Physics Machine Shop Introduction
Mini course information

September 23, 2025

Mike Vignola
Eric Thorsland

Shop Guidelines and things to know.

- Wear appropriate clothing for an environment where metal is being cut, things are moving, and the floor may be dirty.
- This means no loose clothing, no open shoes, long pants recommended, and long hair tied back.
- Machines that cut things can cut you as well. Always be aware of what you and people around you are doing.

- Safety glasses at ALL times.

If you have any question always ask one of the staff, we are happy to be of assistance.

Cutoff or Bandsaw

Used to cut material for parts from larger stock and to do “rough machining” in preparation for moving to milling or turning operations.

The Big Saw

And the Little Saw

The Mill

Used for mill and drilling operations.

Has 2 axis programming for X and Y. Z is manual.

Can be programmed to do profiles and patterns.

Part is fixed, tool rotates. The “bed” can then move the part under the tool.

The Small Lathe, A Hardinge, a very versatile lathe and found in almost every shop.

Lathes are used for “turning” operations.

The big difference from milling and drilling is that in turning work the part rotates, and the tool is stationary.

Advanced machines move both tool and part.

We do not have this in this shop.

The “Big” Lathe A Clausing, holds larger parts, more power, more complicated controls.

The larger lathe is gear driven unlike the Hardinge that is belt driven.

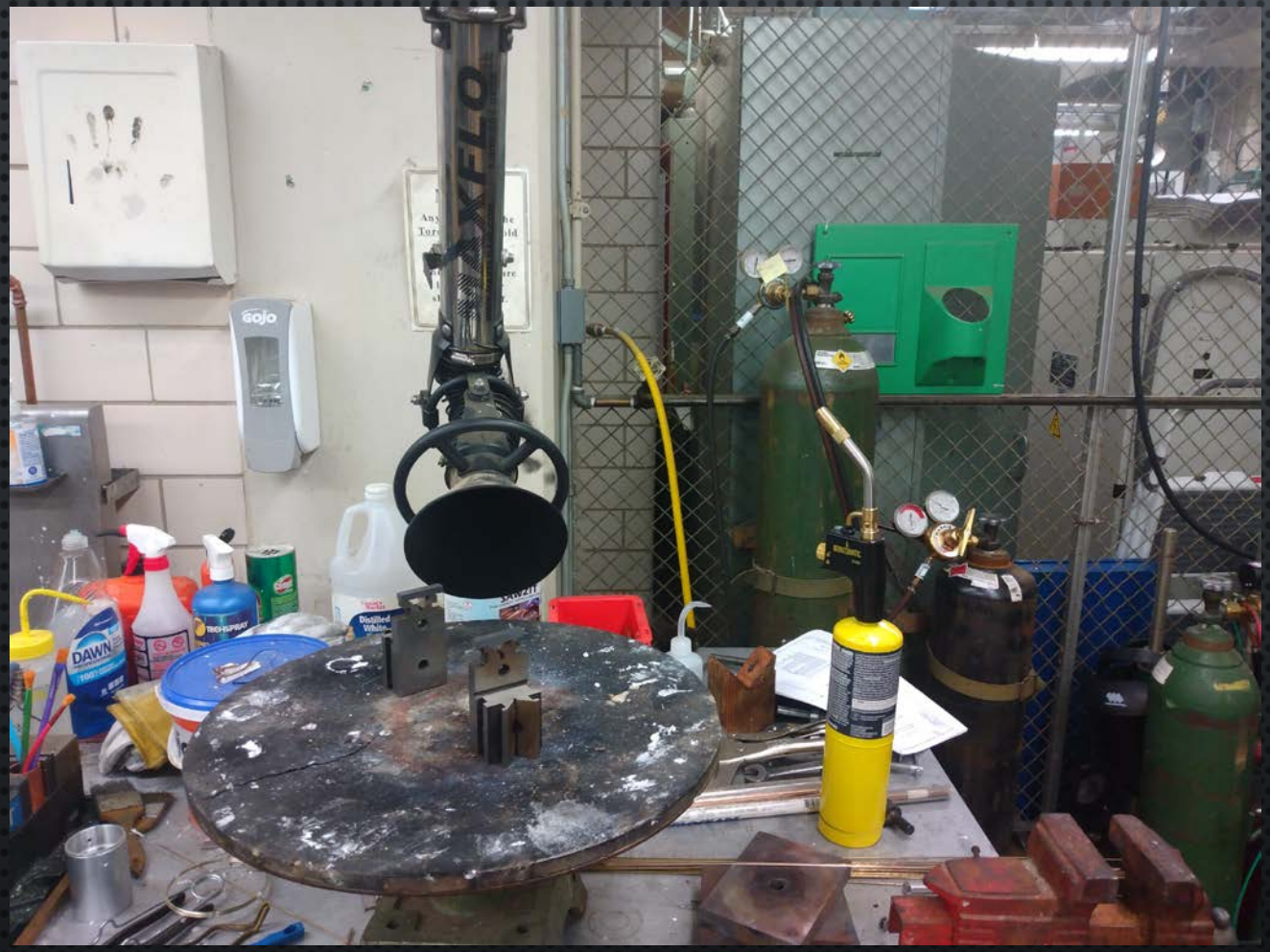
Changing speeds involves changing gears.

Can do bigger parts and larger cuts.

Lathes have powered X and Z movement and depending on controls speed is changed with dials or levers.

Brazing station. Yes, you get to play with fire.

To braze is to join two metals, similar or dissimilar by using another metal with a lower melting point. Usually a flux to provide “wetting” and to keep surfaces clean is used.



You will join several of the parts you make, and some samples using the torch and braze filler.

Parts then need to be cleaned and we will teach you the operation of the Vapor cleaning machine.

You get to keep what you make.

General shop information.

We are here to help; Mike is the main instructor for the program.

If you like what you do here, we have a more extensive course and if your PI or you wish to take the course ask us how.

The most important thing to remember about the machines in the shop is how to turn them off.

We have a record of no accidents in the shop, and we strive to continue that record.

Machining is a subtractive manufacturing process where a desired shape or part is created using the controlled removal of metal by way of cutting with a harder (but usually more brittle) metal.

Cutting with the Bandsaw

The band saw is the preferred way to cut from new metal stock because it produces a more constant surface as well as fewer particulates compared to the angle grinder and reciprocating saw.
Even still, the band saw will produce an irregular surface and a large “bur” on the side the blade exits.
It is important to “de-bur” at least one side of the part before vicing in the mill or chucking in the lathe so as to not interfere with the work-holding of the part.

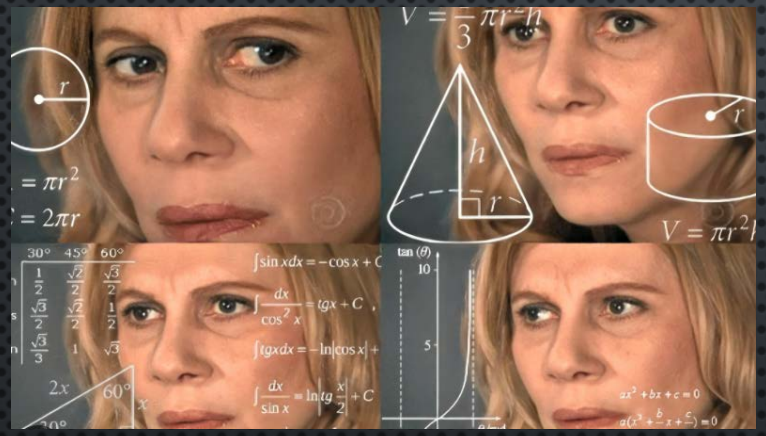
Machinability

“Machinability” is the ease with which a material can be cut (machined); allowing the removal of material to produce a low-cost, satisfactory finish.
Common materials have well-known machinability ratings and therefore SFM (surface feet per minute) scores.

Each material has a unique SFM score, which indicates the RPM speed a cutting tool (mill or drill) must be rotating for it to cut safely and inexpensively. Or in the case of the lathe: the RPM a *part* must be spinning against the motionless cutting tool.

RPM Equation:

(SFM x 4) / ø



All materials have a constant SFM score which must be calculated according to the diameter of the cutting tool (mill or drill), or in the case of the lathe, the diameter of the part against the cutting tool.

The values you get from the worksheet are maximum values so keep that in mind. It's usually safer to go slower.

Machining is a multi-sensory art/science. Listen!! Hooting, whistling, chattering, or vibrating means you should slow down the RPMs. Generally, reserve faster RPM speeds for smaller tools and (for the lathe) smaller stock.

Feed Rate:

RPM x Teeth x Chip Load

HARD SPEED AND FEED RECOMMENDATIONS

TTP HARD Drill bits	Speed & Feed Recommendations	IMPERIAL				
			Feed Per	Revolution	by drill bit diameter	
Material being drilled	Brinell Hardness (BHN)	Surface Speed (Feet/Minute)	1/6" to 1/8"	1/8" x 1/4"	1/4" to 3/8"	3/8" to 1/2"
			all below measurements are in inches			
Aluminium & Aluminium Alloys	40 - 100	325	0.0060	0.0090	0.0110	0.0135
Aluminium Cast	200	225	0.0500	0.0075	0.0090	0.0110
Alloyed Steel	200 - 300	65	0.0025	0.0040	0.0050	0.0065
Brass / Bronze	150 - 200	90	0.0350	0.0055	0.0075	0.0090
Copper	65 - 100	125	0.0040	0.0065	0.0080	0.0100
Low Carbon Steel	85 - 125	110	0.0040	0.0065	0.0080	0.0100
Medium Carbon Steel	125 - 175	65	0.0049	0.0065	0.0080	0.0100
High Carbon Steel	175 - 225	60	0.0030	0.0050	0.0065	0.0080
Cast iron (Soft Gray)	120 - 160	105	0.0040	0.0060	0.0085	0.0110
Cast Iron (Gray)	160 - 260	90	0.0050	0.0080	0.0100	0.0125
Cast Iron (Ductile)	250	80	0.0030	0.0040	0.0060	0.0080
Cast Iron (Malleable)	250 - 330	55	0.0020	0.0030	0.0045	0.0060
Tool and Die Steels	180 - 250	50	0.0030	0.0040	0.0060	0.0080
Tool and Die Steels	250 - 350	35	0.0020	0.0030	0.0045	0.0060
Heat Treated Steel	370 - 420	40	0.0025	0.0040	0.0050	0.0065

Feed rate is the speed at which a cutting-tool moves across the workpiece to remove material. It's an important factor for determining machining efficiency, surface finish, and tool life

“Speeds and Feeds”

This is the colloquial term to refers to RPM speeds and feed speeds: two separate parameters but often considered a pair because of their combined effect on the cutting process.

Zeroing: Touching off while the machine is off or zeroing off after a skim cut

With the mill, you can only touch off while the machine is off in Z dimension (the flat part on the bottom of the mill). This allows the machinist to plunge to a desired depth.

With the lathe, you can touch off in the z and the x/y.

In any case, however, it's always most accurate to take a cut and zero on that cut, ONLY THEN, use a caliper to measure.

Scribing and work holding.

It's important to "scribe" (draw) a line on the part where you want to make your first rough cut with the bandsaw and realize that the band saw blade has dimension! The band saw blade does not cut on a one-dimensional line! Be sure to position the part such that the bandsaw blade is offset on the correct side of the line you just scribed. Using a fine tip marker is usually better because you can get the tip of the marker closer to the point where your measuring tool touches the part you're trying to measure.

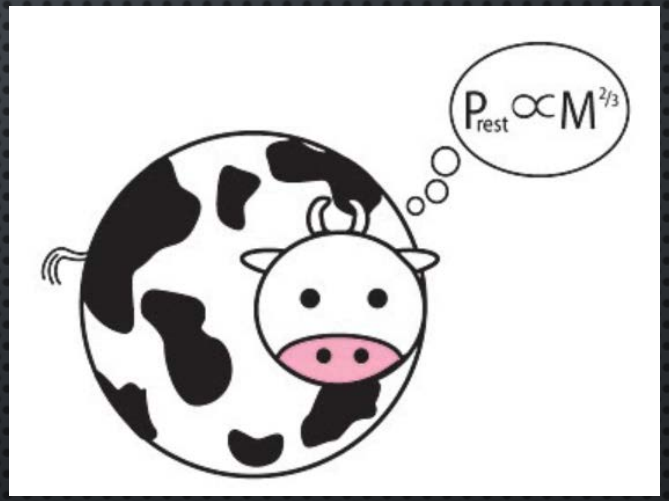
When work-holding your part, it's best to use your non-dominant hand to position the part exactly how you want inside the vice/clamp and then use your dominant hand to tighten the part into the vice/clamp. Use parallels when necessary. Always leave parallels inside of the mill (part is motionless inside of clamp). ALWAYS REMOVE parallels from inside the lathe (part spins inside of chuck) and will fly out and could hurt the operator.

Now that we have some fundamentals let's take what we have and apply it in the shop.

Everything has dimension, and ALL forces are present unlike what you may see in your physics homework.

In other words.
In the shop lines have a dimension.
There is friction.

You can't imagine a cow as a sphere.



For safety reasons, especially in an educational environment,
We always err on the side of caution.

Thank you.