Handout #2

TOPICS: Drilling, Reaming, Tapping, RPM calculations

All reading and figure references are from *Modern Metalworking* by John Walker.

DRILLING

Before starting to drill any hole, the RPM needs to be calculated (see below). Running a drill at the incorrect RPM can cause tool chatter, tool damage, etc. Another critical factor when drilling is feed pressure, i.e. how much downward force you apply. The following rule of thumb applies:

- the smaller the drill the higher the RPM and the lower the feed pressure
- the larger the drill the lower RPM and the higher the feed pressure

If the drill has a high pitch squeak/squeal then lower the RPM and/or adjust the feed pressure if the RPM is set correctly. If this doesn't solve the squeak problem, then the drill is probably dull and needs to be resharpened.

If a drill chuck ever looks out-of-round, a dial indicator should be used to check to see if it's running true or not.

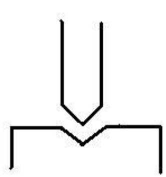
The drill ø should always be measured before use. There is no guarantee that the person before you put the drill back in its correct spot. A drill gauge or a micrometer can be used to measure the ø (Refer to Figs. 24-7 and 24-8 on p. 368).

Peck drilling, which is a pecking (i.e. down then up) action, is necessary when drilling holes. This allows chip removal from the hole as the drilling continues. Rule of thumb: the smaller the drill the smaller the pecks; the larger the drill the larger the pecks.

When drilling holes to a certain depth, the height of the drill point needs to be taken into account (Refer to Fig. 24-49 on p. 383). The drill point is not at the full ø, but we depth find (set Z0) from the tip of the drill point, so the height of the point needs to be added to the depth of the hole to get the correct depth.

Drills larger than ½" ø require a pilot hole, which is a predrilled hole ~1/4 the ø of the drill (Fig. 24-46 p. 382). A pilot hole reduces the amount of feed pressure required for the larger drills. Without a pilot a considerable amount of feed pressure is necessary to make the cut, and therefore, can be fruitless and/or dangerous.

When drilling for a tapped or reamed hole or a small Ø drill, a center drill should ALWAYS be used first. A center drill is basically an extremely short and sturdy drill. It provides a pocket for the drill point to guide into when making the first cut into the part (see below). Without the center drilled dimple, the drill will wander before cutting into the material, and therefore, will make the hole inaccurate; the drill bit can also snap and



ruin the part if it gets stuck in the material. Drill bits are made out of tool steel, which is extremely hard and very difficult to remove w/ another tool.

Whenever drilling through holes, there will always be a burr on the exit side of the hole. This can be removed using a <u>countersink</u> of appropriate size. The countersink is installed in the Rockwell Drill Press with the RPM set on 125, and the machine is turned on in low gear. Holding the part by hand, simply line the hole up with the countersink and apply an upward pressure. This puts a slight chamfer around the edge of the hole, removing the burrs and making the part look more professional.

REAMING

<u>Reamers</u> (Fig. 24-59 p. 386) are used when a smooth, accurate hole is required. Drills do not provide a good enough finish or accurate enough ø for slip fit applications. You will be reaming the holes for a number of your parts. The drill ø is chosen to be ~0.010" smaller than the actual ø desired. The hole is center drilled, drilled, and then reamed. The reamer is meant only to shave off a small amount of material; they are not designed to drill the hole itself.

Reamers should be run at ~1/2 – 2/3 the RPM of the equivalent drill ø. A continuous feed is applied throughout the entire length of the hole; peck drilling is only for drilling. Plenty of oil should be used. Reamers are held very long in the Jacob's chuck b/c it allows the reamer to align itself w/ the hole before making the cut.

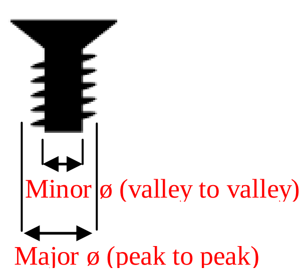
TAPPING

<u>Tapping</u> is the operation where <u>internal threads</u> are created so a screw can be inserted. Tapping is ONLY done BY HAND. Taps are extremely hard in order to cut the threads, but that also makes them very brittle. Caution must always be used when tapping holes, especially blind holes. Pressure build up on the tap will cause it to snap, and these are even more difficult to remove than drills. Tapping can be done at the tapping station or in the mill using a <u>tap guide.</u> Please ask for help when getting to this operation.

A tap handle/wrench (Fig. 9-13 on p. 147) is used to provide the leverage necessary for cutting the threads.

Choosing the correct drill ø:

Once the screw size has been determined, then a <u>tap drill chart</u> (Fig. 9-11 p. 146) is used to find the corresponding drill for the screw. The drill ø is equal to the minor ø of the screw (see figure below). Obviously, if the hole is drilled to the major ø of the screw there will be no material left for the tap to cut the threads. ALWAYS center drill before drilling for tapped holes.



There are two main types of taps (Refer to Fig. 9-7 on p. 144):

1) Taper taps – these have a taper on the tip with ~6-10 threads not at the full thread diameter. This makes it easier to start the tap in the drilled hole. A tapped hole should always be started with a taper tap.
2) Bottoming taps – these have very slight taper on the tip with only ~1-2 threads not at the full thread diameter. These taps are used to thread blind holes (i.e. holes that do not go all the through the part).
   NOTE: A taper tap can be used to tap a blind hole if there is enough material to drill a deep enough hole to accommodate the taper portion of the tap.

CALCULATING RPM

$$\text{RPM} = \frac{12 * \text{fpm}}{\pi * (\text{ø of tool})} \qquad \text{Simple formula} \Rightarrow \text{RPM} = \frac{4 * \text{fpm}}{(\text{ø of tool})}$$

fpm (cutting speed) = feet per minute

Cutting Speeds for Drilling

Material	Cutting Speed (in fpm)
Aluminum	200 - 300
Brass	150 - 300
Mild Steel	60 - 100
Stainless Steel	20 - 27

Cutting Speeds for Milling

Material	Cutting Speed (in fpm)
Aluminum	550 - 1000
Brass	250 - 650
Mild Steel	100 - 325
Stainless Steel	25 - 50

Note: The range of cutting speeds is to accommodate the different alloys of each material. For general purposes, a range in the middle can be selected. The machines in the shop are not built to go above 3000 RPM; if a RPM is calculated to be above that, DO NOT run higher than 3000 RPM.

Ex) Calculate the RPM for a ¼” ø drill cutting aluminum.

$$\text{RPM} = \frac{12 * 250}{\pi * .250} = 3820$$

Practice Problems: (The charts above will be provided on the quiz)

1) Calculate the RPM for a 1/16” ø drill cutting brass. Based on the note above, what is the RPM you would run this drill in the shop?

2) Calculate the RPM for a ½” ø end mill cutting aluminum.

3) Calculate the RPM for a ½” ø end mill cutting stainless steel. Note the difference b/t the answers of (2) and (3).