Milling

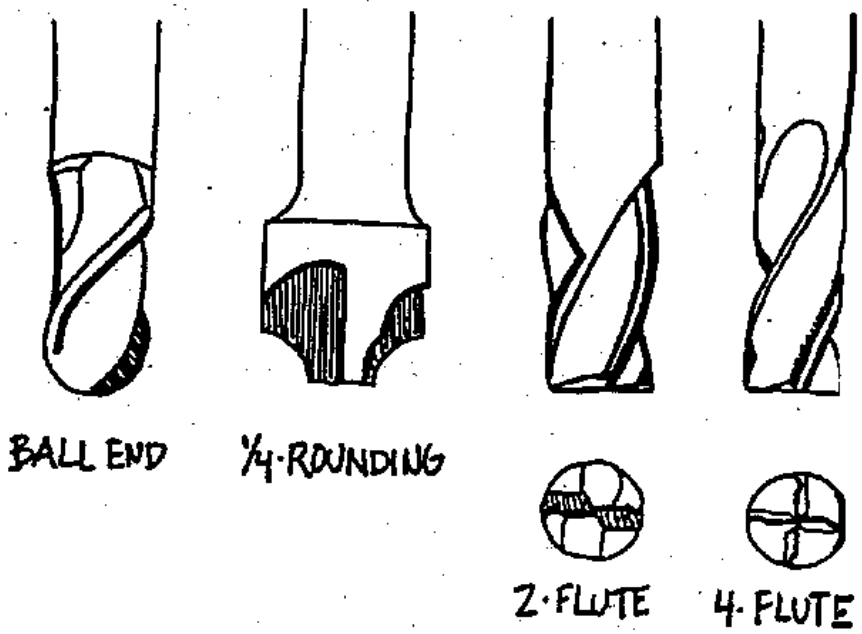
The Product Realization Lab has several different types of milling machines, including manual Bridgeports, 2-axis CNC Anilam Crusaders, 3-axis HAAS vertical machining centers, and the 3-axis Roland. Each of them has specific guidelines for how it can be used, but this section documents the fundamentals that are common to all of them.

General Mill Information

Standard Mill Tools:

Endmills

- Both the side and the end of an endmill can cut
- They are available in different styles
    - 2 flute
    - 3 flute
    - 4 flute
    - 6 flute
    - Ball end, for cutting rounds and doing 3D CNC surfacing
    - Corner-rounding, for rounding corners



- End mills are available in different lengths
    - Stub, a good idea when using a small diameter cutter if the workpiece is not very thick
    - Standard
    - Long, for reaching into deep cavities
    - Extra long, for reaching into extra deep cavities

You should always use the shortest endmill available that will cut the geometry you need. Long and extra-long end mills may leave a relatively poor surface finish because they are prone to chatter and can break easily. Ask a TA for advice when a long end mill seems necessary.

- Available endmill sizes depend on style and length
    - 2 flute standard end mills are manufactured in 1/64" increments down to 1/16" diameter
    - Ball end mills available in 1/32" increments down to 1/16" dia
    - Because small end mills break so easily, the machine shop will typically stock down to 5/16" diameter endmills. We sell 1/4", 3/16", 1/8", 3/32", and 1/16" in standard and long lengths.

- Endmills are either center cutting or non-center cutting. Beore plunging into a part with an end mill, check the end teeth to see if it is center cutting. If the teeth meet at the center, it is OK to plunge. If they don't meet, don't plunge with it.

Slitting Saws

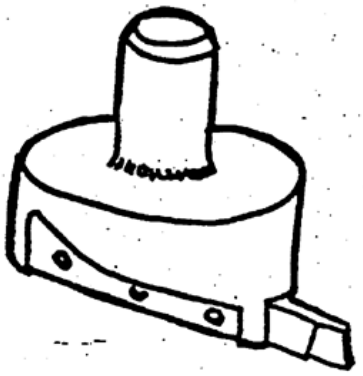
- Used for general slotting or cut-off operations
- Available down to 0.006" thick (thin)
- Because slitting saws break easily, the machine shop has a limited stock. Find out what's available. Get assistance before cutting.

Single Lip Cutters

- Custom made cutters for special applications: in our shop they are typically ground for putting draft angles onto casting patterns
- If you need something specific, you may need to make your own, so ask the staff for advice and refer to the Single Lip Cutter Grinder document in the TA Handbook.

Fly Cutters

- Used for taking light face cuts on large surfaces
- Max depth of cut depends on material being cut
  steel : don't flycut
  aluminum, brass : 0.025" max
  plastic : 0.010" - 0.050" (be careful with plexiglass, the edges will chip with too heavy a cut)
- CAUTION- A spinning flycutter is often difficult to see. Keep your hands well away from the workpiece and tool when performing flycutting operations.



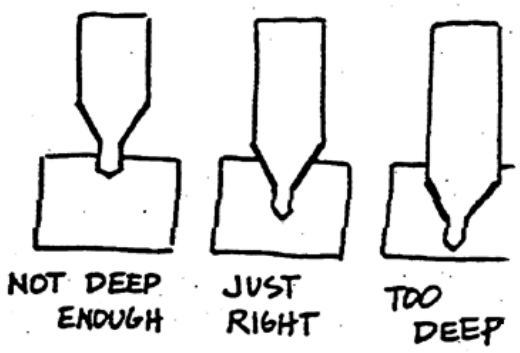
Drilling, Reaming, Boring

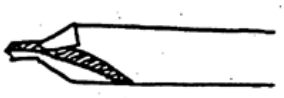
When drilling or reaming holes on the mill, straight-shank tools with diameters less than 1/2" are mounted in a mill drill chuck. Large straight-shank tools mount directly into collets. Large tapered drill bits are mounted into tapered sockets.

> **NOTE**: Do not mount taper shank drills in chucks. Chucks cannot hold tapered shanks securely.
>
> **NOTE**: Do not mount end mills in drill chucks. Drill chucks cannot withstand the forces induced on an end mill in use.

Center Drilling

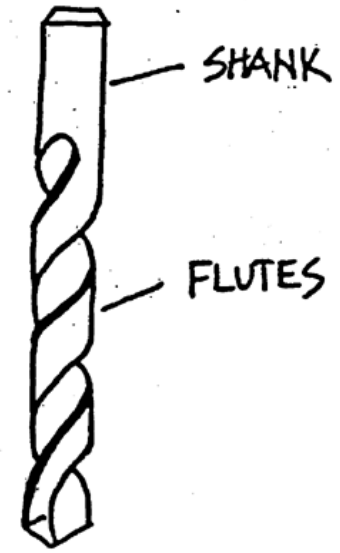
- A center drill is used for starting holes to assure accurate location. Center drills are short and stubby to maximize their stiffness and keep them from deflecting under load.
- Cut to depth illustrated - halfway up the larger taper. This creates a funnel to guide the drill bit accurately.





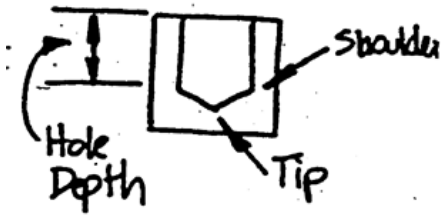
Drilling

- Twist drill sizes less than 1/2" diameter are called out by
  - Fraction: in 1/64" increments
  - Number: to fill in the spaces between fractional sizes <1/4"
  - Letter: to fill in the spaces from 3/16" - 7/16"
  - Metric: ask about availability
- Our drill chucks will hold twist drills <1/2".
- The chuck should grab onto the drill's shank, not its flutes.
- Larger drill bits are available in fractions at 1/32" increments
- Most of the larger bits have tapers that fit into tapered sockets.



Measuring Depth

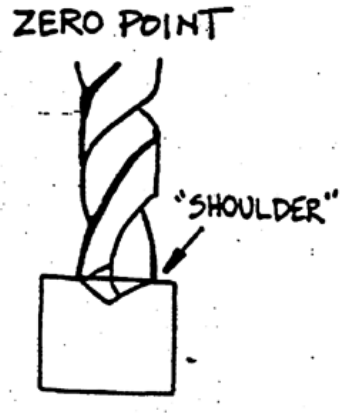
- Hole depth is usually measured from the surface of the part to the shoulder made by the drill bit, as shown at right.



- When designing parts with blind holes, keep in mind the length of the tip of the drill bit. Allow adequate wall thickness so the tip does not poke through the bottom of the part. In depth critical applications, it makes sense to measure to the tip.
- Twist drills do not make flat-bottomed holes.

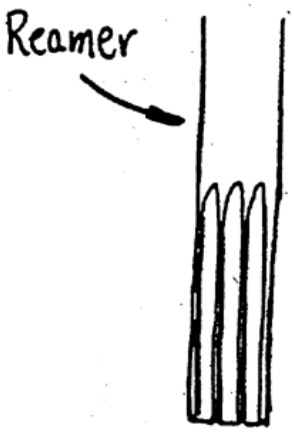
Drilling to Depth

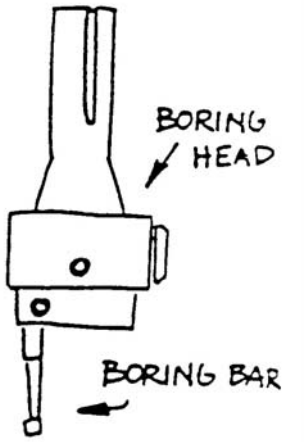
- Set the quill stop to allow adequate spindle travel to drill hole.
- Lower quill to the stop and lock in place.
- Raise table until the shoulder of the drill bit just enters the part as illustrated. This is the zero point.
- Raise the quill to retract the drill bit.
- Raise the table the distance equal to the depth of the hole to be drilled.
- Now, using the quill, drill into the part. Periodically, back the drill bit out to clear chips.
- When you hit the quill stop, the shoulder of the drill bit is at the desired depth.



Making Large or More Accurate Holes

- When drilling large holes, you will need to start with a smaller drill bit and gradually increase to larger drill bits.
- For more accurately sized holes, you will want to drill the hole undersized and finish with a reamer. Ask the TA for assistance.
- Use a boring bar mounted in the boring head to enlarge or true an existing hole or to make an accurately sized hole that is not available with our twist drills or reamers (in-between sizes or >1")



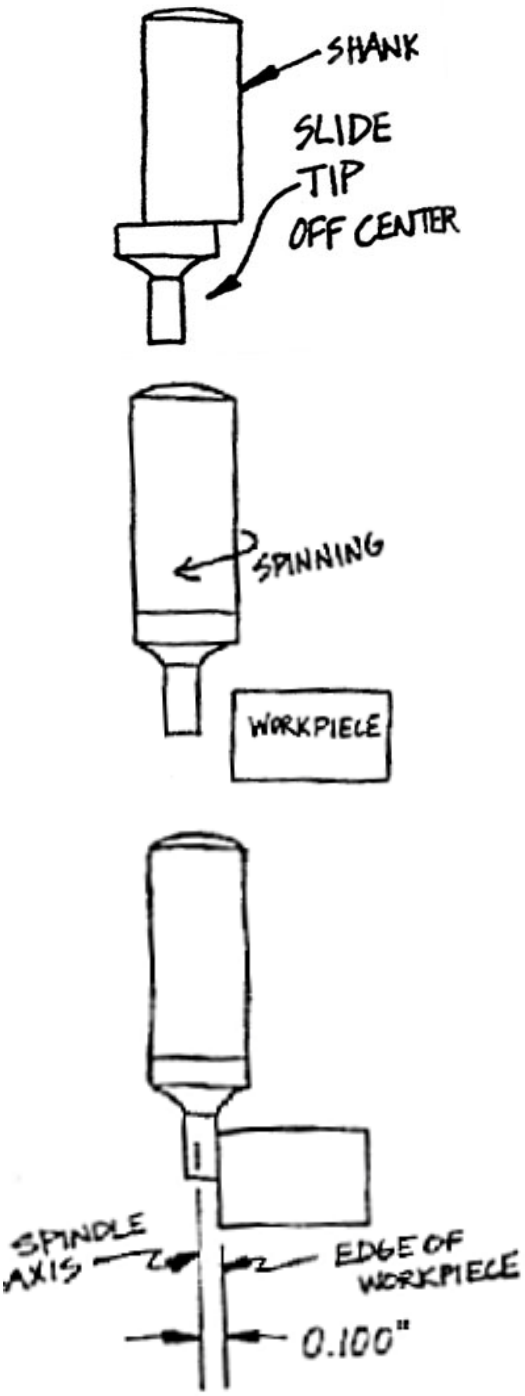


Using an Edge Finder

An edge finder is used to locate the machined edge of a workpiece.

The edge finder is made of a shank with a floating tip retained by an internal spring.

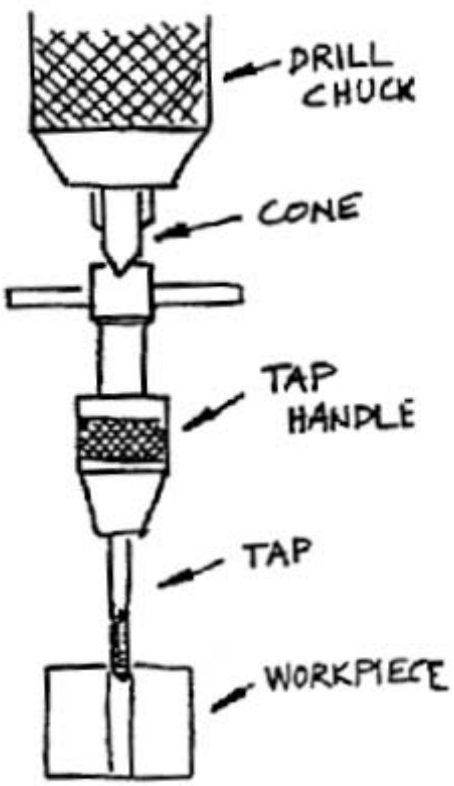
- Mount edge finder in collet or drill chuck
- Set spindle speed to >1000 RPM
- Slide tip off-center
- Turn spindle on. Raise the knee or lower the quill to bring the edge finder tip adjacent to the workpiece.
- SLOWLY move along the x-axis or y-axis to make contact between the edge finder tip and the edge of the workpiece.
- Continue to slowly advance the workpiece toward the edge finder tip. As soon as the tip jumps sideways, STOP feed or you will break the edge finder.
- The spindle is now positioned 1/2 tip diameter off of the edge of the workpiece. (diameter = 0.200", so offset = 0.100")
- Lower the workpiece or raise the quill until the tip of the edge finder is above the workpiece.
- Now advance the table 1/2 tip diameter (0.100") farther to bring the spindle directly over the edge of the part. Set the DRO to zero.



Tapping
Taps break very easily and are expensive, so read on ...

Tapping is used to thread a drilled hole. The tap drill size is found on the chart near the drill index.

- To accurately locate a hole to be tapped, start with a center drill.
- Next, drill the pilot hole with the correct tap drill.
- Tapping is done by hand with the power OFF.
- Put tap in tap handle.
- To align tap... Do not move workpiece in X or Y axes after drilling. Use a dead center in the milling machine spindle to align tap with pilot hole as illustrated. Taps break easily if bending is applied. Proper alignment eliminates bending.
- Use cutting oil or tapping fluid on metals, and kerosene on plexiglas
- Turn tap handle by hand 3-4 turns while applying light pressure into the hole.
- Now, after each 1/2 turn, back the tap out of the hole to break the chip. Continue advancing the tap until you feel the bottom of the part, or until the through hole has been tapped. CAREFUL... stop when you feel the bottom of the hole or the tap will break.



IMPORTANT - Use a light touch. It is VERY difficult (impossible or expensive) to remove a broken tap. Often a part must be re-started due to a broken tap.

Chatter
Chatter is the rattle or vibration between a workpiece and a tool due to the lack of rigid support for the workpiece or tool. You may hear screeching or notice a wavy surface finish on the workpiece.

To Prevent Chatter

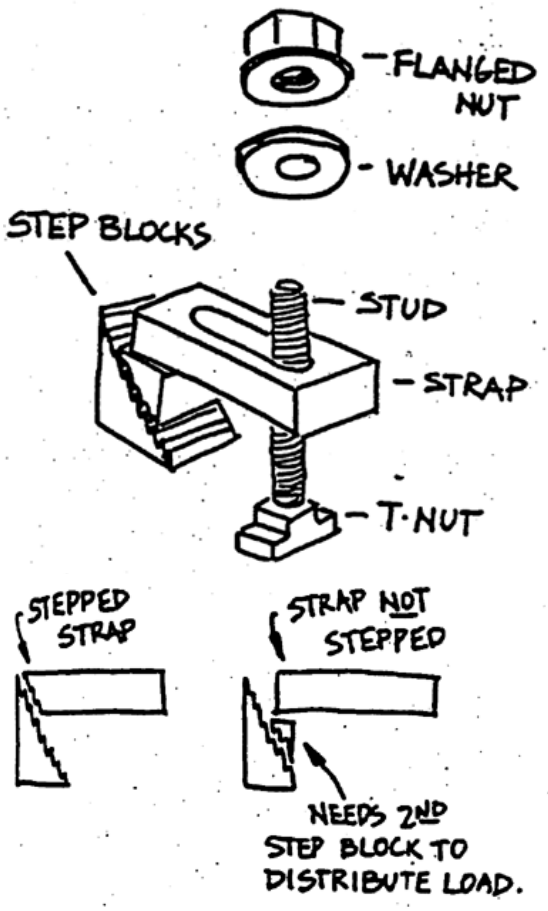
- Check spindle speed. When there is a lot of line contact between the tool and workpiece, keep the spindle speed low.
- Provide more support of the workpiece near the cut to reduce vibration of the workpiece.

If these suggestions don't help, ask a TA for assistance.

Other Workholding Devices
You can often hold parts in the vise on the mill, but some projects require other fixturing methods.

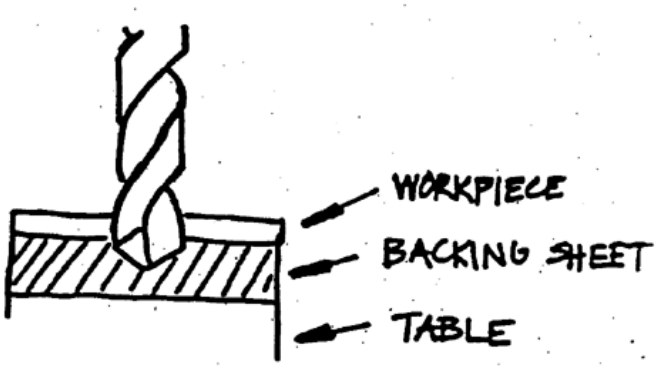
Strap Clamps

- Use at least two strap clamps to hold a workpiece.
- The parts of a strapclamp are pictured at right
- To prevent the strap from marring the surface of the workpiece, use a small piece of scrap material, such as plexiglas, as a pressure pad at the point of contact.
- To get maximum clamping force, the flanged nut should be tightened on the strap as near to the workpiece as possible.
- Parts should be completely disassembled when you are finished. If a nut gets stuck, ask for help with removing it.



Backing Sheets

- Use a backing sheet whenever there is the slightest chance of cutting through the workpiece and into a fixture or (heaven forbid) the milling machine table.
- Use a piece of plexiglass or other scrap material.
- Do not use wood. Wood chips gum up the slides on the machines. Wood will also continue to compress when clamped allowing a "loose" set up.
- When drilling, make sure the backing sheet is thicker than the length of the pointed tip of the drill bit.
- Set the quill stop to insure that you will not drill through the backing sheet and into the table.
- DO NOT MACHINE INTO THE TABLE !!!!

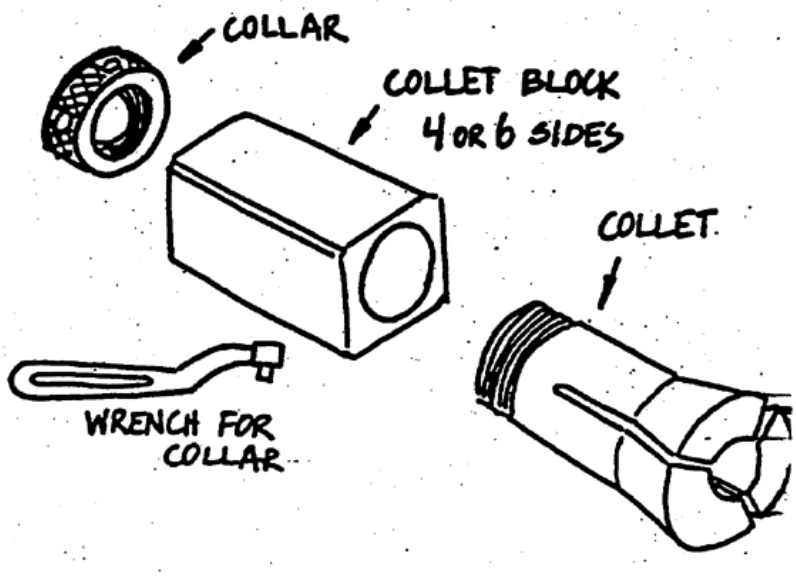


Double stick tape

- Double stick tape works only on thin, plastic sheet stock with lots of surface area to tape to.
- Use double stick tape when other work holding devices get in the way of machining operations.
- Do not use cutting oil or kerosene when fixturing with double stick tape as it may weaken the adhesive bond.
- If you are using a backing sheet, tape your workpiece solid (no gaps between the strips of tape), and "set" the tape by squishing the pieces together in a vise. Whenever possible, use an oversized backing sheet and clamp the backing sheet to the table.
- If you must tape to the table ...
    - Clean the table or fixture with solvent and clean paper towels to remove oil.
    - Apply tape to the workpiece
    - Mount workpiece and backing sheet onto the table or fixture.
    - To set the tape, clamp workpiece to table using a bar or thick plate and strap clamps.
    - Remove clamps and start machining.
    - When you are finished machining, removing your part from the backing sheet will be difficult. Release it with ethanol, heat it with the heat gun, or get assistance.

Collet Block

- Used to hold round parts in a collet (<1" diameter) during milling operations
- 4-sided and 6-sided blocks are available
- Clean outside of collet and inside of collet block before assembly
- Hold collet block in milling machine vise while tightening collar



How to change a milling machine vise

- Do not change a vise or other workholding device without first getting approval AND assistance from the TA.
- The table and vise can be damaged if not treated properly.

revision history :

Ver 1.0	????	????	original text
Ver 1.1	9/97	Bryan Cooperrider	computer formatting and revisions
Ver 1.2	8/01	Katherine Kuchenbecker	minor re-formatting and revisions