CNC USER GUIDE

Allowed Materials:
- Acrylic
- Styrene
- MDF
- Plywood
- Hardwood
- Softwood
- Particle Board

Banned Materials:
- Plaster
- Foam core
- Cardboard
- Foam
- Styrofoam
STARTING UP THE CNC

Turn the machine on by pressing the **ON button (WHITE BUTTON)** on the CNC

The machine will take a minute to run through its diagnostics. (This can be seen at the console)

**Dialog Console**

After the initial diagnostics, allow the machine to seek the machine origin (**seek G54 Origin**) Press Enter

Next, The machine will ask if you would like to warm up the spindles,

Make sure that you have installed the router bit desired for your cut and that it is properly tightened.

Then select YES

On the dialog console

(+/- BUTTON) to change YES or NO

(ENTER BUTTON)

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

Allow the machine to warm up by pressing (ENTER)

Before proceeding, make sure that all materials and students are away from the router.

This will take approximately 5 minutes to warm up. Once the machine has warmed the spindles, you are ready to change the bit and set your origins etc.

CHANGING THE BIT + SETTING USER ORIGIN

*Ball nosed bits are for 3D reliefs + End Mill bits and Viper bits are for 2D Profile Cuts*

**(F12 Enter) Seek Machine Origin**

Change the router bit using the CNC’s wrenches

Once you have the new bit in place and tightened, press (F25 Enter). This will set the tool height.

The machine will now ask you if you would like to measure **Tool 1** or **Tool 2**

For a 2D cut you will select **Tool 1**

The router will then “jog” to the sensor. Now click **ENTER**

Next, move the router bit to the origin of your material using the arrows located on the console buttons 2, 4, 6, 8. The origin will be located in the corner nearest the tool tip sensor. Once in place, press (F10 Enter) to set user XY Origin. This will set your user origin. **IGNORE** THE Z HEIGHT. This step is only to set the X and Y
SENDING FILES

CPU Login:

   User: fablaser
   Password: Password!

Setting up a CNC router file (2D)

First open up ArtCAM

Click on Create New Model

The Job Dimensions Prompt should appear.
Enter your **MATERIAL** dimensions (as it is positioned on the bed)

**X:** correlates to the length of the table

**Y:** correlates to the width of the table

**Z:** correlates to the vertical thickness of your material (not to exceed 5”)

**Units:** should be inches

**Material Z Zero:** is always Machine Bed

**Model Position in Material:** Top Of Block (for 2D Files)***

Once set up click **OK**

*** This setting varies from the 2D setup and the 3D setup. ***
Here we see the two screens that pop up after setting up your material dimensions.

Select the corresponding view to work in and maximize the view.

2D for 2D cuts

3D for 3D cuts
To import a file, Click on the Vector tab at the top of the page

In the dropdown menu, select IMPORT

Through the navigation window, find and select your DWG***

***Import file must be in the format of ACAD.DWG and needs to be saved down to a 2004 Polylines file***

***File must only include lines that are intended for the cut. Any extraneous lines and information will cause problems at setup.***

***It is important to create a boundary around the file that correlates with the material dimensions, this will ensure that the drawing is placed correctly***

Click OK and all vectors should be placed and centered on the material.

All File setup notes above are the student’s responsibility, if their file is set up incorrectly, have them go fix the file and bring it back.
Once you have imported your vectors. Select all vectors that will share the same cut type.

Ie. All OUTER cuts should be selected together, all INNER cuts should be selected together, and all AREA CLEARANCE cuts that share the same depth should be selected together.

Once lines are selected, click on the appropriate tool path at the left of the page.
Above is a guide to the different types of tool paths that are available.

**2D**
- Inner Profile
- Outer Profile
- Area Clearance
- Machine Along Vector

**3D**
- Machine Relief

In order to view these options, you must select the tool paths tab at the bottom.
In the tool path options make sure to set the following settings:

**Profile Side:** This indicates whether you would like to cut on the inside of the line or outside of the line.

**Finish Depth:** This is the depth (from the top surface of the material) into the material that your tool path will cut.

**Machine Safe Z:** This is the Z height that the tool will travel as it moves from one shape to another. (This number should be about 3/4” above your material.

**Profiling Tool:** This is the type of tool you will be using to cut your material.

End Mill: 2D

Ball Nose: 3D

**Name:** Name your tool path so that you will remember what vectors and depths are involved

**Calculate:** Now, This will give you a preview of the tool path created. Above: **OUTER**
Upon clicking on Profiling Tool in the previous step, this window will appear. Select the desired tool from the list.

All default settings under Wood or Plastic, Roughing and 2D Finishing, 3D Finishing should be correctly set for plywood, MDF, particle board etc.

Any cutting of Plastic or other materials, please contact Stephen Mora (505)710-1165 for further information.
Once all settings have been set, click on **Calculate** now and you can view the tool path preview.

Above we see that our OUTER tool path remains in place and we can now see our INNER tool path.
Here is a demonstration of an AREA CLEARANCE. In this view make sure to set the AREA CLEARANCE tool strategy to **OFFSET**

Notice the Profiling Tool settings that are present; these should be preset and default to the material. Stepdown should never be set to a higher value than 0.5

*** If a material other than wood, plywood, mdf etc. is being cut, please contact Stephen Mora for setting adjustments.***
Once all tool paths have been created, select **TOOLPATHS** from the tool bar, and click on **SAVE TOOL PATH AS**
We have previously named our tool paths, here we need to select them and with the black arrow, move them over to the right in the order we want to cut them.

***Remember, we want to cut the least invasive cuts first and continue until we are finished.

Ie.

Area Clearance first,
Inner second,
Outer third etc.

You have now created a G-Code that the CNC Router will understand, Make sure that the CNC Router is turned on and save this file to the Shortcut to Jobs folder on the desktop. Name this file appropriately and we can now move to the CNC machine for the remainder of our setup.