

Brief Procedural Overview: Minitch CNC Mill

Last updated: July 2014
Chadd Armstrong
Remcho Research Group
Department of Chemistry
Oregon State University

Introduction :

The process of designing and fabricating a piece of equipment is a multi-level and reiterative activity. Often, the best approach is to catalog what materials and tooling you have available and work backward from there to see how close to your ideal design you can get within a reasonable level of time and efforts. Additionally, tuning the process to produce a finished result requires experience and a certain artistic sensibility. Just as with organic synthesis, there are many ways to achieve an end goal but only one will make sense given the resources at hand.

WARNING: The below is only intended as review for properly trained students/faculty and is by no means considered a comprehensive instruction manual. Observe all safe handling and operation guidelines.

Design Considerations:

Tooling, also known as “cutters” which encompasses all objects that are used to cut material, must be considered before any design is attempted. A designer knows the dimensions and geometry of the available tooling and from this will draft the part accordingly.

An example: A 50mm x 50mm chunk of PMMA needs a channel cut laterally at a depth of approximately 2mm. The constraints of the design require that the channel have a cross section of 4mm which might lead the amateur designer to draw a square channel. The available tooling contains a 3.125mm diameter cutter and the next size down is a 0.3125mm cutter.

While the 0.3125 is capable of cutting a 2mm wide channel, to do so would require multiple passes but could not, in fact, cut to 2mm deep as the cutting length is only 1.5mm which would lead to breakage or at least damage to the part.

On the other hand, if the 3.125mm cutter were employed not only would it cut the channel in one pass but also at the finished depth of approximately 1.28mm saving time and potentially averting damage to the equipment, cutter, material or even the operator.

Summary of Steps:

1. Tally the materials, tooling and assess the design constraints of your project.
2. Draft the part in CAD, include any extra geometry that might be necessary to fabricate the part.
3. Export the design into DWG or DXF (Autocad – configured to metric)
4. Import into CAM (VisualMill – configured to metric) . If there are errors opening the file make sure the properly supported autocad version is used (may not be the latest available).
5. Select geometry for a given machining operation (MOP) and assign the relevant cutting feeds and speeds, cutters, depths and number of rough vs. finished passes.
6. Assign raw material geometry and run the simulator for a given MOP to troubleshoot any inefficiencies or errors in design. Pay close attention to corner cleanup and cutting overlap.
7. Set the export type to MACH3_MM and process a given MOP or group of MOP (if same cutter) and save the resulting <job_name>.NC file to a flash drive.
8. Load the job on the Minitech, zero all three axis to your material and run the job.

Process Details:

While steps 1-7 require further training, this document will focus mostly on the actual machining process.

1. **LOADING THE DESIGN:** Having processed the MOP and generated (MACH3-MM only!) a NC file (containing G-Code which instructs the CNC on how to move):
 - a. Ensure all house air taps are turned OFF and all power connections are plugged in. (This may require lockout keys.)
 - b. Turn on the Minitech PC
 - c. Turn on the Minitech controller
 - d. While these are powering up, inspect the equipment and log book to determine everything is safe to operate. Begin a new entry in the log book for this job.
 - e. Launch Minitech Mach3 from the desktop
 - f. Insert flashdrive and copy your files to the PC so that you can eject your flash drive prior to machining. (You may require it if you determine changes to later steps are necessary.)
 - g. Load your job file (<job name>.NC) and click the **RESET** button in the bottom right corner of Mach3.

2. VERIFY DESIGN: Having loaded your design, click the Jog Menu button.
 - a. Ensure there is no cutter loaded in the chuck (tool holder).
 - b. Put in place the perforated rubber mat (if using the vacuum table), and then your raw material roughly in the center of the table. Consider grouping open holes into easy to cover regions around your material.
 - c. Select 100% jog speed and traverse the chuck (In X/Y Only!!) until the chuck is approximately over where the zero should be in your material. This should be the bottom (nearest you) left corner of the material.
 - d. Click **Zero All** and then **Regenerate**. Observe your drawing on screen has shifted.
 - e. Using X/Y job, traverse the chuck to the far (away from you) right corner of your design. If you have not properly produced a Mach3MM design you may notice the image and actual chuck position do not correlate with your expected dimensions. Your rough stock material should not be much larger than your expected part.
 - f. If everything checks out, move on to the next step.

3. FITTING THE CUTTER:

WARNING: Proper eye protection is required beyond this step.
NO LONG SLEEVED SHIRTS, DANGLING JEWELRY OR UNBOUND HAIR

- a. Obtain your desired cutter and insert it carefully into the loosened chuck. This may require a certain dexterity to achieve. The cutter should be inserted as far as is practical, the less it extends from the chuck, the stiffer the cutter and more precise its cutting.
- b. Turn on any accessory lights to help with the next few steps.
WARNING: Assume every step from here can cause grievous bodily harm and undertake due caution when working near the business end of the CNC. No horseplay or distractions are allowed near the equipment especially while your hands are on the cutter.
WARNING: The air supply to the equipment should be OFF to prevent any possibility of injury while working near the chuck.
- c. Using the two provided wrenches hold the chuck shaft stationary with one while also applying sideways pressure to the cutter so that it does not fall out (it often does, ruining the cutter). Using the second wrench, tighten the chuck (**DO NOT OVER TIGHTEN**).
Note: One finger, pressed against the far end of the wrench will turn white but not require bodily effort to tighten further, consult with an expert before being confident you are tightening properly.
- d. Ensure the orange valve after the air regulator is in the OFF position (crosswise). Turn on the house air on the left of the PC, associated with the cutting spindle drive. Ease open the orange valve, a working pressure of approximately 0.28MPa is expected.

4. ZEROING THE CUTTER

WARNING: Be wary of moving the cutter in the Z direction (UP/DOWN). Double check that your JOG rate is appropriate and it is advised to move AWAY (UP) from the part to verify this jog speed before moving towards the material. Any time you type in a new jog rate you MUST PRESS ENTER to change.

WARNING: Ear protection required beyond this step.

- a. Switch on the vacuum pump and ensure that any open holes are covered with blank rubber sheet or taped off with electrical tape. Your raw material should be held strongly in place, however, this is related to the material's coefficient of friction and vacuum may not be sufficient to hold your piece in place. Consult with trained staff for alternatives.
- b. Translate the cutter in X/Y until over your desired zero (should be over raw material not at the edge)
- c. Select 50% jog rate and begin easing cutter down in the Z dimension. (UP/DOWN) the objective is to get within approximately an inch (2-3cm) of the material.
- d. At progressively closer distances, change the jog feed-rate to smaller and smaller values. YOU MUST PRESS ENTER AFTER TYPING IN A NEW VALUE.

WARNING: Approaching the material too fast may result in impacting the cutter with the material may result in damage to the cutter, material, equipment and potentially you. ***So be cautious!***

- e. When within a few centimeters of the material, ensure no one is near the CNC except the operator at the PC
On the spindle controller or in Mach3, select manual spindle control and turn it on. The cutter should now spin up to speed, if it does not do so, check your air supply regulator is showing the proper pressure and that all valves are open.
- f. Set jog rate to a couple percent and begin lowering the cutter towards the material. As you approach the material you should be setting 1% jog rate and watching for the cutter beginning to touch the material, which will produce debris. Another way of checking that you have touched the material is to lower in the Z axis, then move the cutter to the left and observe if there are any marks on the material.
- g. Setting zero is time consuming and tedious and may result in broken cutters and a high level of frustration. Be patient and methodical to the relevant degree of tolerance.
- h. When you are satisfied that your cutter is just AT the material surface, select ZERO ALL. Translate the cutter to a safe distance above the part, approximately 10 inches or 50cm, enough to prepare yourself when you actually start the automated portion of this job.

5. RUNNING THE JOB (AKA MAKING CHIPS):

- a. Make sure spindle is in AUTO mode on the spindle controller and in Mach3.
- b. Turn on building air to the coolant nozzles and aim them at the end of the cutter, this will provide much needed cooling as well as chip removal, both necessary to prevent fouling the cutter and ruining the job.

WARNING: Keep hands, fingers and eyes at safe distances while CNC is in operation. Material and potentially cutter debris may fly off at damaging speeds at any point.

- c. Switch to CONTROL menu in Mach3
- d. Hit regenerate and do a quick once over check of all relevant settings (feed rate override should be off – aka 100% normal)
- e. **IMPORTANT:** Keep a finger over the ESC key which functions as the ABORT or ALL STOP key. Only once expected cutting operation is achieved can you remove your hand. When in doubt hit abort and inspect your assumptions.

f. **GO OVER MENTAL CHECKLIST:**

Vacuum pump on

Building air on to cutter spindle and coolant nozzles

Part is properly loaded and zeroed

Everyone is properly equipped with safety equipment and at a safe distance.

g. **Click cycle start**

ABORT if cutter does not spin up or start cutting at an expected depth or if it begins off the material.

Clean chips should be produced, no stringy lines, not wrapping up in the cutter.

If you are new to machining, stay during the entire machining process. Otherwise it is expected that you check on the equipment during operation at 10-15 minute intervals.

6. **TURN OFF PUMPS, AIR, REMOVE CUTTER, CLEAN UP**

Complete your entry in the logbook, noting any odd or damaged behavior or equipment.

Notify faculty/staff if the case of any injury.

BE SAFE

--Chadd Armstrong