Using the AXYZ 4008 CNC Mill

Generating toolpaths and running the CNC Mill is surprisingly easy.
General knowledge of the PC and CAD programs is required.

STEPS FOR SETTING UP MILL

1. Turn on machine. ON/OFF button is located at the base of machine. Press the black button to turn the machine off and the white to turn it on.

* There is a machine reset/shutoff handle that should not be touched unless the machine requires to be disconnected.

2. The machine will take a minute to initialize but will ask you a series of questions.

*Whenever you first turn on the Mill, the chuck and tool tip assembly needs to warm up. PRESS THE GREEN BUTTON TO START THIS PROCESS

3. After the machine has completed this task, it will search for its DEFAULT G55 ORIGIN. In other words it positions itself to the corner of the machine bed. This is the corner that you want to attach your material to. Knowing what origin the machine is set to is important when you are laying out your tool paths on the computer. It also allows you to reference this point while interchanging materials on the machine bed.

4. If you would like to set your own origin, use the control pad to position the tool head and press F10 to set that as your G54 ORIGIN.

5. Instructions for other steps such as changing out the waste board and setting the tool tip can be found on the wall next to the CNC. If the tool tip needs to be changed then loosen the chuck until the tool tip slides out. There are a variety of tools in the cabinet next to the CNC computer that can used for different materials and types of cuts.

6. If you change out the tool tip, it is necessary to calibrate the new tip by pressing F25. This will move the tip to a sensor located at the edge of the machine bed. If you need to replace the waste board you will have to tell the machine where the top of that surface to maintain accurate cuts. This can be done by pressing F28 and moving the tool tip exactly to the top of the waste board. Detailed instructions are located on the wall. FOR 2D CUTS YOU NEED A FINISHING TOOL TIP.

7. Once you have your material fastened on the waste board and machine bed, the origin set, and all of the other machine parts calibrated you are ready to continue.

STEPS FOR SETTING UP ART CAM - 2D CUTTING

1. Art CAM is a fairly intuitive program to use although it will always give you problems when you least expect it.

*It is important that all students have their files ready before using the CNC. In other words, if a file has overlapping lines, un-resolved and open curves, Art CAM will not generate the tool paths needed to communicate with the CNC. YOU ARE NOT RESPONSIBLE TO FIGURE THIS OUT.

2. OPEN Art CAM --> CREATE NEW MODEL. You will be prompted with a screen SETUP JOB DIMENSIONS asking the size and thickness of material being used and also the origin relative to the machine. The graphic on the screen with update as you make these changes. Material Z Zero should be set at MACHINE BED and Model position in material - TOP OF THE BLOCK The origin of your material needs to correspond to the origin set on the machine. The thickness of your material tells the machine where to stop and start cutting in the Z axis. THIS IS A VERY IMPORTANT STEP!!

3. A window will appear in 2D view. At this point you want to import your cutting lines that were previously generated in CAD, Rhino, etc. Go to VECTOR --> IMPORT. Select your .dwg or .dxf to bring into Art CAM. It will come up with a dialogue and press OKAY.

4. Your lines will appear in the window. At this stage you want to position them on the area of the material you want to cut. THIS STEP IS CRITICAL AS IT WILL DETERMINE WHERE ON THE MATERIAL YOU WILL BE CUTTING.

5. SELECT THE LINES YOU WANT TO GENERATE TOOL PATHS FOR. This can happen in steps. For example, if you want to cut a shape out entirely and within that shape you want to do a relief cut, then those two cuts can be made it two or more individual "steps" in other words you can prioritize and order separate cuts. In this case you would cut the relief first and the entire shape out second. JUST BE SMART ABOUT HOW YOU PRIORITIZE CERTAIN CUTS. This will be addressed in the next few steps.

6. Click on the TOOLPATH tab at the bottom of the screen. CLICK ON 2D TOOLPATH ICON THAT IS CALLED 2D PROFILING.

2D PROFILING

2D profiling is the component of the program used to "cut" through your material. In other words, if you want to use the tool tip to cut out shapes you must select this function in order to generate the appropriate tool paths for the CNC. In this window you can also adjust the STEPDOWN, PLUNGE RATE, and STEPOVER which will determine how quickly and accurately your material will cut.

1. In the profiling window that pops up there are various options that need to be addressed. In the PROFILE SIDE you want to choose either INSIDE CUT or OUTSIDE CUT according to how you want the machine to cut on the lines you have generated for yourself.

2. You want to then select your profiling tool according to the tool tip that is currently installed on the machine. For example, in most cases the 1/4" tool tip is used to make 2D cuts. Therefore you need to select END MILL 1/4". If there is an 1/8" bit on the machine then you must select the 1/8" tool in Art CAM.

*As a general rule, these settings do not need to be changed as they are pre-configured with each tool profile.

STEPOVER

The STEPOVER value can determine the resolution of the cut and will affect the rate in which your material will be cut in the X axis. So as the tool moves back and forth across your material it does this in increments. The smaller these increments the less visible the machine lines and cut paths.

STEPDOWN

Once you have selected the profiling tool, you can adjust the STEPDOWN which is a very important value. The STEPDOWN value determines how far "in the Z axis" the machine will cut during each pass. If your STEPDOWN is set at .5", then the first pass will cut into your material .5". For more accurate cuts, in more dense materials, the lower the STEPDOWN will ensure that there is less resistance on the tool and therefore a cleaner, more accurate cut.

FEED RATE

Like both the STEPOVER and STEPDOWN, the FEED RATE will affect both the speed and resolution of your cut. In layman's terms, the FEED RATE will either slow or speed up the rate at which the machine head moves in the Y axis.

PLUNGE RATE

The PLUNGE RATE will determine the rate in which the tool will "plunge" in the Z axis. Like the previous settings, this value can potentially used to reduce resistance on the tool tip and therefore

SPINDLE

The SPINDLE VALUE is the speed with which the bit is rotating. For a 1/4" bit the default value is set at 16000 rpm. This value is good for cutting through wood, although at times it may be pertinent to increase or decrease this value according to the type of wood, or other material you are cutting.

*After the appropriate settings have been made for the type of cut you want to perform, you must select the LINES you want to cut that have been imported into ArtCAM. When selected, these LINES will turn pink. The LINES must be selected in order to generate toolpaths. After the toolpath has been created, it will appear maroon in color.

2D AREA CLEARANCE

2D AREA CLEARANCE is a feature of the tool that allows you to cut releifs into you material. In other worlds you can cut a shape into your material without cutting all the way through it.

1. You must set the START DEPTH which will be the height of your material. For example if your material is .5" then your start depth would be .5". The FINISH DEPTH is the amount you want to cut in the Z axis. So, if you want to cut into your material only a 1/16" for a specific shape then you set this value at .0625 and it will cut out .0625" from the top of your material. You can adjust this setting for each toolpath you create.

2. Like 2D PROFILING, it is necessary to ADD a TOOL such as the 1/4", 1/8", etc, in order to generate the toolpath. Choose the tool and name your file - this will generate your toolpath.

3. Once all your toolpaths are created go to TOOLPATHS --> SAVE TOOLPATH AS and this will bring you to the save toolpaths window.

4. SAVE TOOLPATHS WINDOW is also a very important step in that it allows you to ORDER your cuts. In other words you can tell the machine to make certain cuts in a certain order. This is useful especially when you are doing both relief cuts and profiling cuts at the same time.

*When you have a file that requires a number of different cuts at different depths, you need to set these values individually for each cut and tool path you generate.

5. Click on the red arrow to move calculated TOOLPATHS into a single file to send to the machine. Once the toolpaths are in the right order, click SAVE. This will allow you to save your toolpaths and send them to the machine. You do this by saving your toolpath in the JOB FOLDER on the desktop.

6. Once the tool path has been placed in the JOB FOLDER you can go back to the machine, find the file by pressing the FOLDER ICON on the control panel at the machine.

7. When you find your file, your material is fastened securley to the table, the origion and tool tip calibrated, you are ready to cut. Press the GREEN button on the control pad to initiate cutting sequence.TURN ON THE VACCUM BEFORE YOU COMMENCE WITH CUTTING.

*At any time you can pause while the machine is cutting by pressing the RED button. This will pause the machine immediately. You can make adjustments and resume cutting by pressing the GREEN button. In the event that the machine is making cuts that do not correspond to your file, pause immediately and press the RED button twice to cancel cut all together. You can also cycle through the various shapes that it generates for itself that are articulated by the toolpaths.

STEPS FOR SETTING UP ART CAM - 3D CUTTING

1. Turn machine on, following the same steps for the 2D cutting. SWITCH OUT TO BALL NOSE TOOL. After performing this task, press F25 and set the tool tip. FINISHING TOOL TIPS WILL NOT PERFORM THIS TASK, MUST USE BALL NOSE TOOL

1. Measure material thickness, length, and width.

2. Firmly block all sides of the material

*Be aware of how you fasten the material down. If cutting file has steep drop offs, it is possible that the tool tip, when cutting, will interfere how the material is fastened to the machine bed, or even worse break the tool tip.

3. Perform start up process which includes warming up machine head.

4. At the computer, inspect file in Auto CAD or Rhino. The file must be at the appropriate scale and have surfaces that are closed and manageable. Once the file has been prepared, scaled down, etc, OPEN Art CAM.

5. Export your file from CAD or Rhino as a .3ds, then as a binary file. Save to desktop.

6. In Art CAM create a new file and set up the dimensions of the material you are cutting.

7. Besides the material dimensions, the only change that needs to be made
model position in material--> machine bed. Keep machine zero at machine bed.

8. Once those parameters are established you need to import your 3D model.
Go to RELIEF--> IMPORT 3D model.

9. If your model has been imported properly then it will appear in the 3D model screen
which is one of the two windows in Art CAM.

10. This window will tell you were you 3D model is relative to your material which in Art
CAM is represented by a red box. PRESS CENTER. This will center your 3D file in
the red box.

11. At this window you control where you want your 3D surface relative to the material
you're cutting. You can toggle the z, x, and y positions of your surface and control
where the cuts will be made on the material. POSITION --> PASTE --> CLOSE

12. The next step is to have Art CAM generate a tool path for the surface you're cutting.
Under toolpaths, go to TOOL PATH 3D --> MACHINE RELIEF

13. In this new window, AREA TO MACHINE --> SELECT WHOLE RELIEF

13. The toolpath dialogue needs to be adjusted according to the tool tip being used. If
you are using a 1/2" ball nose, CHANGE THE TOOL TO THE CORRESPONDING
BIT. TOOL --> SELECT 1/2"

14. After selecting the appropriate tool tip there is a grey arrow that will provide a series
of other options such as the STEPOVER, STEPDOWN, FEED RATE, PLUNGE
RATE, ETC. These values are discussed in detail previously in this document.

15. These are all important variables however, pay special attention to the STEPOVER
AND STEPDOWN.

    STEPOVER- This will determine how close the toolpaths will be rasterized in the
X axis.

    STEPDOWN- This value will control the amount in which the tool cuts after each
pass. For example, if you have a material that is 4 inches thick, and you are making 3
passes to cut your surface, you would have the step down set to no more than 1.75",
unless you want a rough cut.

*Both Stepover and Stepdown usually depend on the material being used and the
quality of cut that you want to make.

16. Select --> MULTIPLE Z PASSES. By selecting the Z height of the first pass, you can tell the machine how deep you want the tooltip to plunge into the material in the first pass. Usually you want this value to be less than 1.0". Meaning that the pass will cut into your material not more than 1.0".

17. The value of the Z height of the last pass is the height spec and numerical value of distance between the the top of the machine bed and your material. In other words, if you want the last pass to cut out and leave a certain amount of material left, then this value determines that amount of material in the Z.