quality of finish desired, etc.

vii. **DRILL PRESSES** are frequently used, important machine tools. Their principal use is to drill holes, other drilling operations include reaming, boring, counter-boring, counter sinking, and tapping. Drill presses are variable speed, and are hand feed (z direction) operations. The work is generally clamped to a table or held in a vice. Speeds on a drill press are adjusted in one of three ways; step pulleys, variable speed drive, or a gear drive system.

1. **DRILLING SPEEDS AND FEEDS** All drill presses are equipped with a hand feed lever or handle. Proper feed rate is controlled by the feel of the cutting action and by observing the chip. In soft steel a long stringy chip indicates the feed is too fast. With a proper feed the chip is a tightly rolled helix.

   Cutting speed tables for high speed steel (HSS) drills will show recommended cutting speed for various materials.

   The use of proper cutting speeds and feeds promotes good tool life and machining efficiency. A cutting speed that is too high will cause overheating of the cutting tool and premature cutting edge failure. Too low cutting speed will reduce productivity and increase tool wear.

   Cutting speed (cs), spindle speed, revolutions per minute (rpm) are interrelated terms. Cutting speeds are expressed in terms of feet per minute (fpm). Feed rate is expressed in terms inches per minute. Cutting speed is the distance a point on the circumference will travel in one minute.

EN Bldg Room # L65 - Dept. 3295 – (307) 766-4243 - www.uwyo.edu/ceas/shop/
1000 E. University Ave.
Laramie, WY 82071-2000

Example: a ¼ inch diameter drill with a cutting speed of 80 fpm has a spindle speed of 1222 rpm. A 1-inch diameter drill with the same cutting speed of 80 fpm has a spindle speed of 305 rpm. The spindle speed rpm can be calculated with the following formula:

$$RPM=(CS)(12)/(DIA)(3.14)$$

Generally there is no one correct speed or feed for all materials or conditions. Speeds and feeds are adjusted up and down depending upon the conditions of the particular job setup and materials. Conditions may include such factors as kind of material, the cutter material, cutting fluids, size and rigidity of the machine, setup, quality of finish desired, etc.

2. **SAFETY WITH DRILLING OPERATIONS**
    a. 1. Always wear appropriate safety goggles or glasses.
    b. Never leave the chuck key in the drill chuck or the drill drift in the spindle.
    c. Mount the work securely before drilling. Do not hold thin or small pieces in the hands.
    d. Remove chips with a brush. Never use the hands.
    e. Ease up on the feed pressure as the drill begins to break through a hole. This will prevent the drill from catching or breaking, or pulling the work loose.
    f. Do not attempt to stop the spindle with the hands after turning the machine off.
    g. Never drill plastics, PVC, copper alloys, including brass and bronze, with a drill that is ground for steel. Request a drill ground for these materials. A drill ground for steel may dig in, break, ruin your work, and or cause injury.
    h. Keep long sleeves, other loose clothing, and especially long hair away from the revolving spindle or belts.

b. THREADING, TAPPING, SINGLE POINTING

i. **THREADING** is cutting external threads by hand and is done with a die. A die can be fixed or adjustable.

ii. **TAPPING** is cutting internal threads and is done with a tap. Taps come in three styles: taper, plug, and bottoming. Tapping can be done on a drill press, milling machine, lathe, and by hand. Tap and tap drill selection: determine desired thread diameter and pitch, go to chart (fraction and machine screw tap drill sizes), and determine proper drill size. The chart will give recommended tap drill size for 75% full thread.

iii. **SINGLE POINTING** is cutting threads on the work piece in a lathe. They may be external or internal, straight or tapered, and right or left hand. This method is slower but essential for cutting larger diameter threads (1 inch plus), threads calling for a more accurate pitch or lead, or an unusual thread.

c. SAWING

i. **HORIZONTAL BAND SAWS** are probably the most commonly used piece of equipment in the shop. They have high cutting efficiency, variable blade speed (to select the best cutting speed for the material), and a variable feed rate. The work is held fixed in a vise and the blade is fed through the work.

ii. **VERTICAL BAND SAWS** may be the handiest machine in the shop. They have high cutting efficiency, variable blade speed, and a wide variety of tooth styles and numbers of teeth/ inch. The worktable can be tilted and the work is fed through by hand, allowing various shapes and contours to be cut.

iii. **SAFETY PRECAUTIONS FOR BAND SAWS**

a. Always wear appropriate safety goggles or glasses.
b. Always be sure that the saw guide is set to within 1/4 inch to 3/8 inch above the thickest portion of the work piece to be cut.
c. Support protruding ends of long pieces so they will not fall and cause injury.
d. Be sure that the saw blade is in good condition, and that the machine is set to the correct cutting speed.
e. When starting the saw, stand to one side of the saw frame; then adjust the speed as desired.
f. Avoid getting the hands to close to the blade.
g. Use a stick to remove short pieces of work from the area close to the blade.
h. Be careful in handling parts with sharp burrs on the ends.

iv. **CHOP SAWS & CIRCULAR SAWS**

1. Abrasive chop saws & carbide tooth chop saws are an alternative method of sawing materials available in the shop. These saws are typically used for cutting longer materials down to length & making end miter cuts to prep tubing pieces for welding.
2. Newer technology has yielded carbide toothed chop saws & circular saws that allow the user to cut numerous materials with more ease, lower cutting surface temperatures (less warpage than torch cuts) & less edge cleanup than the older style fiber-wheel abrasive chop saws.
3. Carbide toothed circular saws also run at a lower RPM (1750 RPM vs. typical 3500RPM), but allow the user to cut through sheets of metal (steel up to ½",

Aluminum up to 1”) and make other types of cuts in materials including hastelloys and high-nickel alloys, that conventional sawing equipment simply can’t.

d. TURNING

i. **TURNING OPERATIONS** In turning operations the work piece rotates against a fixed tool such as a turning tool, drill, reamer, tap, die, or tool post grinder. Turning operations are used to make cylindrical and conical parts. The principle turning machine is a lathe.

ii. **LATHES** can be used, for drilling, tapping, reaming, counter-boring, plus threading (internal and external) and taper turning. They have variable spindle speeds and are hand or power fed in the x (longitudinal) and Y (CROSS) directions. The work piece is held in a chuck or collet and rotated. The turning operation (cutting) occurs when the fixed turning tool is fed into the rotating work piece. The tooling may be fed manually or powered in either the x or y direction, the tool may also be manually fed at an angle by using the compound screw. The tailstock is used to center and support work in the headstock, the tailstock can also be used for longitudinal drilling operations. Lathe accessories and attachments include taper attachments, steady rest, follow rest, face plates, three and four jaw chucks, collet holders, etc. A lathe is capable of producing work accurate to three decimal places and 0.0004 of an inch or less.

iii. **TURNING SPEEDS AND FEEDS** The rotating speed of the work held in the headstock is
determined by the type of material, the diameter of the material, and the material of the turning tool. The feed rate is the rate at which the turning tool is fed past the work piece, and is determined by the material of the work piece, amount of material being removed and the desired surface finish.

Spindle speeds are determined by referencing the chart on the Machine Shop wall for recommended cutting speeds. Spindle speeds are adjusted by either a step pulley system with a high and low range or by a variable speed motor and two or three speed gearbox selection.

Powered tool feeds are gear drives from the spindle, so that the ratio of tool feed to spindle revolutions is constant. A wide selection of feed rates in ratio to the spindle revolutions is provided: i.e. 0.006-inch feed per spindle revolution up to ______. A quick-change gearbox changes the feed ratio. This gearbox also controls the ratios of spindle revolution to feed for single point thread cutting. The ratio of the cross feed to the longitudinal feed will vary for
different lathes, the cross feed may be equal to ½ times the longitudinal feed. The

feed rate selection is a matter of operator judgment and experience. Factors to consider are tool shape and material, depth of cut, rigidity of the work piece and the machine, work piece material, finish desired, etc…

1. **SAFETY PRECAUTIONS FOR LATHES**
   a. Wear approved safety goggles or glasses.
   b. See that all guards are in place.
   c. Before starting the lathe, turn the spindle by hand to insure that it turns freely. If the spindle is locked in a stationary position with the back-gear lever as well as the bull-gear pin, release one of these devices for the desired kind of drive and speed.
   d. Stop the machine for all measurements.
   e. Stop the machine to remove chips. Do not remove them with the hands---always use a brush---do not use compressed air.
   f. Stop the machine to make adjustments.
   g. Always stop the machine when adjusting the tool in the tool post.
   h. Keep the machine clear of long chips, rags and unnecessary hand tools.
   i. Use the right type of cutting tool for the job.
   j. Adjust the feed, speed, and depth of cut according to the size and type of material.
   k. Be sure that the tailstock and the tailstock spindle are securely locked.
   l. Always cut toward the head stock whenever possible.
   m. Before starting the power feed, make certain that the carriage has sufficient free travel to complete the cut without running into the lathe.
   n. Be sure that the work is mounted tightly in the chuck.
   o. Always remove the chuck wrench or key from the chuck immediately after using it.
   p. Turn the chuck one complete revolution by hand after the work is mounted to see that it clears the carriage and the ways.
   q. Never allow the cutting tool or tool holder to come into contact with the revolving chuck jaws.

e. MILLING

i. MILLING OPERATIONS use a revolving cutter to shape the work piece, and the work piece is generally moved in relation to the cutter, although drill press like operations are possible. A variety of milling operations can be performed depending on the type of machine, type of cutter, accessories, and attachments. Milled features include flat surfaces (horizontal, vertical, and angular), shoulders, grooves, slots, keyways, etc. CNC milling operations can produce complex geometric surfaces. Milling operations include milling, drilling, boring, reaming, tapping, counter boring, etc.

ii. **VERTICAL MILLING MACHINES** have a vertical spindle that holds the revolving cutter. The spindle (z axis) can be a hand feed or power feed operation (maximum of five inches travel). The spindles have variable speeds and those with power feed have variable feed rates. The work piece is held in a vise or clamped directly to the table that has three axis of travel (x, y, z). The table can be a hand or power operated. The work piece is moved relative to the cutter by using these table feed screws, which are equipped with collars graduated in thousandths of an inch. The accuracy of milling operations is then three decimal places ±0.0005 inch. Accessories and attachments include right angle heads, rotary tables, dividing heads, etc.

iii. **MILLING CUTTERS** are held in the spindle. Tooling used to hold the cutters in the spindle, include collets, arbors, and adapters. Milling cutters are made in many shapes and sizes for milling regular and irregular shapes. The most common type of cutter used in vertical milling machines is end milling cutters. End milling cutters have teeth on the circumference and on the end and may be used for horizontal, vertical, angular, or irregular surfaces. Common operations performed with end milling cutters are milling slots, keyways, pockets, shoulders, flat surfaces, etc. Various end milling cutters are:

1. **TWO FLUTE END MILLS** have only two teeth (flutes) on the circumference, the end teeth are designed so that they cut to the center, and therefore they can plunge cut. They are commonly used on aluminum, plastics, etc. They are commonly made of H.S.S., sizes range from 1/16 to 2 inches in diameter.

2. **FOUR FLUTE END MILLS** or multiple flute end mills have four or more teeth (flutes) on the circumference for cutting. Most are not center cutting, i.e. they will not plunge cut. They are used on steels and other tougher materials and are commonly made of H. S. S. Sizes range from 1/16 to 2 inches in diameter.

3. **BALL END MILLS** are again made of H. S. S. and come in two, three , and four flutes they're used in milling fillets, slots with radius bottoms, rounding pockets, etc.

4. **SHELL END MILLS** have multiple flutes, a hole in the middle, for screw mounting to short arbors. They're used for surface, face and end milling, etc.

5. **FLY CUTTERS** are single pointed cutting tool mounted in a cylindrical holder and are used for planing surfaces.

UNIVERSITY OF WYOMING | College of Engineering and Applied Science Engineering Shop

EN Bldg Room # L65 - Dept. 3295 – (307) 766-4243 - www.uwyo.edu/ceas/shop/
1000 E. University Ave.
Laramie, WY 82071-2000

6. **MISCELLANEOUS CUTTERS** include t-slot, woodruff key, corner rounding, concave, and convex cutters. These cutters are generally special purpose cutters for particular applications.

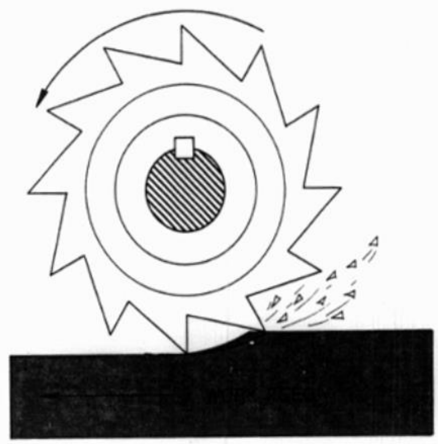

Fig. 80-10. Up milling or conventional milling.

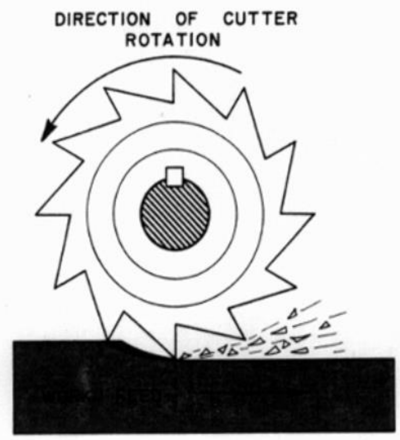

Fig. 80-11. Down milling or climb milling.

iv. **MILLING MACHINE SPEEDS AND FEEDS** Cutter speeds (fpm)(spindle speed) vary according to the diameter of the cutter, cutter material, the recommended cutting speed for the specific material, and the cutting fluid used. Feed rates vary by the type of cutter, number of cutter teeth, rigidity of the machine and setup, and rough or finish cut.

The type of cut is the method in which the work is fed past the milling cutter, the two methods of milling are conventional and climb milling shown in Figures 80-10 and 80-11. Conventional milling is most often used, since climb milling can easily damage the equipment and the work piece.

Spindle speeds for milling operations are calculated the same as for drilling operations, i.e. $rpm = (cs*12)/(\pi*dia)$. Recommended cutter speeds, cs, can be found in tables and on the charts in the Machine Shop. “dia” is the diameter of the cutter. Spindle speeds on a milling a milling machine are changed much the same as on a drill press using step-pulleys with a 2-speed box or a variable speed drive.

The feed rate is the rate at which the work is moved past the cutter. When the feed rate is too high the cutter may fracture or the machine or work piece may be damaged. To low a feed rate will dull the cutter more rapidly and the metal removal will be inefficient. Suggested feed rates per tooth, per revolution are generally found in a table.

Calculate the feed rate by using the following formula:

$$F=(R)(T)(RPM)$$

where F is feed rate in inches per minute, R is feed per tooth per revolutions in inches, T is the number of teeth, and rpm is the revolutions per minute of the cutter.

Feed rates on a milling machine are controlled in one of three ways: manual feed uses a hand wheel or crank and the rate of feed is a matter of judgment. With small cutters the rate of feed per cutter tooth should be less than for larger coarser-toothed cutters. Most machines will have a gearbox to drive the spindle in the z direction with a ratio of 1.5, 3, 6, thousandths of an inch per spindle revolution. Older machines will have a mechanical gearbox for the direct selection of feed rates. Newer machines will have an electric servo motor drive for an infinite range of feed rates.

v. **SAFETY PRECAUTIONS FOR MILLING MACHINES**

1. Wear approved safety goggles or glasses.
2. See that the table is clean and dry before mounting holding devices for the work.
3. Be sure that the bases of work-holding accessories or work pieces are wiped clean before they are fastened to the table.
4. Check to see that the work piece is mounted securely.
5. Select and mount the proper cutter, and see that it revolves in the proper direction.
6. Be sure that the arbor, cutter, and collars are clean before mounting them on the machine. Use a rag to handle sharp cutters.
7. Use only correct fitting wrenches on the machine.
8. Select the proper spindle speed, feed, and depth of cut. Make these adjustments only while the machine is stopped.
9. Use a lead or softheaded hammer to seat work pieces in the vise for setups.
10. Make certain that the table, holding device, and work piece will clear all machine tool parts during the cut.
11. Make sure that unauthorized persons are outside the safety zone of the machine when it is started.
12. Keep clear of the revolving cutter.
13. Use a brush for chip removal, do not use compressed air.
14. When finished, remove and return all tools and accessories. Brush the machine free of chips and wipe up all excess oil.

f. GRINDING, SANDING AND DEBURRING

i. **GRINDING, SANDING AND DEBURRING OPERATIONS** remove material from a work piece by abrasive action. This process is relatively slow, but can be very precise and can produce a very smooth surface finish. Variations of these operations are: surface grinding, abrasive cutoff, lapping, honing, etc.. The accuracy of these operations can be more than ±0.0001inches.

ii. SAFETY PRECAUTIONS FOR GRINDING, SANDING AND DEBURRING

1. Wear approved safety goggles or glasses.
2. Check to see that the rest for hand held work pieces is adjusted correctly near the grinding wheel or sanding belt.
3. Check to see that the grinding wheel or sanding belt is in good condition.
4. Hold or clamp the work piece securely.

g. WELDING, BRAZING AND SOLDERING

i. WELDING, BRAZING AND SOLDERING join work pieces by filling the joint with melted material; material the same as the base materials is the case of welding and a lower melting temperature alloy in the case of brazing and soldering. Since heat is used in these processes, distortion can be cause by thermal expansion and contraction as the work pieces are locally heated and then cool. Preheating the entire assembly can minimize thermal distortion. The accuracy of these joining operations is low, typically about ±1/8 inch.

ii. SAFETY PRECAUTIONS FOR WELDING

1. Wear approved welding helmets or goggles.
2. Check to see that flammable and combustible materials are not present in the welding area.
3. Check to see that other personnel in the area will not be harmed by the UV emission from arc welding.
4. Handle the work pieces carefully to prevent burns.

h. SHEARING, NOTCHING, PUNCHING

i. SHEARING, NOTCHING, PUNCHING are the action of cutting holes and shapes into sheet metals. This action is cause by 2 hardened steel cutting edges sliding by one another very closely, creating a shearing action, like a pair of scissors. The accuracy of these operations is dependent upon the skill and experience of the user, but can be accurate up to ±1/16 inch.

ii. SAFETY PRECAUTIONS FOR SHEARING, NOTCHING, PUNCHING

1. Wear approved safety goggles or glasses.
2. Check to see that the work area is clear and the tools are properly aligned and in good working condition.
   a. Be sure there isn't an excessive gap between the shearing blade and the bottom die. This will cause the material to "rollover" the edge, rather than being a tight, clean shearing action.

EN Bldg Room # L65 - Dept. 3295 – (307) 766-4243 - www.uwyo.edu/ceas/shop/
1000 E. University Ave.
Laramie, WY 82071-2000

3. Keep hands, fingers and toes away from cutting edges and pinch points while using the machines, especially the sheet-metal "stomp shear".

i. <u>WATERJET MACHINING</u>

i. <u>WATERJET MACHINING</u> uses a high-pressure jet of water and an abrasive garnet to cut material into desired 2-D shapes. Most materials include: Steel, Aluminum, composites, wood, plastics, ceramics and foams can be cut with the waterjet. Tempered glass and other very delicate/breakable materials may be exemptions, although we're having more and more success cutting materials like these, as we gain more year experience. Many parts can be arranged so as to be cut out of a single plate of material, making waterjet cutting very efficient and less wasteful than other means of cutting or machining.

ii. <u>SAFETY PRECAUTIONS FOR WATER-JETTING</u>

1. Wear approved safety goggles or glasses
2. Make sure that unauthorized persons are outside the safety zone of the machine when it is started. The pierce can be somewhat violent, splashing water with lots of noise.
3. Check to make sure that the work piece is mounted securely and the machine settings, material thickness and garnet levels are all correct.
4. Keep hands and fingers away from the cutting nozzle and out of the water while the machine is cutting.
5. Be sure the floor is dry and that extra material is placed back in the rack, to avoid making a tripping hazard for other personnel in the shop.