Dear Potential Reader.

This is a brief list of Rocket Team SOP's (Standard Operating Protocols: think the most simple possible explanations with some machining tools. These won't make you Todd or Pat, but they are a good place to start). The shops use include the GELB machine shop, the Area 51 machine shop and Metropolis.

This is NOT a substitute for trainings by the respective shops and mentors.
This is NOT a comprehensive tutorial of how to use a given machine.
This is NOT indicative of the skill of any individual member, nor meant for advertisement.
This is NOT an excuse to use a machine you don't know how to use and are untrained on.
This is NOT to be used by any other teams without my (Hillel Dei - hdei@mit.edu) consent.
This is NOT formally prepared.
This is NOT the final form of this document (fingers crossed this takes off; no pun intended).

This IS meant to aid experienced members who are rusty with a given machine.
This IS meant to support new members and give them a ball-park idea of when to use what tool.
This IS meant to be an extremely simple tutorial of the most commonly used tools.

The ideal usage case is very niche, but regardless, it is one more document to add to the pile in the worst case scenario.

That being said, please enjoy, and do let me know what can be done better. This is the very first iteration of the SOP program, so obviously there is room for improvement.

Cheers,
-HD.

PS: My voice in the Dropbox video is not indicative of my voice in real life.

MIT AEROASTRO

General Rules, Safety & Suggestions (Todd Shop - 33-099)

TO WORK ON ANY MACHINE IN TODD SHOP, YOU MUST GO TO BOTH HIS MILL AND LATHE TRAININGS. It does not matter if you are trained in other shops. He runs these trainings a few times each semester and upperclassmen will notify new members when they are being run. His shop is for AeroAstro students, students UROPing in an AeroAstro Lab, and students on Rocket Team and/or Design Build Fly. Things being made in his shop should be for an AeroAstro-related purpose; not personal projects,

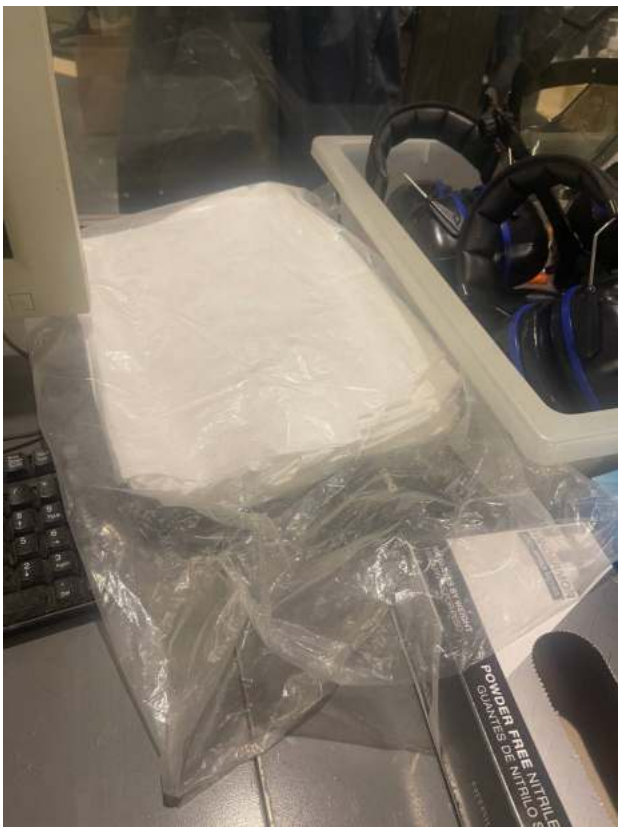
1. When you come in and turn right, safety things are on the bench with sanitizer.

2. Always enter with safety glasses on. There may or may not be communal safety glasses.
3. Next to the glasses are dust masks. These are M95s, blue surgical masks, etc. You must not necessarily wear them EXCEPT WHEN YOU ARE USING THE SANDER. If you are using the sander for over 5 minutes, a mask is highly recommended. The mask chosen is at your discretion.

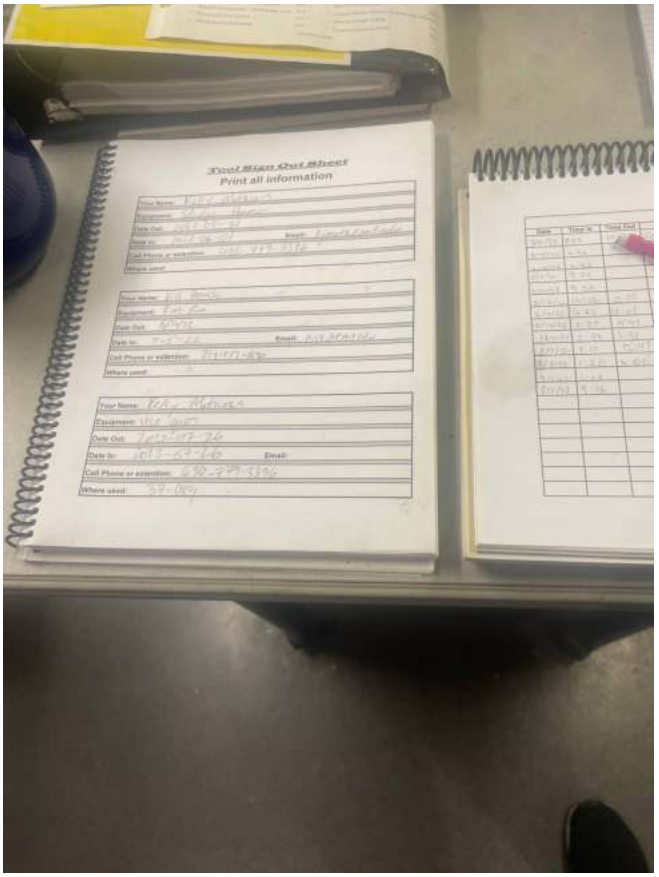
4. There are 2 types of hearing protection: over the ear and inner ear canal. They are equally effective so again this is your preference. These are not mandatory, but if you are sensitive to loud noises, these are recommended.

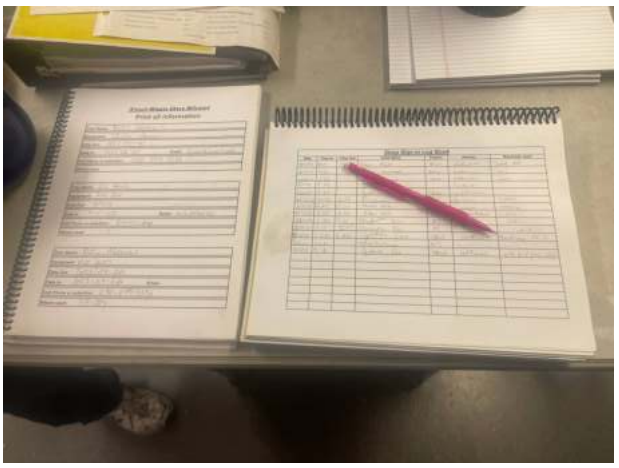
5. With respect to gloves: the latex and nitrile varieties are available. THESE ARE NOT MANDATORY TO USE. The medium option is on the shelf. THESE ARE THE ONLY GLOVES ALLOWED. IF A LATEX/NITRILE GLOVE GETS CAUGHT IN A MACHINE, YOU ARE SAFE. IF A FABRIC GLOVE GETS CAUGHT, YOU ARE SCREWED .

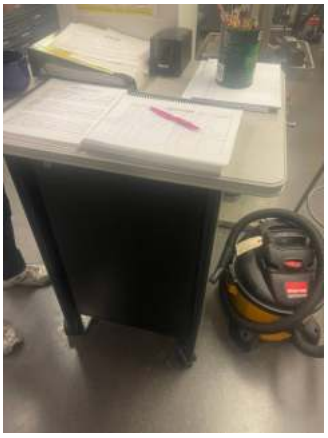
6. There are aprons available and these are also not mandatory. Please do not wear fancy clothes. There is coolant in the shop. As a machine is running, if you have excess coolant and it spills on your clothing, you will need to pretreat the clothes.



7. Once you have your glasses, LOG IN. It is on the table right next to the safety bench. You may have to sign out tools, so PLEASE LOG THIS IN THE TOOL SIGN OUT SHEET. Remember to log the tool back in when you return it. All materials are FREE. The use of the 3D printer, water-jet, etc. Is all free. If it is for a class it is free. If it is for a UROP, you must pay for the 3D printer, and the waterjet. The 3D printer is 10 dollars per cubic inch and the waterjet is 180 dollars per hour. This printer produces in ABS plus and is wind tunnel testable but is NOT a metal printer. The waterjet is charged for run time, not for staying time so you often end up paying relatively little. If you are making anything 2D, waterjet it first and THEN mill it or lathe it. The waterjet is a great time saving tool. With respect to stock, for class work, it is free. For a UROP you are expected to have it or pay for it yourself.







8. You are NOT allowed to have earbuds and it is NOT a shop for hobbyists. This is a SHOP NOT A HOBBY SPACE .
9. In terms of clothing, please wear shoes, boots and NOT SANDALS, CROCS, SLIPPERS, etc. It must cover your foot entirely and be stable. You cannot fall off of or out of your shoes.
10. You may wear trousers or shorts. Trousers keep child out of your shoes. Short sleeved shirts are fine. Long sleeve shirts that are a tight fit are acceptable. BAGGY/LOOSE FIT CLOTHES ARE UNACCEPTABLE. JEWELRY IS UNACCEPTABLE. IF YOU TAKE ANYTHING OFF, DO NOT LEAVE IT ON A MACHINE. KEEP YOUR HAIR TUCKED BACK AND PLEASE TAKE OFF YOUR WATCH. It can easily be scratched by carbide, ice bitumen, etc.

11. The drills and taps you need are on the table next to the lathe and mills and drill press. The taps in the wood holder are used for the hand tap. The taps in metal boxes are used for the lathes and mills. To find the right drill or two size, you may google it or use the conversation table. Note: you may also find gloves at the back of the shop.

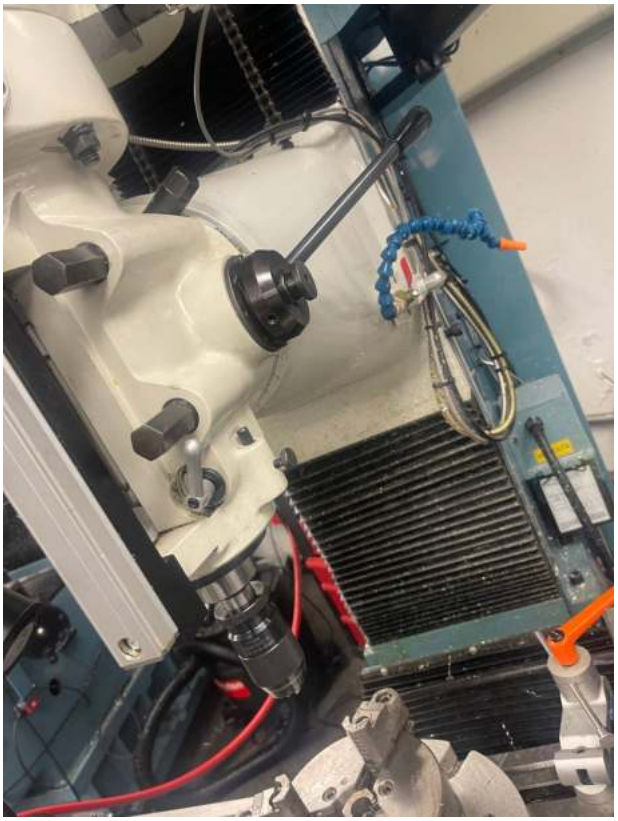
Operation of the Mill

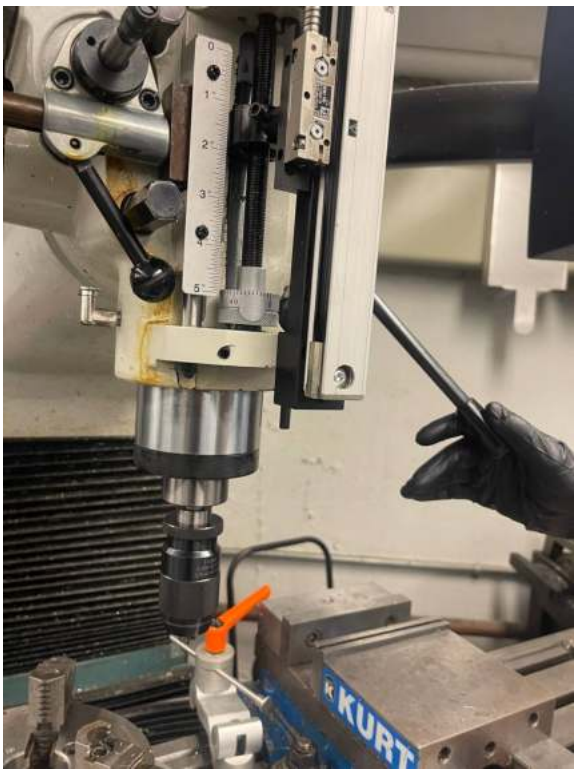
1. Todd used a vertical bed mill. In other words, the bed is stationary and does not move in the z axis. The z axis is the head that moves up and down. Within that head, you have a quill with 5 inches of travel, but the head itself has far more travel.

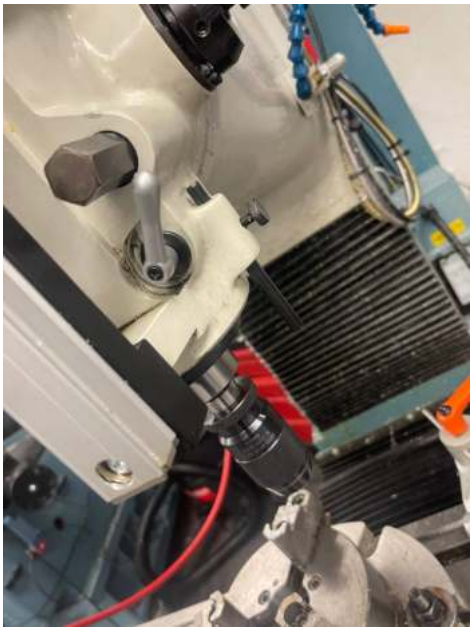
2. Todd's mill used a windows based touch screen interface. DRO means Digital ReadOut and it just tell you where you are in the X, Y, and Z axes. There is no lever for changing the speed, you tap the screen where the number is displayed, you key in the value with the number pad and then you hit ABS SET to absolutely set the value. You can only enter a value between the lower and upper bounds stated at the plate of the machine. Once this is done there are different modes.
3. There are various programmable modes of the machine. You may input these in from SolidWorks as a CAM file, so you need to post-process the CAM file before entering it.
4. Another way of doing it is that you may program the machine directly with a CAM'ed cycle. These let you do different things on your part based on some answers to simple questions. It does the math for you.
5. Lastly, the JOG feature moves the head up and down. YOU CAN ONLY MOVE THE HEAD BUT THE JOG KEY DOES NOT MOVE THE BENCH ITSELF. Once you enter JOG mode, if you hit Z,Y or X, the motion of the axis will ALWAYS be in the positive direction. Z let's it default to positive I.e. up. If you want it to go in the negative direction, YOU MUST SET THE FEED RATE TO BE NEGATIVE. HIT THE FEED NUMBER, press the +- key and this makes your value negative. ALWAYS PAY ATTENTION TO YOUR PART WHILE JOGGING. PLEASE HIT RETURN WHEN

YOU'RE DONE JOGGING AND DO NOT FORGET.

6. Inside the mill is a 3HP motor that drives the spindle in the quill. If you want to keep the spindle in position, pull the small, silver brake lever DOWN. To turn the spindle on and off there is no longer a 012 switch. If you want to turn it on, you just hit FWD (forward) or REV (reverse). By default we almost always use FWD. The hand lever (the big black lever) controls the Z direction and the black wheels control the X and Y directions. The tool is kept held securely in place by the Gold air wrench. This system only works when the SPINDLE IS FULLY RETRACTED AND THE BRAKE IS ON (I.e. the brake lever has been lowered: in the image to the right, the lever is the long black one and the brake is the small, silver-ish one just below the hex and both are currently in a raised position).



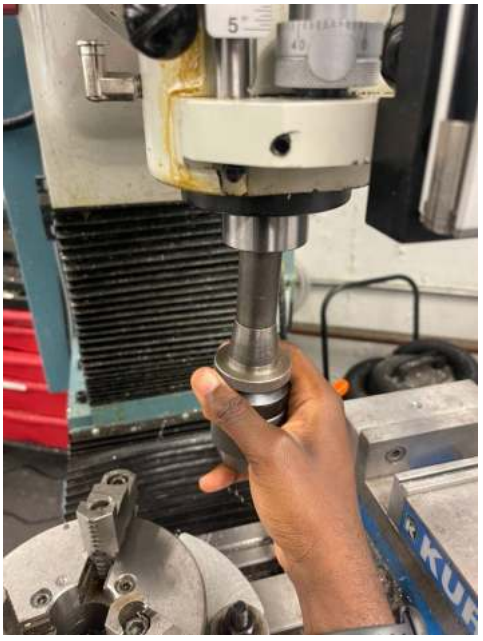


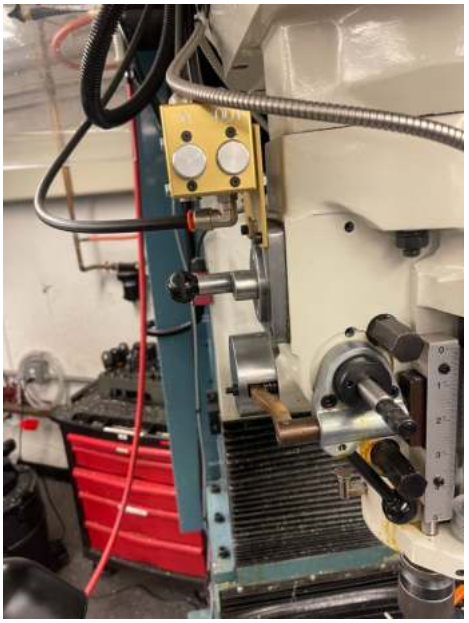


7. To remove a tool, once the spindle is fully retracted and the brake is on, GRAB THE SPINDLE AND NOT THE TOOL AND PRESS THE OUT BUTTON. WHEN YOU THIS, DO NOT PULL THE TOOL. GENTLY GUIDE IT OUT WITH YOUR HANDS . (A drill chuck only holds drills not end mills.
8. When inserting a tool like a drill chuck DO NOT HOLD THE BIT OF IT THAT IS GOING INTO THE QUILL. When putting a new tool in, INSERT, ALIGN, TWIST to align the track. of the grove on the tool with the track in the quill. Not that it's up and in and your fingers are all away from the shoulder, press the IN BUTTON. You will hear 2 tones: a HIGH PITCH AND A LOW PITCH . Once you go from high pitch to low pitch. You CAN NOT MAKE IT GO IN ANY TIGHTER BY PRESSING AND HOLDING. Like a drill, the end mill is sharp on

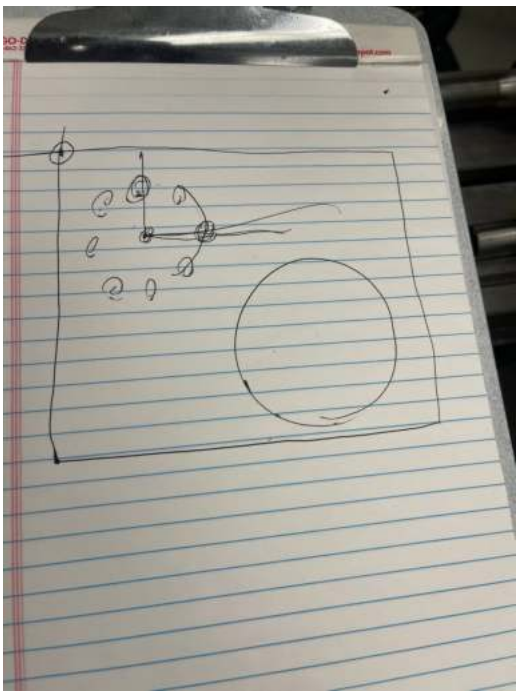
the end, but flat. Unlike the drill. It has cutting edges and when they meet, it is extremely sharp. When inserting an end mill, KEEP YOUR FINGERS AWAY FROM THE SHOULDER AND THE TAPER. Put it in, and just like with the chuck, you twist to align in. All the tools work the same way. (The image on the preceding page demonstrates the WRONG way to insert a chuck. The image to the right demonstrates the CORRECT way).

9. When you go to remove a tool like an end mill, once again, DO NOT PULL. OCCASIONALLY, IT WILL GET STUCK BUT DO NOT PULL (may cause injury). If something gets stuck, call Todd. Press out, and it'll come out. There is a threaded bar inside that spins as you press the IN and OUT buttons.



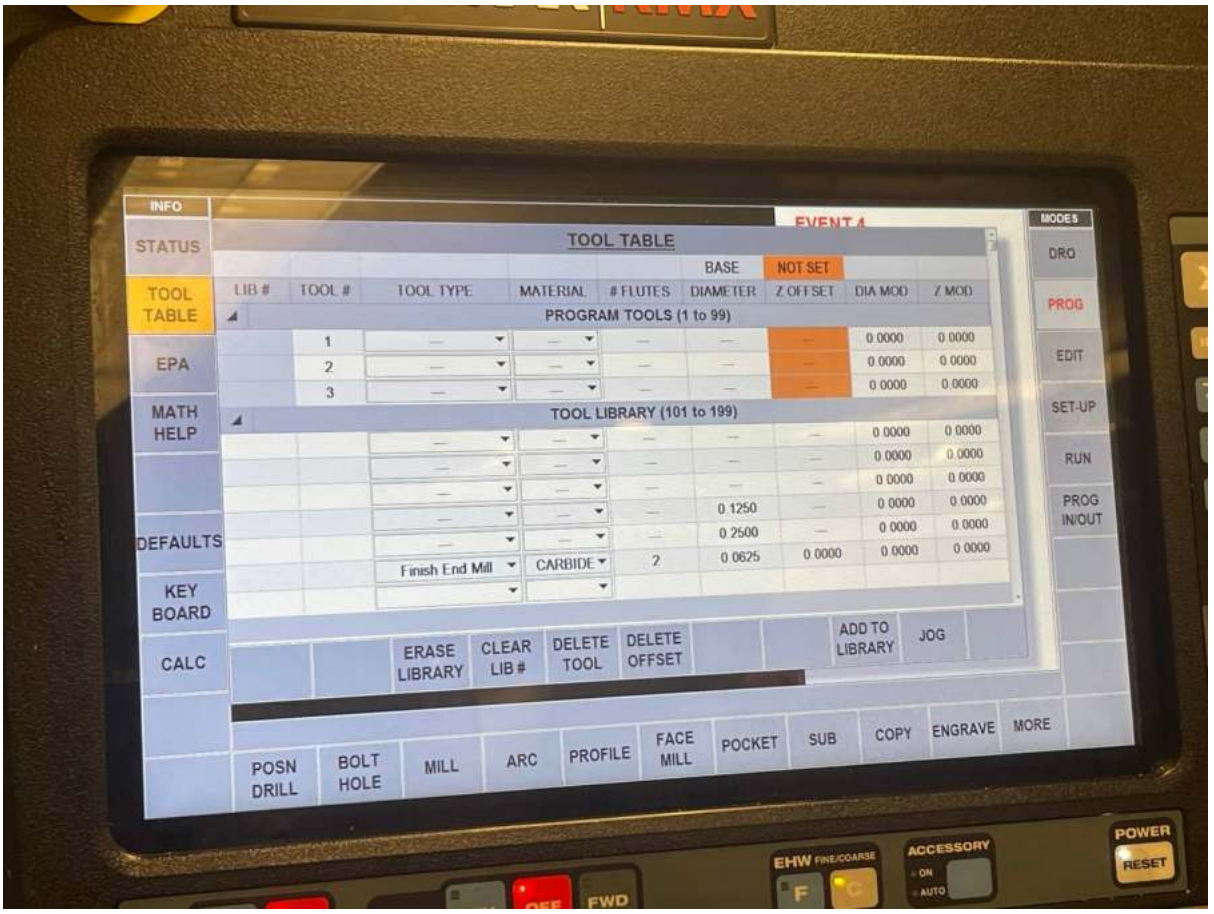


10. If you want to program a part to cut, the ORIGIN IS IN THE TOP LEFT. if you hit the PROG button and you see an image/program already there, check with Todd if you can erase it. You hit one button down to go to EDIT, ERASE PROGRAM, YES. Once you do that correctly you should see the red and white screen. Then you hit the PROG button to begin a new program. If you intend to save it, else if it's a one off thing, hit BEGIN/swipe from RIGHT TO LEFT on the screen.



11. Looks the possibilities of what you want to do at the bottom the screen. First, hit POSITION DRILL. The TOP LEFT corner is the origin. Type in the coordinates with negatives and positives as they are Cartesian, then tap in the X, Y and hit ABS SET. Next, set the RPM. Feeds and speeds take a long time to master, based on the size of tool, milling, drilling, reaming, etc. Ask Todd or use something you already know.
12. You will then enter the tool number. Then you hit ABS SET. Then for the next operation, it will refer to the coordinate of the previous operation as the center point. Next, since in this example we are producing a bolt circle, we go to bolt hole, and enter the number of holes. Then it will ask you the center. You can renter the previous coordinates OR YOU CAN USE THE INCREMENTAL SET (INC SET). This is how much it should shift the coordinate by hence the delta of sorts.
13. If you hit the INC SET, the value will default to 0, hence the last known coordinate you were at for the X or Y or Z you're working on. Radius refers to the radius of the ENTIRE CIRCLE NOT ONE INDIVIDUAL HOLE AND NOT THE DIAMETER (refer to figure below for clarification). Obviously, if you want to change an individual hole size you must change the tool. FOR ANGLE, it starts at 3:00, counterclockwise is in positive and all values are in degrees. So for example 12:00 is 90 degrees, so if you want your first hole at 45 degrees anti clockwise of 3PM you hit 45 then ABS SET.
14. All holes are EQUALLY SPACED by default. If you want to make a new set of holes you must trigger a new event. The RPM automatically populates the last known speed so you hit ABS SET.
15. If you made a mistake, just tap the field to correct your error. The system defaults to the LAST USED tool number. It will remember the tool name you put in and you have to populate that page. If you have the same tool and you labeled it twice, you'll have

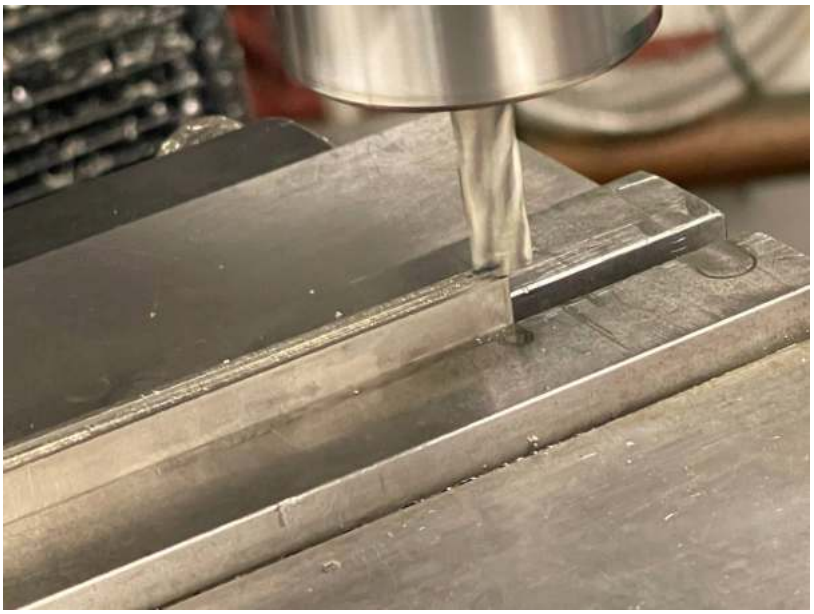
to enter the second name.



16. When done with everything, hit ABS SET. The XY field is 55X and 14Y. If you wanted to make a large hole that is 3 inches for example, since you don't have a 3 inch drill, you will use an end mill to cut out a circle.
17. To cut out a hole you hit the POCKET button. Select the shape, then enter each value and hit ABS SET. Take care and note that DIRECTION matter. When you cut in the inside, the tool spins clockwise and the table moves counterclockwise relative to the tool. This improves BOTH surface finish and size.
18. When milling outside, BOTH THE TOOL AND TABLE MOVE CLOCKWISE. The top doesn't move per se but it rotates. Direction on the interface is stated with respect to the TOOL. FIN CUT MEANS FINISH CUT. DO NOT INPUT A FINISH CUT. Make sure it is 0 and hit ABS SET. Set the RPM to populate from the last known. For FEED RATE PER MINUTE, if you are milling or turning anything besides steel, start at 5.0. If you're working on steel, use 1.0. The harder the number, the slower you go. I'd you want to set to work on a different tool, again, change the tool number.
19. The different tools are in different colors. GREEN AND YELLOW AND NOTHING BLINKING ARE FINE. Now you hit TOOL TABLE. If this is a single time job, you need NOTHING BUT DIAMETER. If this is a repeated job it is HIGHLY ADVISED THAT YOU FILL IN AS MUCH INFORMATION AS POSSIBLE . In theory, you could leave everything back because the drill moves from center point to center point. You could hypothetically fill in nothing AS LONG AS YOU PIT THE CORRECT SIZE TOOL IN. In some cases, you must enter in the diameter of the tool AND NIT THE TOOL HOLDER. ALWAYS MEASURE TO BE SURE. To change units, go to STATUS AND SET TO INCHES. If you tap tool table again to exit, and you swipe to and from

EVENTS BY SWIPING ACROSS. As it reviews the information, a cycle flashes purple. So long as you select the correct hole size and the correct tool diameter everything is fine.

20. To proofread, SET-UP button is hit, then TOOL PATH.
21. Use DRO AND AN EDGE FINDER TO PROPERLY SET EDGE.
22. Use RUN and the clicker. At each step, once the step is complete, you must do some manual tasks ie going down, lock in place manually and change the tool manually. It is ONLY the lateral 2 axis XY motion that is fine automatically. In short RUN, START, CLICK, LOAD THE TOOL, HIT GO, and do the manual stuff yourself. If you want to pause, hit OFF is pause and STOP is completely halt. Start the spindle by hitting FWD.



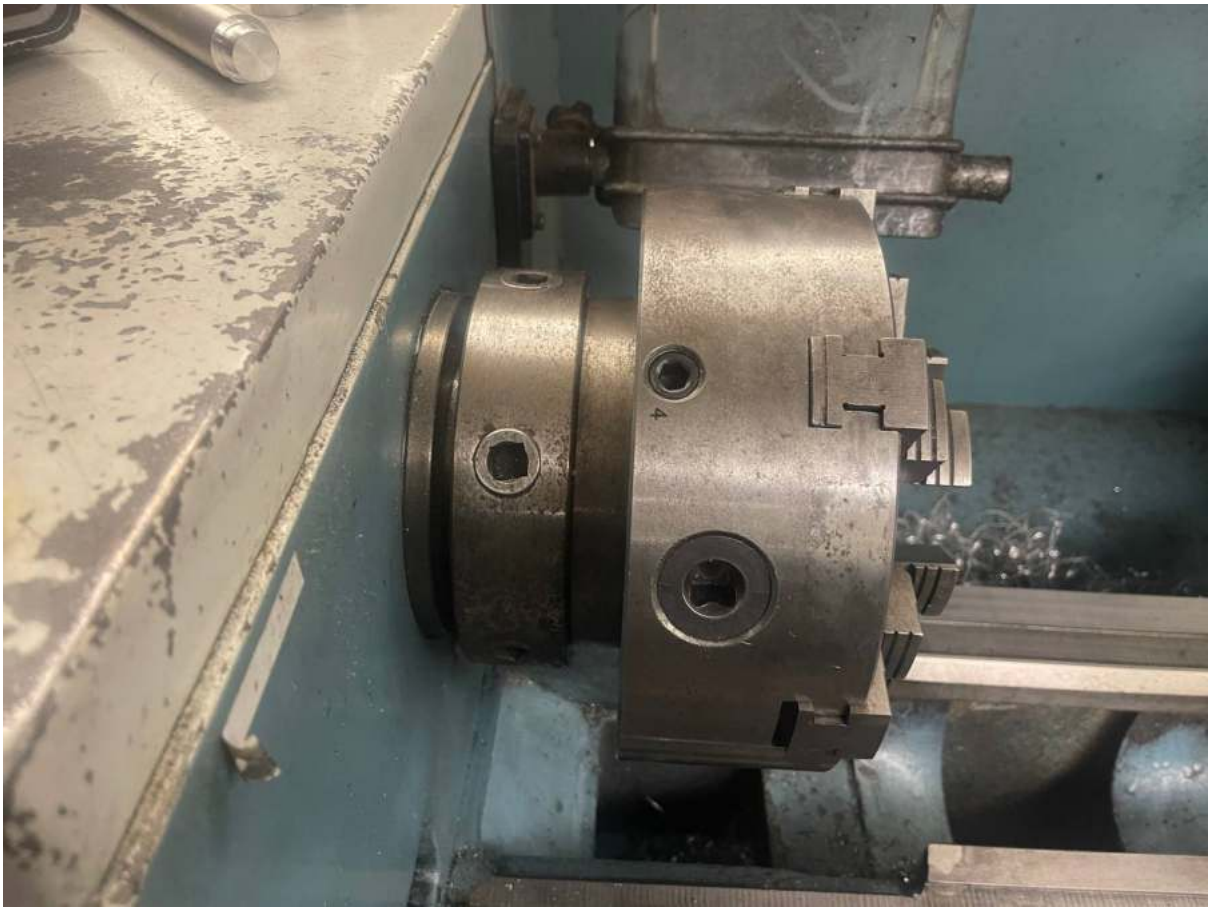
Operation of the Lathes:

1. There are 2 lathes, the small one and the big one. The one you use depends on the part size. We use horizontal lathes and these are CNC capable. The lathe has only 2 axes whereas the mill has 3. The lathe has a readout and an ESTOP button. The lathe is the only machine that requires that you hit the ESTOP button. You twist the ESTOP button on both the control panel and the second one in the wall of the lathe to begin. Go to MODE then DRO. Move to SERVO ON and ensure that it is lit up. If it doesn't repeat the process of hitting MODE then DRO. Do NOT hit the return button here. In the head of the lathe is the transmission and when you change the speed of the transmission with the 2 levers the machine must be off. Do NOT change the speed of the machine while it is running. If you do this you will seize the transmission and jam the gears. The bottom lever has the ABC on the bottom lever to select the row. Then, move the top lever to the desired speed within that row. Pop out the lever and rotate it to set the speed. If it is jammed or refusing to move, wiggle the chuck slightly ie the big thing with teeth and jaws.

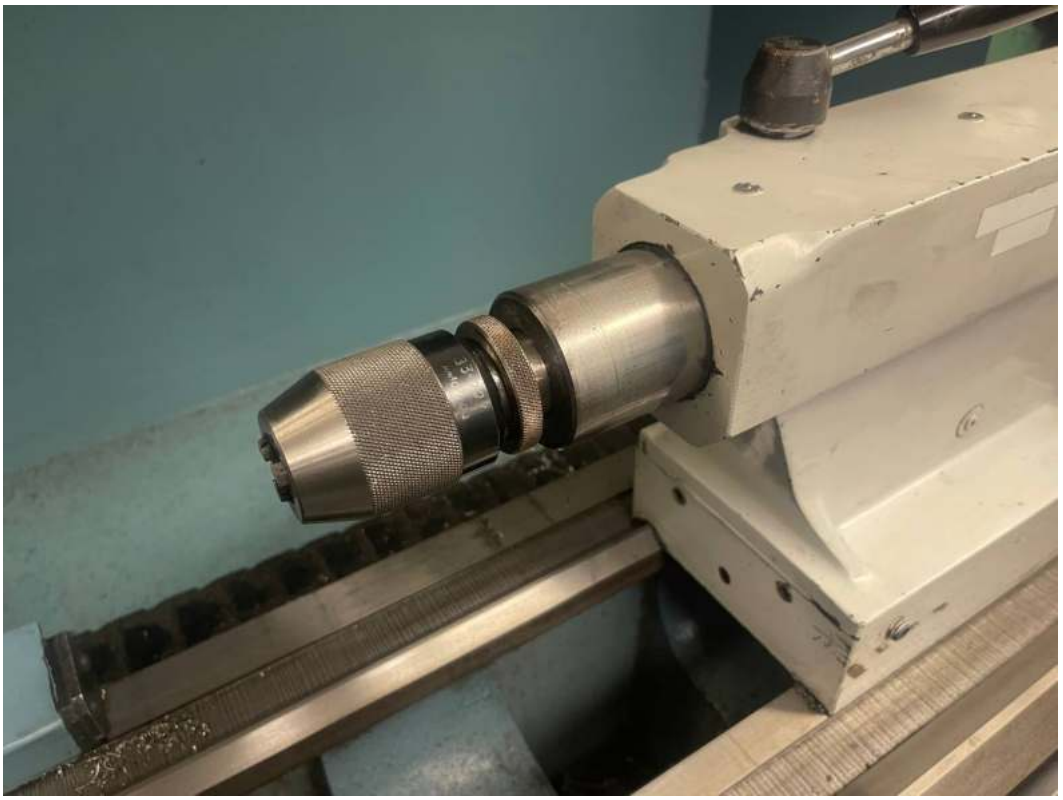
2. The chuck is mounted on a through spindle, so if you were to run a tube through the middle of the chuck and spindle it would pass unobstructed and right on through. DO NOT WORK ON ANYTHING 8 inches longer than the back end of the machine. Remember the spindle he connected to the chuck. The spindle is part of the machine

itself. The chuck is NOT. The chuck is just a tool mounted on. The spindle is on the left and the chuck is on the right.

3. Next comes the saddle. The saddle has its X and Z direction. X is diameter wise and Z is length wise. The knob with a rounded tip is the JOG OR RAPID MOVEMENT TOOL. It only goes in one direction. The big, stained, red tool post is what you mount tools on. Use the drawer for lathe X on Lathe X.
4. The tool fits onto the tool post as is so when in the images below. When the tool is fit on, lock the knob by twisting it clockwise. Once this is done, pay attention to forwards or reverse. 0 means off, F means forward and R means reverse. Forward let's the spindle spin counterclockwise and reverse let's the mill spin clockwise.



5. Next is the tail Stock (the big white thing at the far right end). This will typically have a drill chuck mounted into it. It can itself move forwards or backwards. The wheel at the behind/ far end pushes it in our out. The knob at the top is used ONLY FOR A LIVE CENTER. The knob in simple terms locks the wheel. Try to use the colored or marked point on the drill chuck as a gauge. If you turn beyond the green line, it will push the chuck against a pin inside to eject the chuck.



6. When using the chuck, there is a chuck key to at moves the teeth of the chuck in or out. They are spring loaded so both hands are needed. If you do not maintain pressure, it pops out. Tighten the lathe chuck as much as you cos to prevent inaccuracies.
7. Bring the tool up to but NOT ON THE PART. Leave room on the X and Z clearances. Then turn the spindle on with the F0R knob. Use the turn/face tool then bring the X diameter close to the face to where it's going to touch it and bring the Z direction VERY CAREFULLY towards the part and gently touch it. Once you touch it, let go of the Z direction and pull it off it in the X. Then you hit ABS SET to zero that point as your new Z = 0.
8. Cut into the part in the X direction by a very small amount and completely cut the tip of the surface off. Once it's completely cut the tip off and made it perfectly smooth. Re-zero the Z and make it as your

new reference point. In other words we cut the tip off and make the end as parallel as possible to the outside dimension that we intend to turn. You need to zero the Z TWICE. Always do this in increments of thousandths of an inch. EVERYTHING SHOULD BE IN MOTION. Never ever touch the carbide tip on a non-moving part.

9. We do the same thing to the X but with a slight difference. Move in on the Z diameter. Very gently move in X wise and gently touch the circumference with the carbide tip. Then, move out in the Z direction. Then, use calipers and read the diameter while you are on the part. DO NOT MOVE THE CALIPERS OFF THE PART (in the image below we are not measuring radius or diameter, but rather the height of the cut) when measuring. Take the calipers value and enter That as your X. It may seem confusing and counter intuitive to use the diameter as the X value but the machine does all the work for you. It knows you have entered the diameter and will use the center of the spindle as zero. You DO NOT ENTER THE RADIUS. It is only the diameter you need to know with respect to the center of the spindle.
10. Now to go to a specific point use the MANUAL GO TO button. Input the correct coordinates. The go to will act as a positive bound stop. You can NOT go behind this point that you set as your go to. Verify that it doesn't go behind that point. Then, take about 30 thousandths of a cut.

11. Pull in then back out. ALWAYS LEAVE ABOUT 5 thousandths for a Final Cut. Then, for the final add at the final diameter, DECREASE YOUR FEED RATE ie the rate at which you are moving along Z ie lengthwise of the part. Do Not back out on the Z but rather back out on X. For all the others you go in Z out Z but NOT for the final pass. This is how you take off length OR diameter in the lathe. For the X diameter clockwise is in counterclockwise out. For the Z

counterclockwise is in and clockwise is out. Z is the bigger wheel.

(Image above shows a catcher in use)

Tangents:

12. It helps to turn on the lights with the light switch.
13. Left is grooving tool for outer diameter and right is cutting tool in the image below. Works close to the chuck.
14. Turn the machine on. Move out in the X. Move close in the Z. Repeat the process. If you want a specific size cut, you can use calipers to measure. You may use a parts catching rod if necessary to capture parts.
15. You may have to loosen the chuck's grip on the part a little bit when using the caliper method.
16. Note: return out of the go to mode and then then cut in the X direction. BE VERY CAREFUL TO NOT ACCIDENTALLY KNOCK A WHEEL . Clear a little space out
17. To deburr, you use the debarring tool to remove/scrape and put a slight amount of pressure to remove scrapes from machining. When debuting, you rotate and revolve.


PUNCHES
and
CHISELS
CENTER DRILLS
DEBURRING
TOOLS

Operation of the Bandsaws:
1.) Use the black knob to raise the saw guard so the material can slide underneath the saw freely.

2.) Thoroughly clean the surface of the material so that slides freely and there is no tack/grime/debris/gunk/dirt underneath
3.) Twist the red knob. Push the green button. Adjust the speed knob. For cutting speeds, ask Todd. The machine by default runs in low gear.
4.) you can use your fingers or you can use alignment wood.

5.) When done, use the brush and aluminum scoop to clean up.

6.) Deburr as necessary at a 45 degree angle with a file.

Tangents:

1. When using a collet put the tool just shy of the release use the photo below as a guide: do not let your fingers go above the tool tip
2. With a material we hard as Inconel, go slow in RPM and feed rate
3. Make sure you do not measure on chips: air gun them first.

Operation of the Waterjets (Metropolis):

Unlike the prior walkthroughs, there is documented recording of a full run-through, in addition to the written instructions. These are to be used in unison with the laminated instructions provided in Metropolis on Feb-5-2022.

(I'm so sorry about the poor audio quality at some points. I intended for this to be helpful since it is from the PoV of 2 users who were quite rusty at the time.)

Live Run: https://www.dropbox.com/home/Rocket%20Team%20Tutorials

EXTREMELY Summary of Procedure

1. Open the file in the sequence: Layout > Make
2. Use the Big, "S" for select
3. Follow the sequence: All > Quality
4. Add Tabs where they are necessary
5. Rescale the part with size
6. Use the Lead V0
7. To insert tabs, select the entire path. Be sure to select EVERYTHING. Then follow the sequence: All > Quality (Set to selected at 3)
8. Use CTRL + T to inset a tab BEFORE Auto Path
9. Use the Auto Path Quick
10.

Notes:

1.) ALWAYS cut the interior path first.
2.) ALWAYS double check that the cutting is being done on the inside first.
3.) GREEN indicates the watercutter is just travelling. RED indicates there is actual cutting at that location.
4.) ALWAYS keep an eye on the sand level. Pressurized sand is how the waterjet cuts. If this urns out, you will stop cutting!
5.) Clean up after yourself when you're done. Specifically, clean out the garnet trays.

Waterjet Machine Operation

How to use ProtoMax Make software and operate the waterjet cutter machine. See the ProtoMax Layout guide to generate toolpaths from your geometry.

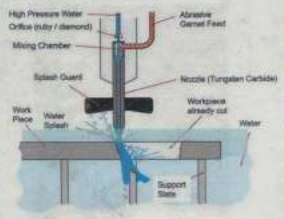
TOOLS:

- Waterjet Cutter (1)
- Dial Caliper (1)

Step 1 — Familiarizing Yourself with the Waterjet

- Familiarize yourself with the terminology specific to the waterjet. They will be used throughout the rest of this guide.

Step 2 — Allowable Materials



- Most Metals and Plastics may be cut
- Glass and Ceramic may be cut
- ⚠ No Lead or Lead Alloys
- ⚠ No Copper or Copper Alloys
- ⚠ No Zinc or Zinc Alloys

Step 3 — Check garnet level

- Check to see that the garnet hopper is full to the level of the sieve. If not, add garnet from the bucket near the computer.
- Be careful not to get the garnet wet or contaminated - it will clog if it is wet.

Step 4 — Check water level in water drum

- The water drum can hold up to 30 minutes of waterjet waste.
- Do not fill the drum above 2/3 full (second line on the side.
- Ensure that there is enough room in the drum before you begin setting up.
- If you need a new drum, ask a mentor to help.

Step 5 — Turn on the waterjet

- Before you can open the ProtoMAX Make software, the waterjet has to be on.
- Turn the switch from the "O" to the "I" to turn on the waterjet.

Step 6 — Open ProtoMAX MAKE

- Log in to the waterjet computer with your MIT Kerberos credentials.
- Open ProtoMAX MAKE after the waterjet has been turned on (otherwise there will be error messages).

Step 7 — Check waterjet bed

- Check that the waterjet bed is **empty and there are no clamps on the bed** before you start moving the nozzle or making any adjustments in Make.
- Flip up the nozzle cup such that it looks like the second picture.

Step 8 — Home the nozzle

- When you open up Make, there will probably be a red error message in the top left that reads: "Machine needs to be homed. Click here to home machine."
- Click that red message box to home the nozzle. That will zero the X and Y for the "Distance from 'User Home'".

Step 9 — Place material in waterjet bed

- Place the material inside the waterjet bed with the corner aligned with the corner of the bed.

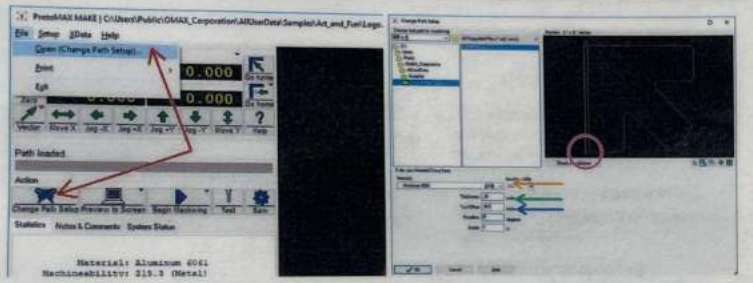
Step 10 — Secure the material

- Use the clamps on the edges of the waterjet to secure the material you are cutting.
- If the clamps don't reach your piece, use the lead blocks next to the computer to secure your piece.

⚠ Make sure the clamps are holding the material flat - not bowed up or down

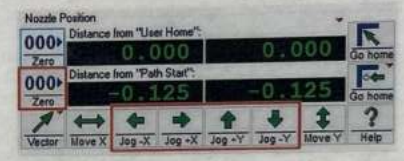
⚠ **CAUTION:** Make sure the waterjet nozzle does NOT run into the blocks or clamps during the job.

Step 11 — Open your part file



- Change Path Setup
- Select your material
- Set the thickness of your material
- Set the Tool Offset. Default is .015"
  - Tool Offset can vary from .010 to .060 depending on material, thickness, and nozzle wear. If you've done a test cut, enter the appropriate value here.
- Note where the Path Start location is relative to the drawing

Step 12 — Jog the nozzle over your material



- Use "Jog +/- X" and "Jog +/- Y" to move the nozzle in the X and Y direction, respectively.
- The arrow keys on the keyboard also jog the machine
- Move the nozzle over your material. If there are any height variations, go to the highest point.

Step 13 — Set the nozzle height

- Lower the nozzle by loosening the knob on the side of the nozzle attachment. **Slowly** bring down the nozzle such that the tip of the nozzle touches the spacer tool and tighten the knob.
- **DO NOT** slam the nozzle into the spacer tool.
- Press the top of the nozzle slide block toward the gantry (right) to help it move move smoothly.

Step 14 — Jog nozzle to starting point

- Move the nozzle to the point where you want your path to start. Cutting will start from this location.
- Save the start position of the path by clicking the **Zero** button to the left of **Distance from 'Path Start**.
- Note the numbers in **Distance from User Home**. This can save you in case of errors later.

⚠ **IMPORTANT:** WHERE THE NOZZLE ENDS UP AFTER YOU JOG IT IS WHERE THE PROGRAM WILL START RUNNING.

Step 15 — Fill the waterjet bed with more water

- Make sure you have completed these three steps:
  - **1.** the piece is secured.
  - **2.** the nozzle height is properly adjusted.
  - **3.** the nozzle is at the appropriate starting (X, Y) position.
- Use the water hose to raise the water level inside the bed to cover at least 1/4" over the top of your material.
- Push the drain strainer up and down by hand until it matches the water level, 1/4" above your material.

Step 16 — Flip down nozzle cup

- Flip down the nozzle cup once your nozzle is at the appropriate starting location.

Step 17 — Lower the lid

- DO NOT IMMEDIATELY PUSH DOWN ON THE LID.
- Lower the lid by releasing the latch before you start running the job.

Step 18 — Start the job on ProtoMAX MAKE

- When you're ready to start the job, click **Begin Machining**. A pop-up that looks like the first image should appear. Click **Start** to start the job.
- **KEEP YOUR MOUSE ON THE PAUSE BUTTON IN CASE ANYTHING HAPPENS.**
- There are additional options to pause if you right click on the "Pause" button, as shown in the second image, i.e., you can pause at the start or end of the next traverse instead of in the middle of one.

Step 19 — Watch the Cut

- Pause the cut and adjust your setup if any of the following happen:
  - ⚠ The nozzle is about to collide with clamps or weights
  - ⚠ Small parts get lodged above the surface of the material
  - ⚠ Water shooting out of the garnet feed line
  - There is more splashing and noise than usual (often indicates a garnet clog)
  - The material starts to move

Step 20 — Removing the material afterwards

- **Wait until the waterjet is completely done running to remove the materials from the bed.**
- Raise the nozzle head to get it out of the way. Tighten the nozzle head such that the **nozzle is NOT touching the water**.
- Unclamp the piece by pulling up on the back of the clamp or remove the lead blocks used.

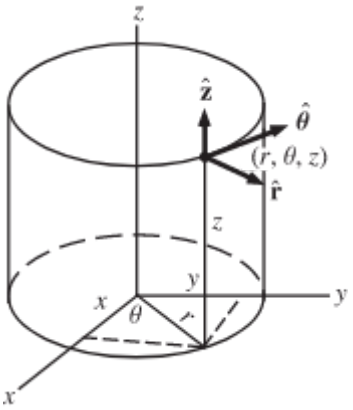
Step 21 — Shutting down the Waterjet

- Log out of the computer.
- Clean up stock and scrap.
- Turn off the waterjet.
- Raise the nozzle to its highest position.
- Remove both spent garnet catcher trays, pour water into the tank, and pour garnet into the trash.
- The lid should be left open.

This document was last generated on 2019-08-09 10:48:23 AM.

Operation of the Radial Indexer/Rotary Indexer (N51):

This is a tool used in conjunction with horizontal bed mills, such as those at N51 in order to place holes laterally. As you rotate the gear of a radial indexer, it causes rotation along a fixed cylindrical axis. In other words:



(Source: Wolfram Mathworld)

In other words, as you rotate the gear/lever of the radial indexer i.e. as you rotate (17) along the circumference of (7), that causes the object to remain fixed in the z axis, but theta changes as rotation occurs.

To describe how radial indexers work in further detail, you must count the number of holes in the track along which the rotating lever moves. I.e. on the item indexed (no pun intended) as 7. We will refer to 7 as the needle henceforth. A complete rotation typically means you will end back right where you started. Thus:

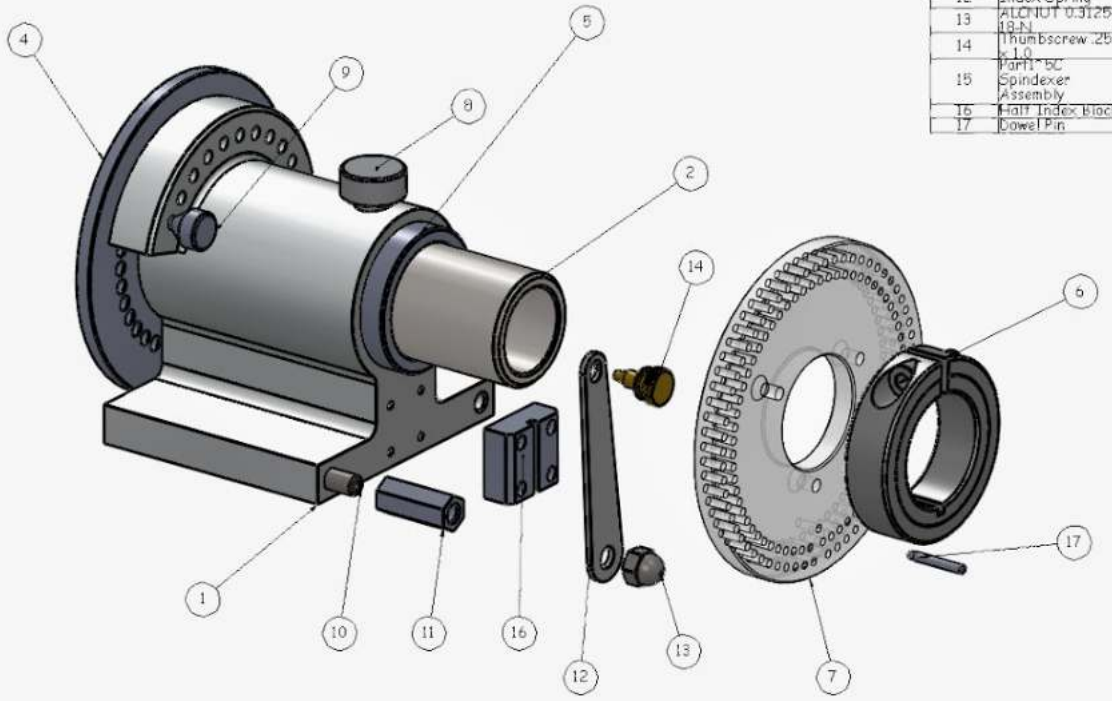
Number of Holes = 360 Degree Rotation

Knowing this, you can rotate the needle a precise amount to cause a specific angular change in the object. For example, suppose I have a cylindrical tube and I want 2 holes that are 180 degrees separated i.e. on opposite sides of the circumference, but a line connecting them would be the diameter. I have an indexer that has 40 holes along the active thread. I will rotate the needle 20 times to reach the 180 degree mark, as each hole corresponds to a 6 degree change in theta.

DO NOT BE CONFUSED BY THE EXCESSIVE NUMBER OF HOLES. YOU ONLY CARE ABOUT THE TRACK ALONG WHICH THE NEEDLE PASSES. IF A HOLE IS NOT ON THE TRACK, YOU DO NOT COUNT IT.

DO NOT SKIP THESE PRELIMINARY STEPS:
Use an edge finder and a deflection gauge to make sure your workbench is properly set up. We aim for 5 to 10 tau of error, and although this may seem annoying, tedious, or inconsequential, the result is extremely noticeable.

Source of Image Below: (http://oxtool.blogspot.com/2013/11/5c-spindexer-indexer.html)



ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	Body Casting		1
2	Spindle Tube		1
3	Front Clamp Ring		1
4	Index Disc		1
5	Rear Clamp Ring		1
6	57445k78		1
7	Index Disc Blank		1
8	Locking Knob		1
9	Lock Pin		1
10	SSCDPSKT 0.3125-18x1-HX-N		1
11	Standoff		1
12	Index Spring		1
13	ALCNUT 0.3125-18-N		1
14	Thumbscrew .25 x 1.0		1
15	Part - 5C Spindexer Assembly		1
16	Half Index Block		1
17	Dowel Pin		1

5C Spindexer Indexer Assembly

CNC Machining: