The Ultimate Guide to Fabrication

Author(s): Ethan Zhao, Justin Skwirz

Editor(s): Rocky Bibeck, Anthony Duffy, Mary Endicter, and James Chi

Bolded words mark importance
Italicized words are words found in the glossary

Bolded words mark importance
Italicized words are words found in the glossary

INTRODUCTION

This is a Shop Guide that was designed to guide anyone through all possible tasks and issues they may endeavor while working in the shop. From basic safety principles, to wood shop knowledge, to basics and advanced skills on the shop floor, this manual is stacked with everything you could need to know as a FRO fabrication member. Throughout this guide, you will find explanations behind all the shop processes we encourage and execute, as well as the step by step instructions of how to execute said procedures. It will also include all of the policies in the shop regarding safety, conduct, and working ethics. If ever you have a question for anything shop related, you can refer to this manual!

General Shop Safety

The first thing that must happen above all else is understanding shop safety procedures. **Do not disobey these rules or else you are putting yourself and others at risk, and we will have to suspend you from the shop. This starts at an hour time-out and may escalate to a suspension from the team if you are a repeat serious offender.** While machine specific rules will be found in their respective sections, here are the general rules we always follow!

GENERAL RULES:

1. **Always Wear *Safety Glasses***
2. **No Jewelry**
3. **Pull up your baggy Sleeves**
4. **Tie Long Hair**
5. **Closed Toed Shoes that are tied off.**
6. **Tuck in Hoodie Strings**
7. **Wash your hands after working**
8. **Pay attention to your surroundings**
9. **No running/horseplay**
10. **Food stays on the *woodshop* side**
11. **Follow machine protocol**
12. **Listen to your superiors**
13. **No Attempting dangerous machine stunts**
14. **Take Care of yourself mentally and physically**
15. **ONLY HIT THE E-STOP IN THE EVENT OF A MAJOR (or near miss) INJURY**

Bolded words mark importance

Italicized words are words found in the glossary

First Aid Protocol

If an emergency or some kind of injury occurs, the first person to go to is the safety captain, a fellow shop member, and mentor on duty. **If the crisis requires the usage of first aid, the kits are in the pit display near the *metal rack*, and in the shop itself inside the bottom left mill drawer. You will not have to leave the building unless the injury is major, but regardless, you must fill out a *First Aid Form*.** Injuries caused by stupidity will also require a suspension from the shop. Please follow the first aid protocol that is listed here, so that our safety team understands how to avoid similar situations in the future. We also would like if a “near miss” were to occur that they be reported as well on the online Near Miss google form. This is to ensure an injury is stopped before it occurs, and so people in the future understand how to prevent potential hazardous scenarios.

How to Read the Drawing

Before even using the machines, we should discuss how to read the *drawings*. *Drawings* are the blueprint for the part you are making and therefore show us the features of said part.

For Mill Parts:

The first part of the drawing you’ll need to know and figure out is the width and height of the piece and the thickness of the walls. This should be indicated in one of the corners of the drawing by showing you the side view of the piece. This will tell you if the *tube stock* is 1x1, 2x1, etc, and what the thickness of the wall is, whether it be 1/16”, ⅛”, ¼”, etc. The piece could also be a flat plate that does not change the processes discussed. After those dimensions are established you must find the length, which is easy to find. **As for the holes, the numbers going on the long side are your X numbers and the ones on the short side are your Y numbers. Hole size is shown through numbers that have an O with a / through it.**

For Lathe Pieces

The first step is identifying the stock type (shape), and whether it's *hexagonal* or *round stock*. The side view will tell you the diameter of the piece. The front view will tell you the length of the piece. **Sometimes pieces will also require a tap, which will be shown in bolded words.**

Chop Saw 101

This is the most basic machine of the shop, the one used every time we make a piece regardless of if it is a lathe or mill piece. After you find the piece that fits the dimension you need, all that's left is to cut it roughly to the length it requires. **PLEASE REMEMBER ALL ITEMS ADDED OR TAKEN FROM THE *METAL RACK*/SHELF MUST BE NOTED IN THE *SHOP INVENTORY.*** Here is the step-by-step guide to preparing and using a machine

1. Identify the length of the piece.
2. Make a mark ¼ inch past the required length, which provides space for anything else that needs to be done
3. Make sure the saw is plugged in, then clamp the piece with the wheel, checking that the guard wall is locked and the saw is on the correct side of the marked line.
    a. **Correct side is defined by the side that you DO NOT WANT**, for example if you're cutting a piece 4.25 inches long, cut on the side of the line NOT on the 4.25 inch side.
4. Spritz the saw and piece with *cutting fluid* in the spray bottle
5. When you are ready to cut, yell "CUTTING" and bring the saw down at a nice even speed while pressing the safety and the trigger buttons. **DO NOT START THE SAW ON THE PIECE, THIS CAN BEND THE SAW OR PIECE.**
6. Once you're through, let go of the trigger and keep the saw down until the blade stops spinning.
7. Sometimes a piece needs to be cut on an angle, there is a lever on the left side of the saw that you can use to unlock the guard wall and move it to make such angle cuts.

Quadrilateral *tube stock* is done on a mill, and hexagonal/cylindrical stock are made on the lathe. There are other kinds of stock such as *I-Beams*, L-Beams, Solid Blocks, and Plates, but those are less relevant and are all manufactured on mill.

Milling 101

This section of the guide will teach you the basic principles of the Mill. It will also teach you the two main goals when utilizing the mill, which are to make the edges of a piece smooth/shave it down to size and to precisely drill in holes. This section will be split up into 3 sections, Beginning Millers, Face the Shaving, and Drilling.

Mill Safety Reminders

The basic safety rules of the Mill that should always be followed are:

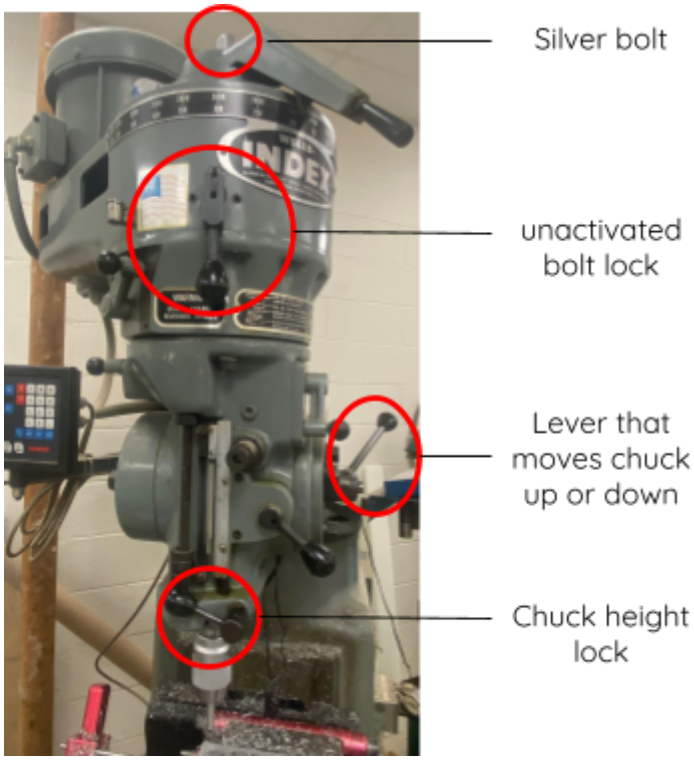
1. Never leave it running when not in usage
2. Never leave it locked while running
3. **Never place your hands near the spinning bit**
4. Remember to *oil* the machine every once in a while
5. The E-Stop can always be a lifesaver
6. Hold bits securely when taking them out
7. Stay clear of flying metal chips

Beginning Millers

This section will teach the basic principles of the mill, the first being how to lock pieces into the *vise*. To do this we only need 5 tools, the piece of interest, *parallels*, our hand, the *jello hammer*, and the *mill key*.

1. Place some *parallels* into the *vises*, ensuring the piece protrudes upward out of the *vise* somewhat when it rests on the *parallels*.
2. Use the key to close the *vise*, so that the piece can slide in without being too snug
3. Press your hand on the piece and tighten the *vise*
4. Now you'll want to use the *jello hammer* and whack the piece moving downward. Continue doing so until the piece is level and the *parallels* under will not move at all. **Be sure to place a solid piece of metal on the piece while hammering to prevent dents from appearing. You can also hit the edges of the piece to prevent dents.**
5. If the piece is long and requires the use of both *vises*, do everything x2.

To take out a *chuck* or bit in the machine, first, ensure the lock of the mill is activated, that way the *chuck* or other parts aren't spinning while you're working. To activate the lock, you need to spin it to the left and pull down on it diagonally to lock it. Once activated it will look like this:



Once you ensure that the mill lock is activated you want to loosen the silver bolt at the top of the mill. First, take a *wrench* and twist it to loosen it. **Then take a hammer and knock that bolt, holding on to the *chuck* or *endmill* and NOT BY THE SCREW.** If the *chuck* doesn't come out, repeat the process. Once the *chuck* is out, stick whatever bit you desire in and tighten the bolt with your hands and then a *wrench* to lock it in. If you need to lock the *chuck* at a certain height, flip the *chuck* lever at the desired level. **The front wheel allows you to shift the table with the piece forward and back, while the left wheel allows you to move the piece left and right.** You can also move the table left and right quickly if you press the axis to enable the button on the lower control panel, then push or pull the small lever next to the rightmost wheel in the direction you want the turbo. **The button above that lever enables speed turbo.**

Face the Piece

This section will guide you through how to face (shave) the sides of the piece and how to bring the piece down to its accurate size. *Facing* is the simple part of the section, to face you'll want to put the longer **½" end mill** into the Mill. You'll want to start by lowering the *endmill* and locking it in place. This will take off the entire edge of the piece. Then you'll move the *endmill* with the side wheels so that when you move it towards yourself with the front wheel, it will take off nothing.

Hit the reverse button to power the machine on, apply oil, and using the front wheel, move the *endmill* towards you, *facing* the side **(MAKE SURE THE LOCK AT THE TOP OF THE MACHINE IS OFF WHEN YOU HIT REVERSE).** Repeat on the other side of the piece. After you make the second face, move the *endmill* so that it is behind the piece, **DO NOT SHUT OFF THE MACHINE**, and follow the next steps.

1. Set a temporary zero by going on the *digital readout* control panel by pressing the X button, the negative button, and typing in **-.25**, because that's the size of the endmill.
2. Measure the length of the piece, then subtract from that the size the piece needs to be. This will tell you how many more inches you need to take off the piece (for example, piece length is 6.73, It needs to be 6.50. 6.73-6.50=.23 so we would need to take off .23 inches).
3. Use the side wheel to shift the *endmill* over more to take off more of the piece, **ONLY TAKE OFF .05" OR LESS AT A TIME.** You will use the temporary zero you set, plus some math to guide yourself. (for example, the zero is at -.25", you need to take off .23" inches, so the final number should be -.02". **You can only go .05" at a time, so you would take the *endmill* to the following positions: -.20", -.15", -.10", -.05", -.02").**
4. Repeat until the piece is the size you need it to be.

Now that you have completed *facing* the piece, take your piece and *deburr* the edges and we can move on

SPECIAL EXCEPTION

In certain scenarios we can shrink the size of the process significantly based on whether or not the piece is sticking out of the *vise* on both sides. When this happens, we can use a special technique that makes bringing the piece down to correct size significantly easier! It goes as follows:

1. Complete a SINGLE face and set zero as before.
2. THEN, move the *endmill* to the opposite end of the piece.
3. Doing this sets your zero up so that if you complete a pass at a number .25 above the actual size of the piece, it will automatically make it that size for you!
    a. **Note that you must cut this much above the number since that's the radius of a ½" endmill**
    b. **Also note that Step 3 from normal *facing* still applies here!**

DRILL GO BRRR

Now that you've learned how to Face, we'll move on to drilling the holes. Once you've *deburred* the edges of your metal piece, put it back into the machine, swap out the *endmill* for good friend CHUCK, and put in the *edge finder* a bit (located in the top drawer in a red pouch). This bit is what we'll use to set a permanent zero on our pieces. To zero, start by making sure the lock is off and then you can power on the mill, causing the bit to spin. Then you'll want to make sure the part of the zero bit that is capable of shifting is lined up with the top wall of the piece. **Then you'll use the side wheels to slowly nudge the zero bit until it is uncentered, at which point you can zero it by pressing X, then -.1.** You'll do the same thing for zeroing Y, except your approach will be from behind the piece and moving with the front wheel. **Also, you'll press Y, then .1 to zero the Y.** The final step to setting the permanent zero is to take the ⅜" *ALLEN KEY* and the ⅛" *ALLEN KEY* found in the bottom right mill drawer. The big key can be used to raise and down the red zero stop on the left side of the left *vise* and the small key is used to loosen the screw that controls the pole sticking out. You'll want to shift the pink mill attachment called the zero stop by loosening the big bolt with the big *Allen key* and locking it so the pole is lined up with a piece wall, then move the pole with the smaller *Allen key* so that it is touching the piece, setting your permanent zero. **Note that there are other red zero stops that approach from different angles and use the same *Allen keys*, improvising as necessary.** That is the tough part of drilling, the next steps are super simple.

1. Look at the drawing for the drill size by comparing the number they gave you on the drawing and finding what number/letter/fraction drill bit it lines up with on the chart
2. Put the lock on and put in that *twist bit* based on our *shop drill chart* (Example, .201=7 bit). If you are missing a drill bit, looking in the old mill drawers under the PPE drawers may help.
3. Look at the drawing to see the X and Y values for the holes.
4. Shift the *chuck* to the position using the wheels.
5. Turn off the lock, turn on the machine, oil it up, and lower the drill through one or both sides depending on the drawing.
6. **Note that not every hole will be the same distance apart or the same size, take note of them so that a dumb mistake isn't made.**
7. If there are holes on another side, all you gotta do is flip your piece to that respective side.

And that's it the process of drilling holes!

Cleaning Mill Pieces

Once we have a mill piece that has been *deburred* and every aspect is complete, we can clean our piece in multiple ways:

1. Take a paper towel, put it inside the piece, then take a stick and use it to push the paper towel out through the other side to clean metal shavings and oil within
2. Take a *GORILLA PAD* from the mill drawer labeled “drawer” and use it to rub off any marks on the metal
3. Finish it by wiping it with a towel of Denatured Alcohol found in the yellow fire hazard cabinet
4. Get Shop Leads Approval

Once the piece is cleaned and approved, send it to the Assembly lead, and congratulations, you learned Basic Mill

Milling 102

This section on milling will discuss the aspects of the mill that are of moderate difficulty. The main things I will be covering in this section are:

1. Advanced holes, holes that cannot be done with our standard drill bits
2. *Slotting* with the endmill

With that said, let’s start learning some more complex skills!

Advancing Holes

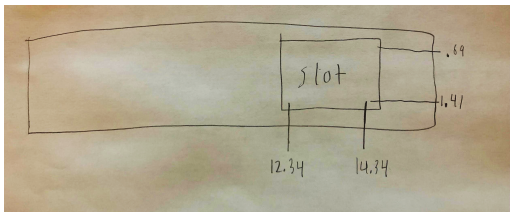
As you may have seen already through demonstration, the normal drill bits we use for drilling cap off in size at ½”, any drill bit size larger than that requires creativity. If the drill bit size is ½”< but <⅞”, you can use an *endmill* of that size to do the hole, but note that you must do a starting hole that is ½” before going to a larger hole. For large holes that are either ⅞”, 1”, or 1 and 1/8”, we will use our collection of *hole saw bits* for the job. These bits cut only the outer edge of the hole, leaving behind a little disk, and you can do them straight away without doing a smaller hole first. These drills require that we put them in what’s called a *collet*, they hold large drills/end mills in them and are stored in the *endmill* drawer. They are relatively simple to use: find the *collet* that is the correct size, put the *endmill* or hole saw a bit in them, and put the *collet* straight up in the mill without anything else and lock it in. Once it's in and you unlock it, just follow the same steps as always when drilling holes.

Note going slower and more *oil* is needed for these holes. Also when using the *hole saw bit*, it will leave behind a super hot disc that needs to be removed before drilling the other side, either by using a metal stick from the shop or any other long tool, and make sure you shut off the mill before doing so to prevent injury.

Slotting

If you're in the shop, you'll most likely have to slot at least once, especially for our first and most complicated piece, the frame rails. *Slotting* utilizes the ½" end mill that is also used in fa to make rectangular or rarely circular slots in the metal pieces. Here are the steps for *slotting*:

1. Insert your ½" endmill



2. Using the drawing, the drawing will tell you where the Half Inch *Endmill* must go both in the X and Y directions to create the corners of the slot. It will look something like this
3. Drill with the *endmill* somewhere in the center of all of the numbers and lock it in place with the lowest lever on the upper part of the Mill.
4. Move the *endmill* with the wheels to every coordinate point, In this example, you would move the *endmill* to the four following points: (12.34,-.69), (14.34,-.69), (14.34,-1.41), (12.34,-1.41). Doing so will create the slot you need.
5. **THIS PART ABOUT *SLOTTING* IS MOST IMPORTANT, YOU WANT TO GO NOT TOO FAST OR SLOW SO THE *ENDMILL* DOESN'T BREAK, YOU SHOULD ALSO USE A LOT OF *OIL*. WHEN YOU GO TO THE COORDINATES, BE SUPER PRECISE AND ACCURATE, EVEN BEING A BIT OFF RUINS THE PIECE**
6. When *slotting*, if you hear screeching, it means you are *slotting* too slowly or you need more *oil*. If you see smoke, you are moving too fast and need more *oil*.

Step 5 is why *slotting* is considered one of the more difficult skills of the mill, but you won't have to do it in a rapid amount of time. And with that, you've now learned moderate Mill Skills.

Milling 103

This section is for those ready to go above and beyond and are ready to tackle obscure more difficult skills of milling that can be difficult to grasp at first, but with enough practice are masterable by anyone. This section will cover

1. Working with Solid Dense metal blocks
    a. *Facing* them
    b. Drilling and *tapping* them
    c. Slotting them 😲
2. Eyeball Milling
3. Metal Grinding (Bearing Exclusive Skill)
4. Using small end mills for slots
5. Utilizing the depth gauge to your advantage
6. CRAZY PIECES

Solid Metal Block Training

The key difference between working on hollow tubes on the mill as opposed to solid metal blocks is that with hollow tubes, you only need to shave down a single dimension, the length. When working with a solid metal block, not only will the lengths, widths, and heights be too large, but all 6 sides of the piece will need to be faced to give them a smooth finish. In short, these pieces are significantly more difficult to complete because all 6 sides need to be faced and shaved as opposed to 2 sides with normal pieces. To make this process as bearable as possible, you're going to want to use a special bit known by our shop by the name of “Charles”. Charles is a much faster and stronger *endmill* that faces the top of the piece when placed into a *vise* and makes the process of *facing* and shaving as quick as possible. Here is the step-by-step breakdown of how to face and shave solid metal blocks.

1. Rough cuts on the chop saw all dimensions about a ¼” above the actual size.
2. Place the piece into the *vise*, and ensure you use *parallels* tall enough so the piece protrudes above the *vise*.
3. Insert “Charles” into the mill.
4. Eyeball it and get an initial face on the top side of your piece.
5. Flip your piece to the opposite end.
6. Face this end of the piece as well.

7. Power off the mill and use a *caliper* to measure the dimension you've been working on **(pay attention to which one you started with so as to not screw up your dimensions).**
8. Use a calculator to figure out how much more needs to be taken off.
9. Zero the Z axis by turning the *dial loadout* on the Z wheel to zero.
10. From here, the process of shaving on the lathe is mimicked, with the z lever being your dial. (you may go 40 at a time here instead of 30).
11. Continue taking 40 off at a time until you reach how much you wanna take off and stop.
12. Check with a *caliper* to confirm your measurement.
13. **Repeat steps 4-12 on all 3 different dimensions (KEEP TRACK OF WHICH ONES YOU'VE DONE BY MARKING THEM OFF).**

That's quite the mouthful, but realistically the process of *facing* and shaving solid metal blocks is quite simple, just very tedious. Now let's address the next important factor of milling solid blocks, drilling the holes! Drilling the holes is almost as simple as with a normal mill, however, there are some key differences

1. Long drill bits are near non-viable unless there is a starter hole.
2. Drilling with large drill bits will require a starting bit to be used to drill first.
3. **Holes may only be drilled partway through in which case you'll need to set the depth gauge (refer to lower section).**
4. You will **never** be using *hole saw bits* here.
5. You may have to tap these solid holes.
    a. This is done with the same *t-chuck* and tap bit, as well as a special hand tap holder found in the lathe drawers, bottom drawer of the upper section
    b. Remember lots of oil when *tapping*!

The last most important aspect of milling is, of course, *slotting*, which is almost identical to a process as with normal *slotting*. The only major difference is depending on the thickness of the piece and how deep you need to slot; it may be wise to do multiple *slotting* passes that do 1/3rd or 1/4th the depth at a time so your end mill bit does not shatter. **Again refer to the lower section for how to use the depth gauge in this process.** Otherwise, that's all there is to working with solid metal blocks. With enough practice, you can master these annoying pieces as well.

Eyeball Milling

This form of using the mill is very much what it sounds like, we are going to be milling without the usage of the very nice and very fancy *digital loadout*. Instead, we'll be milling with just our eyes. Now this section is much different than a section later on in the manual that covers milling with the green mill, which also lacks a functional *digital loadout*. That section will cover how to use more old-fashioned tech and skills to drill out your holes and do everything you can without the *digital loadout*. This section however is meant for super fast rough jobs that don't use precision. The first step to eyeball milling is mapping out the hole or slot you want on your piece. When mapping out you'll want to use a *caliper* in collaboration with your drawing. Set the *caliper* to the measurement indicated on the drawing, and with the sharp jaws, you can very easily mark a line indicating how far into the piece your measurement is. You may also have to use a ruler to help you map out if your *caliper* is too short. **Once you map out all the dimensions you need, you'll either be left with a cross-section for a hole you wanna drill or a quadrilateral cutout where you want a slot to be.** Now you can load your piece onto the mill, and with your eyeballs work with the marks you've been given to both drill holes and slot, and proceed with **caution** so you can work within the marks.

Metal Grinding

We use this process whenever an edge of a piece needs to be rounded **and** there is a spot where a bearing can be placed into the piece (rounding without a bearing is covered later under "obscure machinery"). It's one of the most dangerous processes that can be done in the mill, however, it creates a very nice finished product. Most of the time you'll see this be done on hollow *tube stock*, however, there have been times where it was used on solid stock, and here is how one goes about this process:

1. Borrow a bearing from the pit.
2. Place the bearing in the large hole of the piece.
3. Place cylindrical or hexagonal lathe stock into the bearing.
4. **Clamp the LATHE part into the *vise*.**
    a. You've set it correctly if you can rotate your piece without it moving.
5. Then put in the ½" *endmill*.
6. Line up the *endmill* so it's **centered** with your piece.

This is all the setup before you power on the machine, from here:

7. Inch the *endmill* so it is somewhat close to the piece.
8. Get an initial rotation done on your piece to set the zero.
9. **Then you move the end mill closer to your piece by .025” at a time.**
10. **Grind the piece by rotating it AGAINST the direction the bit is spinning.**
11. Repeat the previous two steps until the entire edge is rounded.

You must have a very firm grip on your piece during this process because if not, you are going to have a piece, a bit, yourself, or something going flying. It's also super important you don’t grind in the direction the bit is spinning, otherwise, you’ll get a messier product and a more dangerous method to grind **since doing it this way causes the bit to catch and throw the piece with lethal force.** With that said, the biggest tip with metal grinding is to not be afraid of it and to win with full confidence and stamina to beat the mill.

Small End Mill Hell

Of all the bits in the shop, the one that has seen the most combustion and breakage possible is that of the small end mill, specifically the ⅛” diameter end mill. While it is typically reserved for CNC usage only, it will occasionally be used with the mill, and when this does occur, you better watch your back. This bit is very small, so it is very easy to shatter if you are not careful with it. ALWAYS REMEMBER TO:

1. Move slowly and steadily.
2. *Oil* the piece and bit heavily.
3. Change the feed rate to be very high.
4. **USE THE DEPTH GAUGE.**

The last part is the most important, as the biggest reason these tiny endmills end up shattering is when you try and cut too deep at a time, which results in catastrophe. When cutting multiple depths at a time, we highly recommend you do passes that **go no more than ⅛” deep at a time, or else the chances of an exploding bit increase exponentially.** Unfortunately for our shop, the depth gauge installed on the main gray mill is measured in metric, which makes using it a much larger hassle. Below are the 2 main methods we can use to gauge depth on the mill, and thus allow us to do complex cuts.

Depth Gauge tutorial

Method 1:

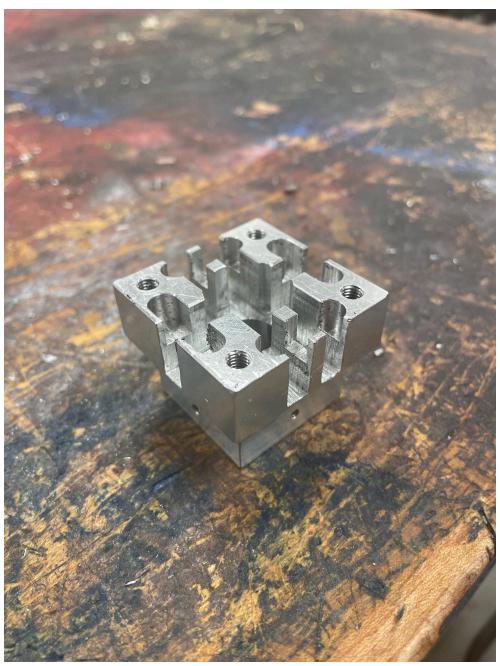
1. Bring the bit down so that it touches the top of your piece
2. Lock it in place.
3. Zero the Z lever by turning the wheel on it to 0.
4. Use that zero to gauge your depth and make your passes according to said depth (reminder every 10 is .01 of an inch lower you are going).
5. **Requires lots of math and memory.**

Method 2:

1. Bring the bit down so that it touches the top of your piece.
2. Lock it in place.
3. Lock the first clamp on the depth gauge so this depth is now zero.
4. Use either a *caliper*, a drill bit size, or a piece you know the dimension of to set the second depth gauge stopper.
5. With this second stopped set, you can unlock your bit and lower it to the second stopper, which prevents your bit from going deeper.
6. **This method is more precise, but more tedious.**

CrAzY pIeCeS eXaMpLe

An example of one of the most insane pieces we've created in the shop looks like



Ultimately, this is the pinnacle of a piece to be made in the shop, and creating something like this truly marks you with legendary status with the team. It utilizes every skill we have covered thus far and is truly a feat of machinist.

Lathe 101

The lathes are the two machines that surround the shop table, utilized whenever we need to work with a piece of metal of cylindrical shape or hexagonal prism shape. Working on the Lathe will require you to learn these 6 main skills: Basic Lathe Etiquette, FPS (Facing, Parting, and Shaving), Drilling Holes, *Tapping*, *Countersinking*, and Filing/*Deburring*. Let's start the adventures of the Lathe.

Lathe Safety Reminders

1. **NUMBER 1 RULE OF THE SHOP: NEVER LEAVE THIS KEY IN THE LATHE.**
2. Never make contact with the spindle during spin cycles.
3. If locking the lathe or unlocking it, keep the machine off.
4. **DO NOT turn on the machine while you are making contact with the spindle.**
5. Do not touch the piece as it's spinning.
6. Don't interact with drill bits if they get caught.
7. Don't reach over the lathe while it's spinning.
8. **Don't bend over to closely inspect the lathe (Russian Lathe Incident Reference).**

Lathe Etiquette

The Lathe machine in its whole design and how it works is in many ways simpler than learning the basics of milling. The first thing to notice is that the lathe when turned on by pressing the green button spins a wheel known as the spindle, which is where you put in Lathe parts. To insert a lathe part on both machines, you can use the T-shaped keys on the bottom of the lathe to open them up, then insert your cylinder/hexagonal prism piece, and turn it the other way to lock.

The other parts of the lathe are simple as well, there is a locking switch on both of the machines that stop the spindle from moving. On the big one, it's on the wall next to the spindle, and on the small one, you open a hatch behind the spindle to reach it. **Note that you should never turn on the machine while it's locked.** Both machines have a little block in the front that can be moved left and right, forward and backward, and diagonally via some wheels on it.

Next, The big front wheel one is left/right, and the smaller front one controls the front/back. The aforementioned block is where we attach the facers for the machine, using the level on the top of them. On the end of both of them is a *chuck* that you will use for putting in drill bits and for *tapping*. These chucks come with a wheel to extend them, and a lock to stop them from moving across the lathe machine. The chucks can also be locked from extending outwards if you turn on the small switch attached to the top of the chuck. Lastly, you can control the speed of the spindle with some cool panels on the side of the section holding the spindle wheel, **but don't touch it unless the shop lead or the shop mentor asks you to.** That's about it for basic lathe stuff. Now to the first skill, known as FPS.

FPS (Facing, Parting, Shaving)

Also known as facing, parting, and shaving, this skill will heavily use the block I mentioned earlier on the front of the lathe. Let's start by discussing the F of FPS, facing. As I mentioned earlier for both Lathe and Mill pieces, you get a rough cut on the saw first and then you can proceed with *facing* on the respective machine. Start by going to the lathe drawers, unlocking them with the keys in the bottom right cabinet. These are located underneath the small lathe, which can be opened by pressing the button on the door. Then use the golden keys to unlock the lathe drawers, the small one for the top section and the big one for the bottom section. Then go to specifically the first drawer of the bottom section. This drawer has both *facing* and *parting bits*, the gold ones for the small lathe, and the silver ones for the big lathe. They look something like this:

With something like this

attached to it

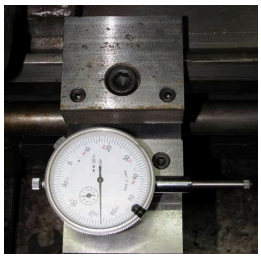
The thin ones are the parting ones while the fatter-looking ones extending to the left are facers.

Now that you have the bits, here's how to face step-by-step:

1. Load both the bit and the piece into their respective positions. You put in facers by unlocking the block with the lever on top and sliding in the *facing bit* from the top.
2. Use the wheel to line up the *facing bit* so that it will shave off the bare minimum as to not mess anything up.
3. **If you're on the Big Lathe, you'll need to lock the block in place by using the little lever on the right side on top of the block. However, the small one auto locks so you don't need to worry. Now is also when you check to make sure the spindle is unlocked.**
4. Press the green button to turn on the machine, and apply some oil.
5. Spin the smaller front well on the block to move the facer forward, taking off the whole edge of the piece to smoothen it out.
6. Repeat for the other side.

Once you've done this step, you've done the F of FPS, now time for the S of FPS, which is Shaving. You may be wondering what P stands for, it stands for parting, which is not as useful and thus is at the end of this section. Just like on the mill machine, we use the process of shaving to bring our piece down to the actual correct size it must be. Unlike shaving on the mill, we do not have a convenient *digital readout* that is super accurate, so we use a *dial readout* that is stored in the top hatch of the Lathe drawer.

The lathe *dial readout* should look something like this:



To use the dial, you'll want to use the magnet on the back of it and attach it to the side of the Lathe machine, making sure the rod sticking out from it is both parallel and the side meant to be pressed upon is *facing* the block holding the facer, similar to the image. It should look like this: After you attach the dial, you want to use either a *caliper* or a ruler to see how long your piece is after *facing* both sides, then use some math and the drawing to figure out how much more you need to take off the piece. Once you identify this number, put the piece back into the lathe. Time to zero the dial.

1. Shift the *dial loadout* to the left until the prong that is supposed to be pushed down is just slightly past the side of the piece.
2. Extend the *facing* tool so that when you shift it to the left, it will touch the piece.
3. Shift the facer with the big front wheel until it is touching the piece, then retract the facer without shifting the wheel left or right. You may need to use the lathe lock for this.
4. Turn the dial numbers using the outer rim until the dial line matches up with zero.

This will make the front of the piece position 0, allowing this *dial loadout* to work just like the *digital loadout* on the mill for shaving. From there you can turn on the machine and shave according to the number you found earlier with your math. **Important to note is that every 10 on the dial is equal to .01", and you can only go 25-30 numbers at a time, or only .025"-.03", can be shaved at a time.** Also, remember on the big lathe, you must lock the block in between every single shave, and you need to defect. It is annoying all this math, but it is necessary for perfection. The final skill of FPS is the P part, which stands for parting pieces. Unlike the other two steps, this step has nothing to do with the other steps. Parting is a process that is interchangeable with chop sawing, and basically, all it is is attaching a sharp saw-like facer on the *facing* block, and lining it up with a mark on a piece to cut a lathe part without the chop saw. It's incredibly simple but also useless because the chop saw makes this process repetitive. All you need to do is mark the piece and line up the *parting bit* with the mark before you turn on the machine, then turn it on and part away. **Part at a slower than usual speed.**

Drilling Holes, Lathe Edition

This is by far the simplest step when it comes to creating your lathe part, mostly attributed to the fact that you might not have to do this step at all if you are lucky. Sometimes if you luck out, the hole on the drawing that you need to drill will already be on the part, since some parts come with predrilled holes. **(It's always more convenient if you look for parts with correct pre-drilled holes)**. However, if you do need to drill the holes, the process is incredibly simple. S**ide Note, if the piece doesn't have a hole in it, take a smaller drill bit or a starter drill bit and drill into the piece a small amount first before going in with the actual bit.**

1. Place the piece in the lathe, lock it in, and make sure the spindle is unlocked.
2. Find out the drill bit needed, then use the chart to find it in the mill drawers and insert it into the chuck. **Shorter bits should be used, longer ones will break.**
3. Spin the *chuck* wheel so the ruler markings are at 1 inch, and do not ever retract it from this spot otherwise the *chuck* may fall out. If the *chuck* wheel is locked, unlock it using the small gray switch on top of the *chuck* holder.

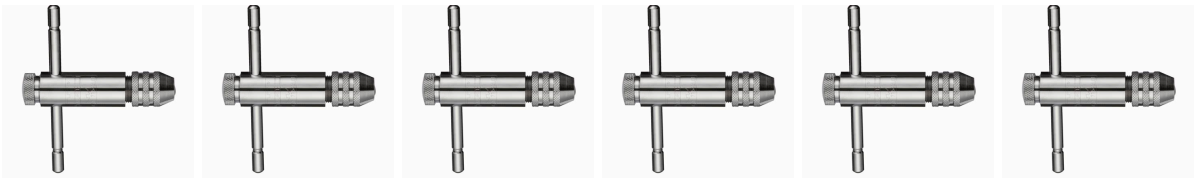
4. Move the *chuck* as close to the piece as possible without letting the drill bit touch, and lock the *chuck* in place with the lock on the moving part
5. Turn on the machine, apply *oil*, and spin the wheel until the drill bit is touching the piece.
6. Look at the built-in ruler on the chuck, then go 1 inch more than that, or however long the drill bit you chose is.
7. **Once you get deeper into the drill, the Lathe may cause the drill bit to spin, so you might want to hold the *chuck* and keep it locked as tight as possible to prevent this.**
8. When you're far enough in, move the *chuck* to the other side of the lathe and power down the machine, then repeat the process for the other side.

Unlike drilling holes on the mill, you don't need any sort of loadout because the *chuck* is already centered on the piece, and there will never be a Lathe piece with an uncentered hole.

Tapping, the Devil of the Shop

This is by far the most painful aspect of working on the lathe, and sadly a common skill that everyone must know if you want any chance in the shop. It also happens to be statistically where the most mistakes have occurred, simply from breaking the tap. Breaking a tap both ruins the *tapping bit* and messes up the piece you've been working on. If you follow this guide well, it should hopefully steer you away from disaster. Why do we tap? **It creates a screw outline inside the hole of our lathe piece, allowing screws to fit in much easier and better.** Hence, this process is more complex and different than drilling a basic hole, and the steps go as follows:

1. Identify the *tapping bit* indicated on the drill chart, the tap size is written next to the drill bit size you used.
2. Lock the spindle, **this is the only Lathe process that requires the spindle to be locked.**
3. Go to the bottom lathe drawer of the upper section and grab a *plunger* out of the black pouches, then put it in the chuck.
4. Then you'll want to go to the lowest drawer of the top Lathe drawer section and grab the *T-chuck* tap holder that looks like this.



5. Go through the smaller drawers on the upper Lathe Drawers and find the correct tap size, putting it inside the T chuck.
6. **DUNK THE SUCKER IN *OIL*, IT NEEDS TO BE LATHERED WITH *OIL* BEFORE YOUR TAP.**
7. Put the *tapping bit* touching the hole of the piece, then slide the *chuck* with a *plunger* against it, then lock the Mill.
8. Now comes the tough part, where you will need to manually spin the *T-chuck* with the tap bit attached to it slowly so that it goes into the piece more and more. The smaller the tap, the more careful and less pressure you need to exert. If you feel that it is really hard for you to spin the tap, then stop and spin the opposite way to loosen the pressure. My suggestion for smaller taps is to do 2 full rotations, 1 backward, and repeat.
9. As you are *tapping*, be sure to spin the *chuck* wheel to reset the *plunger* and see how far into the piece you've gone with the tap using the built-in ruler. **YOU MUST GO AT LEAST ½" INTO THE PIECE WITH THE TAP, PREFERABLY MORE.**
10. If you are getting uncomfortable with how much force you need to move the tap, you can take it out, dunk it back in oil, and repeat the process until you've gone in deep enough.

If you are going to pay laser focus to anything in this guide, it will be this section about *tapping*. Like I said before, the majority of mistakes made in our shop are related to *tapping*!

Countersinking

Countersinking is not the most common aspect of Lathing, but it is a useful tool for your Lathing kit. What *countersinking* does is it makes the end of the hole bigger than the rest of the hole by using a larger drill bit that is written on the drawing. **We do this just so that the head of a screw is not protruding past the outer edge of the piece**. All you need to do when *countersinking* is to take a larger drill bit than the one you drilled with previously, follow the steps of drilling except only go about a 1/16th of an inch into it, maybe even less than that amount. The countersink size will either be specified in the drawing **or the shop lead will give you a recommendation for the size.**

Filing, *Deburring*, and Cleaning the Pieces

To finish this discussion about Lathing, it's time to talk about adding in the final details. *Deburring* holes is just like on the mill, but as you might've guessed, *deburring* the outside will require a new tool that is not the edge *debur*, known as a file. The *files* are stored in the 2nd drawer of the lower section of our Lathe drawers. To efficiently file the piece, put the piece in the lathe, turn it on, and bring the file flat down at the edge of the piece, angling it as you apply pressure and moving off the edge to round it and smoothen it. **We recommend you put one hand on the file handle and a second hand on the file blade edge.** Repeat this process on the same side around 5 times and do the same for the other side. As for cleaning Lathe parts, the outside just needs to be wiped with a paper towel, and the inside gunk can be cleaned by hitting the piece vertically on the table or by blowing air through one side if the hole goes across the entire piece. Please do not place your mouth on a piece, use the compressed air stored in the CNC drawers for this. With that said, you've now learned about basic Lathing.

Advanced Lathe

Like the section Milling 103, this section will be discussing some of the more complicated lathe skills that may or may not pop up this build season: PVC stock work, Shrinking diameter, Giant Taps and holes, *Tapping* the outside of the piece

PVC Lathing

PVC lathing is very similar to normal lathe, and if anything it may require less effort than standard lathing practices. PVC lathing refers to performing all of the standard practices one may perform on the lathe, including facing, shaving, drilling, tapping, *countersinking*, etc. The only difference is you are working with plastic stock as opposed to metallic stock, which inherently means the stock is going to be more "**gummy**" and of a weaker texture. For this inherent reason, here are the differences when working with PVC as opposed to metal.

1. ***Oil* should not be used, it will melt the plastic**
2. Work slower and steadier to not warp the plastic
3. Don't tighten as hard, or else you'll bend the plastic
4. **The feed rate should be higher for cleaner-cutting**
5. Tap more carefully!

Shrinking Diameter

Welcome to the most annoying skill by far when working lathe, and that is shrinking the diameter of a lathe piece. You will almost always be executing this process on lathe pieces, and it will almost always be done if we lack a stock size the specific diameter CADD has designed for us. **Keep in mind this is a last resort, as in we cannot buy the stock with the correct diameter already.** Shrinking the diameter is very tedious of a process, and basically we are gonna use a parting bit, and move it into the piece until the diameter is the size we need it to be. Because of the nature of the parting bit and how thin it is, you'll have to repeat the following process numerous times that is listed out, making this process super tedious. Listen carefully to the following steps

1. Place your piece in
    a. Ensure it protrudes out of the spindle somewhat
2. Place your parting bit in
3. Shift your parting bit over to the left until it is over the piece
4. Move your parting bit **while the machine is off** so that it is touching the piece
5. Once it hits, turn the dial loadout on the front and back wheel to zero
    a. This zeroes out your parting bit
6. Retract the bit, and power on the machine
7. Oil as always
8. Turn the wheel until it is at zero and is hitting your piece barely
9. Here's the tricky part, turn the wheel slowly and pay attention to your dial numbers
    a. Every 10 on the dial is .01"
    b. However your taking off from all sides, so really diameter shrinks by .02" for every 10 on the dial
    c. You should've figured out already from caliper measurements how much more it needs to be shrunk **(piece is .5", needs to be .25", so gotta move the dial 125)**
    d. Move the dial slowly but surely to this number
10. Once your near your number, retract, and check your piece
11. If the diameter is good, move the left and repeat the process on an not shrunken part
12. Continue repeating all of this until you have a lath part the correct diameter

The repetitive and confusing nature of this process is what makes it very annoying to execute, and thus is a skill reserved for more advanced shop members and pieces that absolutely necessitate such a process.

Giant Taps and Holes

As of the time of making this guide, giant taps have not ever been a necessity and on average the largest tap we do is a 5/16-18 tap size. However, in the scenario where we need to tap with a really big bit, those bits can be found in the storage closet or the old drawers underneath the personal PPE drawers. These taps are much sturdier and thus much safer to utilize. As for drilling larger and deeper holes, the only main difference is using lots of *starter bits* and incrementally increasing the drill size and length. The recommendation is never to drill more than ¼" a drill bit size up at a time after you use the starting bit. So let's say if you need a hole ⅞", start with a starter bit, then a 7, then a ⅜", then a ½", and then a ¾", and then a ⅞". Note that you **always want to drill as deep as possible with a short bit before switching to a long bit to prevent a long drill bit from snapping into two and making the shop very sad.**

Dies and Taps on the outside

When we talk about dies, we are not talking about dice that you'd roll at a casino, we are talking about a special bit known as a die, and these bits allow us to tap the outside of a piece. We may need this skill throughout the season if we have a *round stock* piece that needs a threaded end so it can screw into another piece. You will rarely have to use a die on a piece that isn't *round stock* since it isn't round. Their sizes use the same tap size chart system as normal taps, and *drawings* will indicate what size they want the threads to be ahead of time. To use a die, you'll need the die itself and the die holder, both of which are depicted below

And in order to use:

1. Clamp the piece you will be using the die on into a *vise*, or into the lathe itself.
2. Loosen the die holder, and place the die in.
3. Tighten, and then *oil* your bit.
4. *Oil* the piece.
5. Place the die on the edge of the piece, and begin spinning clockwise, when you tap as far as you need to, back out.

The toughest part about doing this process is having **steady hand control of the die.** If you do not spin it straight, your threads will come out crooked and thus screw up the entire piece you were just working on. DON'T LET THIS HAPPEN!

Wood Shop Fundamentals

This section of the manual will cover all of the essential basics one will need to understand if they plan on using the *woodshop* for any reason whatsoever during the robotics season. This may include

1. Field building parts
2. Prototyping parts
3. Fun while bored pieces
4. Miscellaneous wooden projects (ex: safety signs)

There is not much complexity involved in all of these tools and machinery so this section will be done in a very brief, general, and bulleted format

General Tools:

1. The first section (closest to the door)
    a. Drills
        i. Impact drills for screwing
        ii. Standard drills for drilling holes
    b. Palm sanders in this section
    c. Holesaw, Forstner, and standard *twist bits* in this section
    d. Sandpaper in this section
2. 2nd section
    a. Levels
    b. Saws
    c. Screwdrivers
    d. Files
    e. Squares
    f. Wood picks
    g. Rulers
    h. Hammers
    i. Crowbars
    j. Other necessities

3. Red cabinets are located by the dust collector controls, and this stores what is labeled on each drawer, **INCLUDING YUMMY WRENCHES (do not consume).**

Miter Saw

1. **Used for cross cuts only (cuts against the grain of the wood).**
2. Place the piece flat against the guard walls.
3. Press down against the wall.
4. Pull the cutting trigger and bring the blade down on the mark you wanna cut at.
5. **Leave the blade down until you are done cutting.**
6. You can also adjust the angle on the miter saw with the lever on the bottom.
7. Flick it up to unlock, and flick down to lock on the degree you want.
8. Lastly, a switch on the back of the miter saw can lock it to keep it from moving

Table Saw

1. **Used for rip cuts (cuts with the grain of wood)**
2. To use, set the measurement you want to cut on the measuring system on the table
3. Flick the switch up to unlock, and down to lock.
4. Then place the piece firmly against the guard wall.
5. **Push the piece through (use a push stick to help if your hands are too close to the saw).**
6. Power off the machine once the piece is through, and don't touch the piece until it's completely off.
7. **DO NOT ALLOW CONDUCTORS TO MAKE CONTACT WITH THE SAW BLADE.**
8. You can change the height and angles with the two wheels on the bottom of the saw.
9. Loosen the center knob to be able to crank them.

Planner

1. **Used to thin wood.**
2. Cannot be used on end grain or **plywood**.
3. Turn it on with the green button (red must be pulled out).
4. The ruler on the side shows the thickness you are cutting too.
5. The wheel increases or decreases this thickness.
6. When running wood through the planer, only take off a **sixteenth** of an inch at a time.
7. **MUST HAVE DUST COLLECTOR ON WHILE USING.**

Drill Press

1. Used for rough drilling.
2. Can use the bits in the mill drawers.
3. *Chuck* works the same way.
4. **Use a backstop with this.**
5. **HOLD WOOD TIGHT.**
6. Pull the circular switch out to power on.
7. The spinning wheel controls up and down.
8. A laser pointer is used to help point where the drill is going.
9. Beware of jumps.

Belt Sander

1. Used to round the edges of parts.
2. Mainly for wood, but can be used for some metal.
3. **Clean up metal chip residue, please.**
4. Round with gloves to save yourself from burning hot material.
5. Eyeball skills needed.
6. **Apply even pressure and even circular motion.**
7. Round till it's to your liking or to a rough sketch someone gives you.

The Rockwell/Band/Jigsaws

1. Used to make weird-shaped cuts in wood and Polycarb.
2. Sometimes needed to make special parts for the robot.
3. To use, simply flip the switch on.
4. Carefully feed the wood through the blade with your hands.
5. Use the clamp if there is one from above.
6. As for hand jigsaws, find a flat surface first, place the piece you want to cut off over the side of a flat surface and use the hand jigsaw to cut it out from there.
7. **Very important to have a steady hand.**
8. Pencil to pre-mark out the path you wanna cut.
9. 4 of them, a small Rockell, Mr. Squiggle, a Big boy with a table saw-esque guard wall, and the hand one

The Legendary CNC

The CNC is a very special machine in the shop, different from any of the other aforementioned machines. This is because this machine runs automatically based on precoded CAM files, which stands for Computer Aided Manufacturing. Although this process can be very tedious, it is vital to the functionality of the team. The CNC allows ` for very complex cuts on very thin stocks to be executed without the necessity of manual labor on say a milling machine. However, using it is no simple feat, and to do so one must master 3 components, basic properties and usage, CAMMING files, and advanced CNC cutting work and technique. Master these and you'll master this difficult machine that has the potential to cut down on loads of time spent on parts for the robot.

CNC Safety Reminders

1. Always keep the emergency stop button within reach at all times.
2. Stay clear of flying stock debris from the machine
3. Always use the proper feed and coolant settings to prevent injuries, burns, and flying tools.
4. Keep personale nearby behind the polycarb shield in case something is not screwed in correctly
5. Make sure to turn off the spindle when it is not cutting.
6. Ensure that there is enough power to machine else it may short circuit
7. USE THE CNC E-STOP BEFORE THE INJURY OCCURS!
8. Never Assume everything is correct

The Basics of CNC

To begin this section on CNC, we are gonna go over all of the most basic principles of how to simply **run** the CNC. This is the much simpler portion of working with CNC, it involves the processes of putting a piece onto the CNC machine, interchanging end mills and drill bits depending on the type of cut you need to perform, zeroing the CNC, and uploading/running the CAM file. **At a bare minimum, every shop member should know how to operate the CNC machine. That way, if the person who knows how to CAM and CNC is unavailable for a meeting, they can hand the CAM file to a shop member, and they can simply run the code without them so production is not slowed down.**

Here are steps in order of how to run cnc

1. To turn on the CNC, flip the red lever on the breaker box behind the CNC. Then turn the switches on the right side of the machine to the on position.
2. To power on the laptop, make sure it is plugged in, then enter the password, “Tesla”. Once it’s powered on, open ThrottleStop and Mach4.
3. To manually move the CNC, use the arrows. (From behind the glass) The UP arrow will move forward, the DOWN arrow moves backwards, the LEFT arrow moves left and the RIGHT arrow moves right. To move the CNC up or down, use the PAGE UP or PAGE DOWN keys. However, make sure to tap them instead of pressing and holding.
4. Upload your CAM files onto a USB and transfer them to the laptop. Select ‘Load G Code’ and upload the files.
5. Place the sheet stock of wood/metal/polycarb you're working with on the CNC table, making sure it’s lined up wherever it needs to be so that the piece will come out straight.
6. Screw in the corners and other places so that chunks of your stock will not fly everywhere, but be careful to make sure they are not in the way of where the CNC is cutting.
7. Next, zero the piece. There is a metal block on the side of the table, lined up the bottom edges with the corner of the stock you’re using/where it was Cammed. You’ll want to have another person help you during this process by holding the block in place.
8. To begin zeroing, select the button that says ‘Auto Z Touch Plate.’
9. A screen will pop up with 9 squares, and at the top there will be a type box labeled ‘Tool Diameter.’ Enter the size of your bit, inches for decimals and millimeters for whole numbers. Then select the square you’ll be zeroing- it’s typically the bottom left corner.
10. Next a prompt will pop up reminding you to put the magnet on the spindle. This magnet should be placed on the black part of the spindle.
11. Make sure the spindle is centered as best as you can before selecting this square, because it will move to touch the block. You’ll also want to have the emergency stop button with you for the entire zeroing process..
12. After this has finished, a box will pop up asking you to orient the flutes with the Y-axis. Do that and then click ok. It will move to the side, and then back to the center.
13. Next, it will ask you to align with the X axis. Do that, and click ok.
14. Once finished, remove the block and the magnet.
15. Select ‘Go to work X/Y Zero.”

16. Before you begin cutting, trace the outline of the piece with the machine. Make sure everything fits and it won't run into any screws or other obstacles, and that it's zeroed correctly. You should also make sure you have enough coolant if you're cutting a metal piece.
17. Turn on the spindle with the OFF/ON slider to the right of the screen. You can adjust the speed it spins at using the Spindle Speed.
18. Turn on the coolant with "Relay 1" and "Relay 2."
19. Select "Cycle Start G Code." This will start the program.
20. To adjust the speed the machine moves at, change the 'Feed Rate.' However, make sure it is not too high once it begins cutting.
21. Keep the emergency stop button nearby at all times, and keep an eye on the machine to make sure it doesn't get too close and burn your piece.
22. If anything is off with your code or cutting but is not an emergency, or if you need to stop early, press the red 'Stop' button.

Camming on the CNC

In this part, it will explain how to CAM all potential contours, holes, and other aspects of a CNC piece one may encounter. **In general at least 2-3 shop members should be able to adequately CAM a file quickly so production will not be slowed down.** Below are guidelines to help anyone be able to CAM a file that can then be run on the CNC, a very useful tool in kits of many:

Importing the Model:

1. **Open Fusion 360:**
    - **a.** Launch Fusion 360 on your computer.

2. **Import the STEP File:**
    - **a.** Click on "File" in the top menu.
    - **b.** Select "Open" from the dropdown menu.
    - **c.** Choose "Open from Computer."
    - **d.** Locate and select the STEP file, then click "Open" to import it into Fusion 360.

Setting Up CAM

1. **Switch to CAM Workspace:**
    a. At the top of the screen, click on "Design" to return to the design workspace. Then, click on the "Manufacture" workspace.
2. **Create Setup:**In the Manufacture workspace, click on "Setup" in the toolbar. Select the model or body intended for machining and set up the stock size and orientation. Click "OK" to confirm.

CAMming a Contour

1. **Create Contour Operation:**
    a. Click on "2D" in the toolbar, then select "2D Contour" from the dropdown menu.
2. **Select Contour Geometry:**
    a. Choose the contour to the machine by selecting the edges or curves from the STEP file.
3. **Specify Cutting Tool:**
    a. Click on "Tool" in the toolbar, then select the appropriate cutting tool for the contour operation.
4. **Set Machining Parameters**:
    a. Define cutting parameters such as cutting depth, toolpath strategy, and feed rates based on material and machine capabilities.
5. **Simulate Toolpath:**
    a. Click on the "Simulate" button in the toolbar to preview the contour toolpath. Ensure it looks correct and doesn't collide with the stock or model.
6. **Generate Toolpath:**
    a. Once satisfied with the simulation, click on "Generate" to create the toolpath for the contour operation.

CAMming a Hole:

1. **Create Drilling Operation:**
    a. Click on "Drilling" in the toolbar, then select "Drilling" from the dropdown menu.
2. **Select Hole Geometry**:
    a. Choose the holes to drill by selecting the points or edges from the STEP file.
3. **Specify Drilling Tool:**
    a. Click on "Tool" in the toolbar, then select the appropriate drilling tool for the operation.

4. **Set Drilling Parameters:**
    a. Define drilling parameters such as hole depth, toolpath strategy, and drilling feed rates based on material and machine capabilities.
5. **Simulate Toolpath:**
    a. Click on the "Simulate" button in the toolbar to preview the drilling toolpath. Ensure it looks correct and doesn't collide with the stock or model.
6. **Generate Toolpath:**
    a. Once satisfied with the simulation, click on "Generate" to create the toolpath for the drilling operation.

Post-Processing and Verification

1. **Post Process for Mach4:**
    a. After generating each toolpath, click on "Post Process" in the toolbar.
    b. In the Post Process dialog, choose the Mach4 post-processor from the list.
    c. Adjust any additional settings as needed.
    d. Click "Post" to create the G-code files.
2. **Verify G-code:**
    a. Before sending the G-code to the CNC router, review it to ensure accuracy and correctness.
3. **Save and Exit:**
    a. Save the Fusion 360 file, and exit the CAM workspace.

Uncommon Machinery/Skills

This next section of the lovely shop guide will discuss the more obscure less important machinery of the shops. These machines are not used super often, but they will appear in the shop *drawings* this upcoming build season. So although these skills are useful, I would master and memorize the basics of Mill and Lathe before trying to master the topics I will discuss here. In this section I will be discussing the following: Metal Sanding, Jump Shearing, Metal Bending, and Wish Brand Milling.

The "Metal Grinder"

Okay, a major disclaimer about this skill, it is one of the more dangerous skills you will need to know if you choose to work in the shop. The sander in the *wood shop* was not designed to sand metal, and thus you must be incredibly careful when using it on metal, as it wasn't designed to be a "*Universal Grinder*". Additionally, **DO NOT GET CAUGHT BY MR. RUGARBER SANDING THE METAL, WE DON'T WANT PROBLEMS WITH THE SCHOOL.** With this warning said and done, why bother sanding metal pieces you may ask? Sanding allows us to create rounded edges on our Mill pieces if the drawing specifies a rounded edge. Usually, the drawing will indicate how far down the piece in inches the rounding begins, and the top of the semicircle you create from rounding is the already defined edge of your piece. Here is how to sand metal step by step

1. The toughest part is marking where you are sanding. First, mark the piece where the drawing indicates the beginning of the round. Then to your best ability, draw a semicircle connecting the lines on either side of the piece and the center of the piece's edge.
2. Take your piece to the sander, **preferably with welding gloves**, and press the green button on the sander
3. When sanding, keep the piece flat on the platform, and push the piece against the sander, **KEEP IN MIND IT WILL GET VERY LOUD**
4. To your best ability follow the mark you made and create as smooth a circle as possible
5. When you are done, wait some time before you *debur* the new edge of the piece, as it is probably incredibly hot

Jump Shears

The jump shears are the green manual machine next to our certified shop table and are kinda like a super primitive chop saw. The reason I call it primitive is that it is a 100% manual machine, **and it can only be used to make rough cuts on flat metal plates that are aluminum.** Though a barbaric machine, it is super simple to use

1. On the back of the jump, shears are some long bars with a built-in rule and two wheels attached. The small one loosens the big one allowing it to move, and vice versa. Use the Big wheels to set the desired measurement for your cut, ranging from 1/16”, to like 3 ft
2. Place your metal piece so it is up against the side and back wall of the jump shears
3. Contrary to its name, don’t jump on the yellow platform to activate it, simply stomp on it.
4. **You may have to repeat multiple times before the cut is made**

And now you’ve cut a metal plate you can give to the CNC man or use as a mill piece

The Mill’s Great Grandfather (Green Mill)

In case you have yet to notice on your shop journey, next to the gray mill is this much older, more beat-up green mill, that surprisingly is 100% fully functional. However due to the fact this mill lacks the digital readout that is found on the gray mill, working with this mill is much more tedious and annoying than the other mill, and makes for overall a much greater struggle in creating a piece. You may be asking yourself then why do we bother using this green mill, well using the green mill will help in speeding up production of parts during build season. This green mill can shave/face very easily and get that step of pieces done and then they can be sent to the gray mill for drilling holes and other things. So basically having a second mill opens up the opportunity for an assembly line and hence why this section here exists to explain how to use it.

Facing/Shaving on it

Facing with this machine is no different than if you were to face with the gray mill, you would have to just take off a tiny amount on both sides of the piece, however once you have to execute shaving the piece down to size, that's when it all becomes different significantly. Because we have no digital loadout we need to figure out

A. How to zero our piece to the endmill
B. How to figure out how much we move at a time spinning the wheel
C. And how to make sure we take off accurate amounts at a time

All three of these factors seem very difficult without the digital loadout, but I ensure it is much simpler than it appears on paper. The key detail to remember is that these mills have on their wheels a dial loadout, just like the ones we attach to the lathes for facing and shaving. In case you haven’t figured it out yet, the face and shave with this green will just use these dials and the same facing process as on lathe!

1. “Zero” your dial after your second face by turning it to zero
2. Tighten it so the dial can’t rotate unless you move wheel
3. **Move 50 at a time until you get to number you gotta take off**
    a. Cause 50 on dials equal .05”
    b. Get how much you need to take off with calculator math
4. Remember to cut slightly over just in case
5. And of course you'll want to oil.

Because of how simple the facing and shaving process is on this mill, it is 100% a viable method to shorten production time especially when the assembly line method used with lathing is put into play.

Drilling holes on it

This is the section where using the green mill begins to become a lot trickier. First off, the green mill doesn’t have a zero stop so even if you manage to set a zero on your mill it's very tedious regardless of having to continuously fix the zero on this mill. Hence why this mill should only be used to drill as a last resort for efficiency. As far as how to zero on this mill, you use the same edge finding bit and process, except for inputting the zero. **To input the zero, once the bit shifts, take the wheel on the X-axis and set it to 90 (equivalent to the -.01” on the digital readout), and turn the Y-axis wheel to 10.** He permanently sets your zero so long as you don’t remove the piece. Now drilling your holes is rather a simple process however

1. Map out your piece well with where all the holes are so you can constantly double check yourself
2. Move the wheels according to how far into the piece you need to go
    a. Full rotation on the wheel is .1”
    b. You need very good memory and precision (going .5” deep means you have to count 5 rotations from 0)
    c. Makes this process way more tedious
3. Don’t forget to oil
4. **Don’t forget the rules of working with a big hole**

So in reality, not a terribly difficult process but not the most ideal method for drilling holes.

Slots of Hell on it

Now unless it's the apocalypse or it's an absolute emergency, I VERY strongly recommend you do not use the green mill for executing out a slot. Not only is it significantly more annoying because of how the dial loadouts work, but the amount of memory you need and precision is ridiculous. Basically once your piece is zeroed

1. Meticulously use the wheels to move yourself to a coroner or starting position of where the slot is.
2. Drop your endmill down
3. Lock it in place
4. Use your wheels and very carefully move however far left/down/right/up the drawing wants you to move
    a. **Reminder: a full rotation on the dial is .1", so keep that in mind when you have to move the endmill 1 or 2 inches**

Step four is where it all gets tough, because having to keep track of the rotations while checking and making sure your piece is solid gets to be very annoying. Just keep in mind the steps of speed when slotting and that you should always map out your slots so you can easily double check with your slotting process. Slotting without the digital loadout is a nightmare, so I recommend doing it as a last ditch effort.

VISE REALIGNMENT

Vise realignment is **THE** most annoying thing anyone in the shop may have to conduct for the mill machine. It occurs when the bolts of the mill are not tightened properly, and over time the mill *vises* shift ever so slightly leading to the *vises* not being straight and being misaligned with each other. When this happens, pieces you attempt to work on have slanted edges from facing, uneven hole spacing, and a multitude of other problems that make life a nightmare on the mill. To make matters worse, there are not only two individual visas that will need to be aligned themselves, but they need to be aligned with one another. Though it is a tedious process, the saving grace is that it's simple in execution, and the steps are as follows:

1. Grab the aligner in the top lathe cabinet
2. Place it into a *chuck* on the mill
3. **Clean both *vises* so they are free of any debris**
4. Move the bit to as far left of the *vise* as possible
5. Move the y-axis so the *vise* is barely pressing up against the dial so that the dial goes to 5
6. Activate turbo, and move the dial to the other side of the *vise*
    a. **If the number increases, loosen the bolt on the right and tap the *vise* with a hammer on the front end until the number is 5**
    b. **If the number decreases, loosen the bolt on the right and tap the *vise* with a hammer from the back end until the number is 5**
7. Then go back to the left side of the *vise* and repeat steps 5 and 6 until the number stays 5 consistently
8. Repeat all of the steps for the second *vise*
9. Once both *vises* are aligned, put a long metal piece in that is long enough to stretch the length of both *vises*
10. Move the dial to the left of the piece and against it so the dial is 5.
11. Move to the right and conduct the same process as in step 6 EXCEPT:
    a. If increases, tap the right *vise* in the center from the front
    b. If decreases, tap the right *vise* in the center from the back
    c. LOOSEN BOTH BOLTS so it moves as evenly as possible
12. Then go back and keep testing steps 10 and 11 until it's 5 the whole time
13. Then repeat step 8 for the right *vise*
14. **Finally repeat steps 10, 11, 12. If on the first try, you get it to be 5 the entire time, DO NOT REPEAT step 8. If it takes multiple fixes to get it to be 5 the entire time, REPEAT STEP 8. If you repeat step 8 be sure to repeat step 14 until it's all good and aligned and JESUS CHRIST** 😮‍💨

All of these steps in repetition combined is how you may spend an entire 4 hours spent trying to realign the vises, and we recommend only executing this process on days where work is light and requires both vises on the mill be aligned.

How We Work in the Shop:

In this final section of the shop guide, we will be discussing the basics of shop procedures, that way you will understand how we work in the shop and how we apply the skills you've learned.

Personal PPE Drawers

Near the shop table are personal PPE drawers each with the shop members' names listed on them. Inside these drawers, you may feel free to store any personal belongings or essentials for your safety. In particular, we highly recommend that you have

1. **Own safety goggles!**
2. Working gloves
3. Ear protection of some sort
4. Chargers for phones
5. Parts you were working on previously
6. Jewelry during work hours
7. Other belongings

Working in Groups

First off, CADD tends to send us *drawings* in large groups rather than individually, which works out perfectly for us because we also complete parts by grouping them. For example, if we get a bunch of Mill or Lathe Pieces to do at once, rather than work on one at a time and have to rapidly switch between the bits or chucks or *endmill* and much more, it is easier to work on them all at once. **If we had a bunch of Mill parts to do, cut them all on the saw, face them all, shave them all, drill the holes on them all, and do every step you'd do for all at once. It allows us to work more efficiently** and more quickly, even if it annoys the assembly team. Also, a side note about working on the Lathe, the auto lock feature on the smaller lathe for *facing* makes it superior in that department, and the bigger lathe has a less broken *chuck* making it better for all other tasks. **Hence, use the smaller lathe for *facing* and shaving, and the big one for holes.**

There is no "I" in Team

Remember that in the shop, you are not just a one-man crew trying to do everything yourself. One of the biggest problems that shops face is the fact that there are only so many machines readily available for people to use. Because of this conundrum, members may often find themselves with little work to do and lots of boredom if all the machines are preoccupied with a body working on them. To combat this, every member of the shop needs to be aware of this issue. and should take turns working on the machines so that every shop member gets even working experience on the machines. **It shouldn't be a shop lead doing everything kinda scenario.**

What to Do With Your Questions

If you are unsure about anything when it comes to a procedure or drawing, especially in build season, **I wouldn't hesitate to ask a superior such as the shop lead, mentor, or the CADD team.** Questions can help avoid incorrect assumptions of parts leading to another incorrect part for the trash can. This Guide can also assist with questions you may have about shop work.

Wrong Drawing/Piece chart

Starting the year of 2022-2023 we have the wrong piece and wrong drawing part, that way we can get an accurate gauge of where we are skill-wise and where to improve next year. Please if you mess up a part, report it and don't try to hide it, because someone will find out about it eventually and you don't want that on your conscience if that incorrect piece causes our team's demise. More importantly, if you find there is an error on a drawing, you have permission to yell at the CADD, and then add a tally to the wrong drawing list. This chart is on the *shop board*. **Remember: Be nice to the CADD team, but correct them on their errors so they may not occur again the next time.**

Inventory Systems

In a shop with 3 main inventory systems, you'll want to be keeping track of. We have a massive inventory spreadsheet known as the tool spreadsheet. This spreadsheet is updated bi-yearly, and these updates are followed by order lists to replace what tools are low on the shop including but not limited to drill bits, taps, *wrenches*, CNC parts, and more. The second inventory spreadsheet is the *metal rack* inventory, which keeps track of all the metal stored in the *metal rack*, mainly consisting of mill and lathe stock. This allows us to keep track of whether or not we have enough metal to make all the parts or if we'll need Kevin to order more metal as the season goes on. This is updated yearly before the build season in November/December. Lastly, there is the first aid inventory system, which is updated monthly, and should be edited as first aid supplies are used by shop members accordingly. **All three of these spreadsheets can be found on Google Sheets in the team 5401 Google Drive, and should be edited frequently as much as possible as the supplies are used up by members on the team and shop sub team.**

Monday

This website online is how we keep track of progress for all parts and pieces of the robot that are designed by the CADD team. Starting from the year 2023, this online resource has been vitally important in making sure everyone on the team is receiving the same communication on whether or not parts are being completed or not. **There will be a more in-depth video on how to use Monday pinned-in announcements**, but it is essential during build season that when you complete or receive a drawing for a part, someone in the shop will go ahead and update the status of said part on Monday.com. This will ensure everyone receives communication on the progress of parts for the robot so meetings can be planned accordingly. **Also note that drawing and parts progress will be kept track in the shop binder, and it will be organized by date and by part of the robot the part is for.** This is the same binder you are reading this out of.

Onshape

Onshape is the special CADD software we use on team 5401, and as a shop member, you're gonna have to utilize it to its fullest potential as well. This is for a multitude of reasons, one example is for CAMMING, as CAMM and CADD interact a lot with each other so knowing how to use Onshape is vital for using the CNC. Additionally, Onshape allows the shop to understand what the heck we are building through visual representation and can help us better understand what we need to do to get a part made. Lastly, having access to Onshape as CADD software allows us to easily answer questions about whenever we can alter pieces due to shop limitations without the hassle of visiting the CADD lab over and over again! **All shop members should have access to the team's Onshape and should be allowed to view the design and drawings digitally at all times.**

Dinner Breaks and Sick Policy

During the Off-Season, we are lenient with your breaks for Dinner, you can take up to 45 minutes to eat. **During the hectic Build Season, we ask that your break is no longer than 20 minutes and that if possible, eat when there is no work so you don't have to eat once the work pops up**. We are not doing it to be mean, but this policy will increase our productivity, and think of it like this, the faster these parts are done, the quicker it will take until there are no more parts, and the better our robot shall be.

Additionally, **if you are feeling unwell or sick for any reason, don't show up to robotics and tell your shop lead ahead of time (this includes feeling unwell mentally, lack of sleep, etc.) We would prefer if you didn't hurt yourself or get others sick, so just take the day off.**

Respecting the Space

This is another reminder to the shop members that by technicality, Team 5401 does not own the shop, it's the school that ultimately does. For this reason, we must respect all of the machinery as we own none of it, we only have permission to use it on behalf of the school board allowing us. Every machine and all parts of the shop from the tables, to the chairs, will be respected at all times like they are your own. The shop thus must also remain tidy, tools must be put away, and anything moved should be put back where it was found by the end of the night.

Safety Suspension Reminder

This is the final reminder, that we take safety very seriously here at team 5401, and if we ever catch you not following a safety protocol to the point where you are or come close enough to being seriously injured, we will have to suspend you from the shop setting for approximately 1 hour. This punishment may get more severe as the season progresses and as more violations are committed. In the most severe case, you may be suspended from the team for unsafe actions and may be asked not to return. Please take safety as your number one priority while you work here. We intend to send everyone back home with all 10 fingers and toes.

Serious reminder and final note from me however. The best way to become a good machinist and to beat the machines at this war is simply to have confidence and show no fear. The more afraid you are of the machine and question yourself, the less efficient, less skilled, and less advanced you'll be. Accepting the danger and showing no fear to the process is the best way to build your craft. Do not fear the mistakes you may make, instead embrace, laugh, and learn from them! Follow what we discussed here, and just remember the point of robotics is to have fun and to learn and grow through innovation and engineering. Take advantage of this opportunity to its fullest extent, and never allow the idea of safety and danger make you afraid to try and afraid to be a machinist. Anyone can do it with minimal experience, and you should never have to feel as though you do not belong on the team because of this!

Acknowledgements Section

These sections of the shop manual should be updated yearly as new members of the Student Leadership Team take office, as new mentors move into the Team 5401 setting, and as more members join the shop. **Anyone listed in the first part of this section should have full editing privileges for this manual and thus should be allowed to change, edit, or add parts of the shop manual as fit for the shop.**

Shop Lead: Rocky Bibeck

CNC Lead: Justin Skwirz

Safety Captain: Ethan Zhao

Shop Mentors: James Chi and Pop

Certified Editor: Mary Endicter

Members: Calvin Tran, Nishanth Bethala, Sean Hennessy, Jack Carney, Caesar Solis, Syd Simmons, Sidon

Fabrication Glossary/Index

Allen Key: L-shaped hand tools that are mostly used to tighten the pinkish-red mill attachment that is to the left of the left *vise*, allowing one to set a permanent zero. They are stored in the "*Allen Key*" drawer of the mill drawers

Caliper: A digital measuring device stored in the multitude of green, gray, and blue cases inside the top mill drawer. They can be set to mm or inch for measuring, and can be zeroed with the button labeled "zero". You measure with them by pulling on the section displaying the numbers, extending it outward, and pushing it to clamp on a piece to see how long it is. It also comes with a top spike that can be put in a hole, then you pull on the *digital readout* side of the *caliper* to find a hole size.

Chuck: the long silver part that holds drill bits and can be found on the Mill, Lathe, and drill press, and extras are stored in the top mill drawer

Collets (type of chuck): *Collets* are *chucks* that are designed to hold very specifically sized drill bits or end mills, usually indicated on the side of the *collet*. They are primarily used on the mill, are stored in the *endmill* drawer of the mill drawers, and hold either *hole saw bits* or end mills.

Countersinking: A process that expands just the outer part of the hole

Cutting Fluid: The fluid used for the chop saw is stored in the WD-40 Canister with a Red Cap on it. This fluid is ONLY for the chop saw, all other machines utilize cutting oil

Deburring: The process that utilizes small hand tools found in the file/*debur* drawer of the mill drawers. These tools are used to scrape away sharp edges of the metal piece, making them safer and less likely to accidentally cut open a wire

Dial Readout: Little dials stored in the top lathe drawer that are used when *facing* and shaving lathe pieces giving us a manual version of the *digital readout*.

Digital Readout: It is the numbers and buttons attached to the upper left side of the mill and is used to set zeroes and as a guide to see where on the piece you are drilling.

Drawings: The blueprints/instructions we use for the creation of metal parts

Edge finder: A special bit that is inserted into a *chuck* to set a position 0 of a piece, making the numbers on the *digital readout* correspond to the measurements on a drawing.

Endmill (type of chuck): end mills are the big fat drills that are stored in the *endmill* Drawer of the Mill Drawers, and are used when *facing* a shaving, and putting slots on pieces. As the name suggests, they are only used on the Mill and can either be attached using *collets* or are already pre-attached to a *chuck*-like piece.

Facing: The process utilized on both the lathe and mill machines that transform the rough edges created from the chop saw into factory smooth edges. The process varies per machine

Facing bit: Tool you attach to the block of the lathe machine used in *facing* and shaving, the silver ones attach to the big lather, and the small ones to the smaller lathe

Files: Hand tools you use on the lathe to smoothen the edges of pieces, found in the second drawer from the top of the bottom section of the lathe drawers.

First Aid Forms: The first aid forms are an incredibly important part of making sure everyone is safe while working in the shop, as they allow the safety team to be informed of recent injuries to know how to prevent them.

Gorilla Pad: Although not technically the name, these are pads used to scratch off and "clean" metal pieces. They are gray rectangles and are stored in the "Drawer" Drawer

Hexagonal Stock: Stock shaped like a long hexagonal prism that is manufactured on the lathe machine

Hole Saw bit: Drill bits that leave behind a disk after they create the hole, typically used for large holes

Inventory List: An online Google Sheet that notes every type of stock we have stored in either the *metal rack* or metal closet and must be updated when new stock is added or taken out.

Jello Hammer: It's a hammer that has a yellow top to it, making it appear like jello, and is mostly used when clamping pieces into the mill

Lathe Key: A key that is used to lock pieces into the lathe spindle and is shaped like a T, utilized by inserting the hexagonal bottom into a hole on the sides of the spindle and turning.

Metal Shop: The section of the shop where metal is processed

Metal Rack: The rack of Metal next to the field that stores all of the metal we use for building.

Mill Bolt: It is the big bolt that sticks out the top of the mill machine and loosening it allows *chucks*, Endmills, etc to be taken out of the mill machine.

Mill Key: It is the key that is used to unlock the *vises* on the mill and has a hexagonal bottom so it can fit into the *vise* bolt.

Oil: May not be cooking oil, but it is the fluid stored in rapid tap bottles and used for every machine other than the chop saw when performing tasks and cuts.

Parallels: Small thin rectangular pieces are used on the mill *vises* to raise the piece so they aren't resting on the *vise* itself because otherwise we can't drill or mill the piece. They are stored in the mill drawer labeled "Parallel."

Parting Bit: The tool you attach to the block of the lathe machines used for parting, the **silver ones for the big lathe** and the **gold ones for the smaller lathe**. Both *facing* and parting bits are found in the top drawer of the bottom section of the lathe drawers.

Plunger: Stored in black pouches in the top mill drawer, the piece that goes in the *chuck* while *tapping*, has the fat section that goes into the *chuck*, and the mini spike that retracts when pressed.

Round Stock: Metal stock that is shaped like a cylinder and is manufactured on the lathe machine

Safety Glasses: The most important piece of equipment of the shop, because as the name implies they are protective eyewear that can be incredibly helpful.

Shop Board: The whiteboard that is behind the wall holding up the drill bit chart, and is where the wrong drawing and wrong piece chart can be found, it also houses shop reminders.

Shop Drill Chart: A chart that shows all possible drill bit sizes and their corresponding decimal number, it also can show tap sizes for lather pieces.

Slotting: The process of using an *endmill* on the mill to create a rectangular/circular opening that wires and other parts can go through.

Starter bits: Drill bits that are fat and have very small drill bits on either side of them, used when drilling a small hole into a lathe piece to make drilling the larger hole easier. Found in the top left small drawer of the upper lathe drawers.

Tapping: A process used to add screw threading to pieces, making screws go in easier.

Tapping bit: A bit used for *tapping*, found in the bottom right small drawer of the top lathe drawers.

Tube Stock: The stock that is a perfect quadrilateral and are metal pieces to be manufactured on the mill machine.

Twist bit: Standard drill bits that can be found in the drill bit mill drawer and are used for **almost all holes under ½”**. There are 4 cases of them, one for numbers, one for letters, one for fractions, and on with all of them except longer-sized

T-Chucks: Miniature *chucks* we use to fit in *tapping bits*, and are found in the bottom drawer of the upper section of the lathe drawers.

Vise: The parts of the mill that you use to clamp pieces into place with the help of the *mill key*. They can also be found on every table in the *woodshop*.

Wood Shop: The section of the shop where wood is processed.

Wrench: Hand tool used to turn bolts, found in the mill drawer labeled wrenches, most often used on the top screw of the mill for putting/taking out *chucks*, end mills, *collets*, etc.