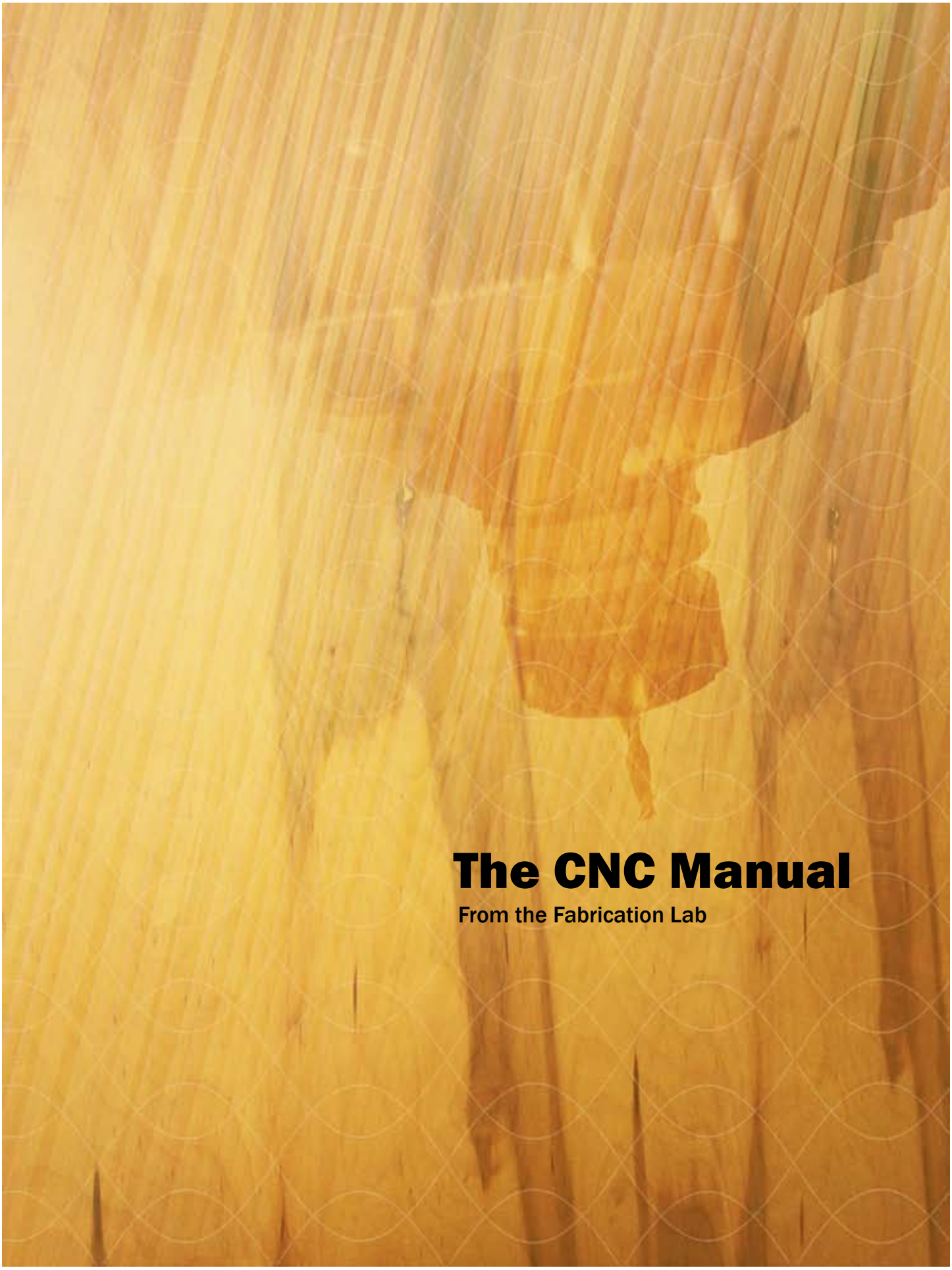


The CNC Manual

From the Fabrication Lab

Table of Contents

General Information

Section 1

General CNC rules

The following are a series of safety rules and recommendations for the appropriate operation of the Milling Machine.

In order to maintain a safe work environment please follow them strictly.

Safety Rules

1. At least one qualified person needs to be present at all times when the CNC machine is running. (This is by far the most important rule.)
2. Eye and hearing protection need to be used when operating the CNC router.
3. Whenever the shop is in use the shop door towards the milling must be freely accessible in case of an emergency.
4. Long hair should be tied back prior to using the tools. Sandals and open shoes are not allowed. Neckties and bracelets should be removed.
5. Students are expected to provide their own material. Students must provide their own bits for 3D jobs in any material other than foam.
6. Students are only allowed to use previously approved material and router bits. If in doubt, ask Steven or the TAs.
7. Privileges will be suspended for abusing or damaging the machine. Keep work area clean. Remove any tools (ie. adjusting keys, wrenches) before operating.
8. Privileges will be suspended for not cleaning up all excess material after your job has finished running or contributing to an unsafe environment. The TAs will check for compliance with the above rules.

Access

Fabrication Laboratory

The fabrication Laboratory is restricted to the following students

1. Students employed under the work-study program for Shane Williamson.
2. CNC TA's employed under the work-study program for Steven Beites.

Milling Room

Access to the milling room is limited to the previously mention students and granted to students who have booked milling time.

Material Storage and Cleanliness

The milling room does not provide space for the storage of student materials. As stated in the general rules, it is the student's job to clean up the CNC room after their job has been completed. This includes removal of excess material, sweeping or vacuuming up dust, and at time helping empty the sawdust bags. You will get dusty during your milling job – please dress accordingly.

Job of TA's

The Job of the CNC TA is to verify that the student has properly constructed a milling file in RhinoCam. They will verify the proper bit has been selected and that the feeds and speeds associated with the job are correct. The primary job of the TA is to set up the job on the CNC table and to monitor the job as it is running. The student must be present in the milling room for the entirety of the job.

Booking Milling Time and Cost

Students must verify their toolpaths and run a collision check in RhinoCAM before creating their .NC file. This file must be cleared by a TA before milling time can be booked. Before the student can book milling time, the CNC file must be completed and verified by the milling TA. The TA will officially schedule milling time with Steven. Payment for the job is due immediately after the job has concluded. (**20$ per hour)**

Step by Step Guide to Submitting a File

1-Review the CNC Manual

2-Become familiar with RhinoCam

A comprehensive video tutorial has been created specifically to assist you in using RhinoCam. For additional documentation, review the manual "RhinoCAM2GettingStartedGuide" which can be found in the same folder location as the video tutorials.

RhinoCam video tutorials and documentations: \Root(\\ald.utoronto.ca)(R):\Groups\CNC Milling

3- Create your NC file using RhinoCAM

RhinoCAM is installed on the computers in the HOK. Students must verify their toolpaths and run a collision before submitting their file.

4-Submit files.

Students must submit their original rhino file and NC file.

a- Drop both files in a subfolder within the main Submissions Folder (\Root(\\ald.utoronto.ca) (R):\Groups\CNC Milling\Submissions). When creating the subfolder, rename it using your first initial and last name.

b- Download and fill out the submission form.
Found here:

c- Email the submission form to: CNCmilling@daniels.utoronto.ca

Once the file is reviewed, verified and approved by the TA, he or she will contact you with your scheduled milling time. Payment for the job is due immediately after the job has concluded. Cost - $20/hour

Software

Section 2

General Information

The software associated with the Daniels CNC machine is called RhinoCam. This is a plug-in for Rhino. Before opening the RhinoCam plug-in you must complete the following:

- Units- make sure the units are set in inches
- Scale- 1:1
- Orient Geometry- the origin (x=0,y=0,z=0) corresponds to the origin on the CNC table

RhinoCam

RhinoCAM is a unique CAM product plug-in that runs inside of Rhinoceros 4.0. Plug-ins can be considered as independent applications that can be loaded and unloaded on demand from the host program, which in this case is Rhinoceros 4.0 (Rhino). This fully integrated RhinoCAM plug-in seamlessly integrates Rhino's CAD functionality with toolpath generation and cutting simulation/verification, in one package that is easy to use.

You can work with the native Rhino design data as well as use any of the data types that can be imported into Rhino such as solids, surfaces and meshes. Then you can use RhinoCAM with its wide selection of tools and toolpath strategies to create machining operations and associated toolpaths. These toolpaths can then be simulated and verified, and finally post-processed to the controller of your choice.

Rhinocam Documentation and Tutorials

1. Please review the RhinoCam PDF available in the CNC Milling folder. You will not understand rest of the software section without reviewing the document.
2. RhinoCam online help can be found in Rhinocam drop down menu.
3. RhinoCam online tutorials are available at: **http://www.mecsoft.com/MillSelfTraining.shtml** (make sure to select the Rhinocam2 tab).
4. RhinoCam video tutorials are available on the network's "CNC Milling" folder.

Knowledge Base

The Fabrication Lab has developed a series of preset milling operations that students can load and use freely. These preset operations are designed for beginner students who have limited knowledge and experience with milling. This set of operations is call the Knowledge Base and is ivided into 2 sections.

- 2d Cutting operations

▪► 3d Cutting operations

The 2d cutting operations Knowledge Base in located on the networks CNC Milling folder under Knowledge Base. The folder contains Knowledge Base files associated with 3 different materials: 1 MDF, 2 Plywood, 3 Acrylic/Polystyrene. Under these headings there is also a variety of material thicknesses to choose from.

The 3d cutting operations Knowledge Base in located the same folder as the 2d cutting operations. The 3d cutting operations Knowledge Base offers two surfacing templates using a variety of tools.

Daniels Tool Library

The Fabrication Lab has developed a consistent tool library that outlines the tools the school has made available. The school provides a basic set of tools required for 2-d cutting operations in MDF, Plywood, and acrylic. The school also provides surfacing tools for 3-d operations in FOAM only. The Tool library can be loaded into RhinoCam. It is located in the Student milling folder under tools. The student must purchase tools required for operations that lie outside these parameters. A tool catalogue is located in the tools folder.

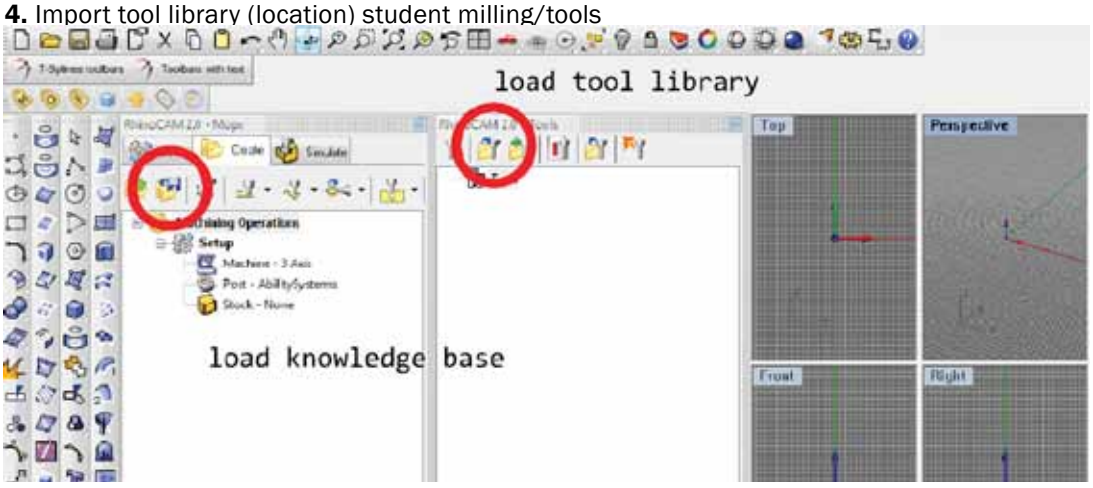
To open the tool library ▪►

1. Open Rhino

2. Open Rhino Cam

3. Open Tool Browser in the drop down menu

4. Import tool library (location) student milling/tools



Post Processor (very important)

You must use axyz-router as the post processor in Rhinocam.

Location student milling/rhinocam/rhinocamposts

Hardware

Section 3

Equipment

There are five pieces of equipment in the Milling Room:

1. Main Power (CNC)

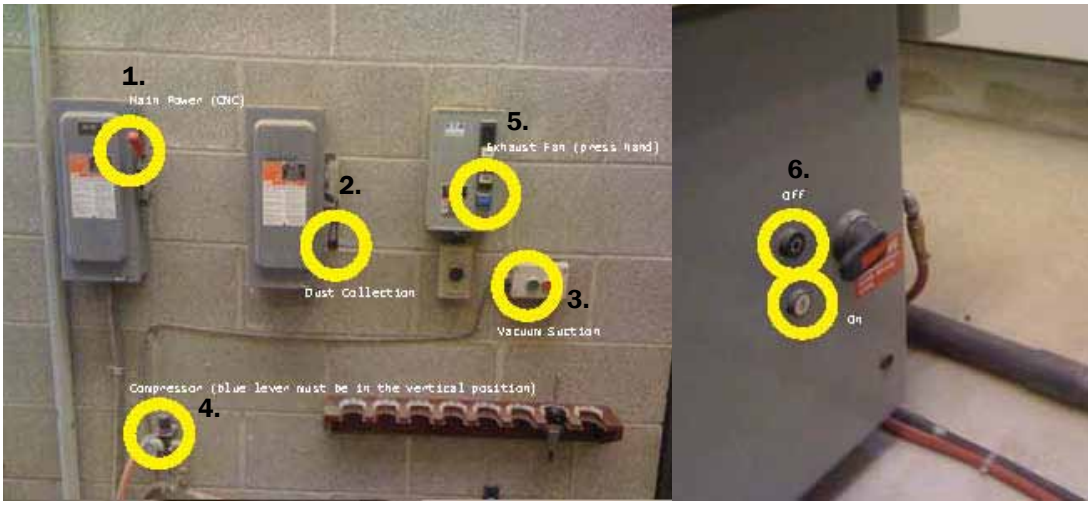
2. Dust collection

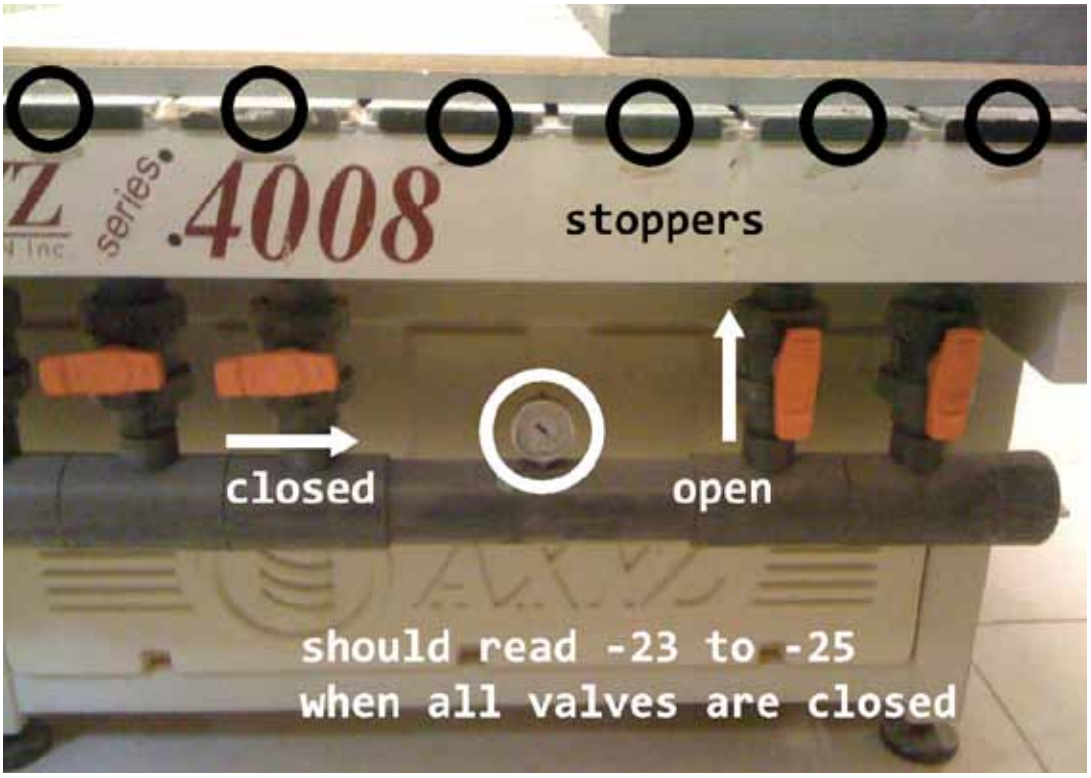
3. Vacuum suction compressor

4. Air compressor

5. Air Filter

6. CNC router - OFF / ON





ALL INSTRUCTIONS BELOW ARE PERFORMED BY THE CNC TA

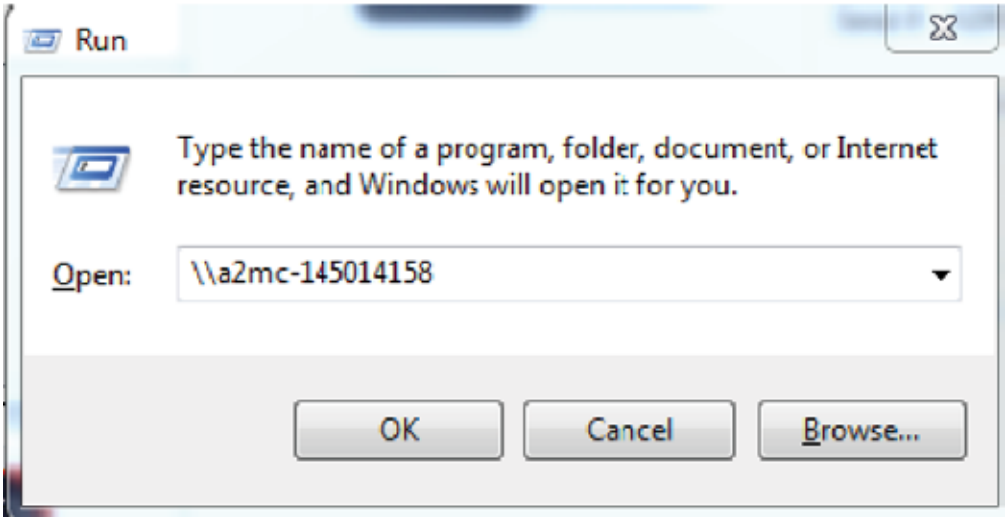
Running a Job GUI (graphic user interface)

Please read the AXYZ manual located in the Student Milling under AXYZ.

Step by Step instruction for running a job

Descriptions of all functions associated with Machine operation are available in PDF form in the GUI help index

Place NC file in Machine directory folder



Run this network location to find the jobs folder on the Machine network

A2mc-145014158/HDD1/Release/Jobs

Qualify tools :: [F25]

1. You must associate the same tool in the tool library to the tool number on the machine ex.. If you used tool_3 and Tool_8 from the Daniels tool library then you must give the same number to the tool at the machine.

2. Press F 25+-3 to qualify tool_3 the machine will move to the S mouse (digital measuring device) to qualify depth.

3. Repeat process for all tool required in for the job

Setting Origin x/y :: [F10] [F11]

The CNC machine has the capability of recording six different origin points and storing them in its memory. They are described in the following way ▪►

Machine Origin G54
User Origin G55
Preset Origin 1 G56 all files built from Knowledge base
Preset Origin 2 G57
Preset Origin 3 G58
Preset Origin 4 G59

The Machine Origin is the absolute 0,0,0 point on the entire system (not the milling table) User Origin can be set on the fly. All preset origins can be set with the Machine head or on the GUI.

1. Press F 11 to select the origin you wish to use (or that you wish to set) use the +- key to scroll through the origin numbers.

2. If you want to set your own origin select G55 in the previous step. Locate the machine head at your desired position and press F10.

3. It is our recommendation that 2d jobs be run at G56 and 3d jobs be run at G57.

The fabrication laboratory specifies that for 2 d cutting operations use G56. This origin is designed to be used in conjunction with 2 d knowledge base templates. This origin has a preset z offset to allow the tool to cut all the way through the material.

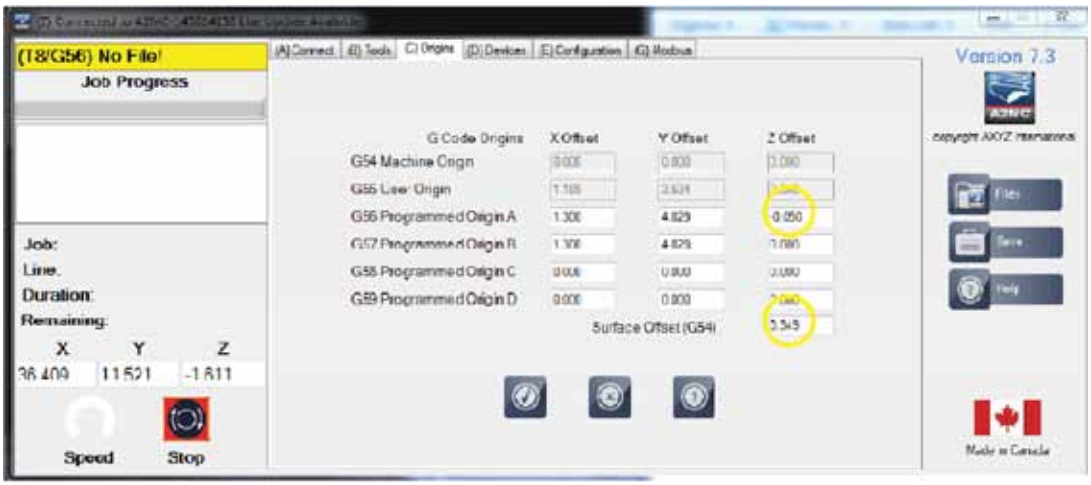
Setting Z depth :: [F27] [F28]

Function 27 allows you to set z depth for the G55 user origin only. This function will not work with other origins. To set z depth for G55 press F27 and move the tool tip to the desired z height. Function 28 has a series of options:

1. Sets spoil board height to input the spoil board height press F28 and select set waste board, touch the tool tip to the waste board. Record the number indicated (this is the z offset – the difference between the s mouse height and the top of the waste board height).

Input this number into the GUI/ origins Once this number is entered the z depth of the table is a permanent value. This value only needs to be changed when the waste board has been planed or changed to a different thickness.

2. F28 also has a built in planning file that can be set on the fly.



Surface offset (this refers to the distance between the s mouse and the top of the waste board

G56 z offset (this is the distance that the tool will cut into the waste board)

Other Important [F] codes

- F9 abort job
- F12 seek machine origin (absolute 0,0,0)
- F13 return to current origin
- F41 set feed rate (change the feed rate of the current job)

CNC Basics

Section 4

General Information

Project Size

Our CNC router has a functional bed size of 4 feet by 8 feet. Height limitations are more complicated and may vary depending upon project dimensions, tool choice, and toolpath design. The combined height of the material and cutting tool cannot exceed 15 inches. If your material is too tall, the cutting tool will not be long enough to cut all the way through the material.

Aesthetics

The final appearance of your project will be affected by both tool selection and toolpath design. The use of different cutting tool end geometries will change the tooling marks on your project. The specified step-over will determine the smoothness of the machined surface.

Tool Selection

CNC router tools are optimized for particular applications on specific materials. The Fab Lab maintains a large inventory of tools for different tasks. Ensure that you have chosen the best tools for your project by reviewing the tool chart. In general, larger tools will cut more efficiently, but smaller tools may be necessary for more detailed geometry. Use large roughing tools to take away large volumes of material, large finishing tools to refine broad regions of material, and small finishing tools to refine detailed areas. Always try to use the largest and shortest tool possible for your specific application.

Tool Selection

Typical end geometry on CNC router tools is either flat or rounded. End geometry will affect the appearance of your machined surface. The cutting edges of the tool, the flutes, come in two primary varieties: up-shear and down-shear. Another commonly used tool is called a compression bit, which is a kind of hybrid, combining up-shear and down-shear flute geometry. More specialized flute geometry can provide more sophisticated cutting action. Review the tool chart or speak with a Lab Assistant to learn more about specialized tools.

Glossary Rhino Cam

2½ Axis Operations

In 2½ axis toolpath, the tool can move in X and Y directions, but Z movements are limited to set levels. Because 2½ toolpaths do not relate to either part or stock geometry, machining regions must be selected; these define the boundaries of tool motions. This type of machining is useful for machining prismatic parts – extrusions of curves along the Z axis. Because of its straight sides, a prismatic part can be machined by locking the tool at the first Z level, performing XY motions, then repeating for subsequent Z levels.

Profiling

This method machines open and closed regions by tracing along one side of their contours. You can define offsets so that the tool makes multiple passes relative to the regions.

Pocketing

This method machines closed regions as if they were pockets - completely enclosed by inner and outer regions. The tool cannot go beyond the outer region, and cannot go within inner regions.

Facing

This method machines closed regions as if they were completely enclosing material to be removed. This means that the tool can approach the material from outside the outer regions, creating reverse pockets.

Engraving

Typically used for engraving text or logos on a finished mode, this method machines open or closed, 2D or 3D regions by tracing along the contours.

3 Axis Operations

In this type of machining, the tool can move simultaneously in all three axes. This is appropriate for parts that have complex, curved, and non-vertical surfaces.

Horizontal Roughing

This is RhinoCAM's principal method of roughing, also known as waterline or constant Z cutting, in which the material is roughed out in horizontal layers. This type of machining is very efficient for removing large volumes of material, and is typically performed with a large tool. Roughing is typically followed by semi-finishing or finishing toolpaths.

Horizontal Re-roughing

This is used to create toolpathsv in areas that were not machined by previous operations. Unmachined areas are determined by comparing the part to the stock remaining after the previous operation.

Parallel Finishing

This is an efficient method of finishing or pre-finishing, typically used when part surfaces are relatively flat. A 2D linear zigzag pattern is generated on the XY plane above the part geometry. The tool moves along this cut pattern, following the contours of the part geometry.

Pocket Finishing

This method is used for pre-finishing and finishing of pockets with sculpted bottoms and/or sides. The pockets are defined by regions, and successive inner offsets of these outer regions are generated.

Horizontal Finishing

This method is used for pre-finishing or finishing in constant Z levels, typically used when the part has large vertical surfaces and when Parallel Finishing will not yield satisfactory results.

Glossary AXYZ

Machine Top

This is the physical top of the Z or vertical stroke. The controller automatically seeks this position using a sensor. This is the only fixed vertical position so the controller will often seek this sensor in order to calculate all the other Z positions.

Machine Bottom

This is simply the physical bottom of the Z or vertical stroke. The distance between machine bottom and machine top is the stroke or Z size of the machine. This is defined in the ToolPath for Windows setup menu. Both the Machine Top and Machine Bottom positions are defined by the mechanics of the machine, they cannot be altered by the controller.

Datum Zero

This is the Z origin or zero position. The vertical or Z co-ordinates are measured from this point. Jobs that require a material reference point, such as two and three dimensional engraving will require you to set the material surface by touching the end mill to the top of the material.

Lift Bottom

This value can be defined by selecting Function 4. As you can see from the diagram it is possible for the Machine Bottom value to cause the tool to cut into the machine bed. Lift bottom is set as the lower limit to the vertical travel. Note: When running a 3D job, particularly G-Code, the lowest point will override the Lift Bottom position. Make sure your material is thick enough for the deepest part of your cut.

Lift Top

Often the Machine Top position is far above the material surface. When the machine is running a job and travelling between cuts it will move up to Lift Top before travelling to the next plunge point. Setting Lift Top to be just clear of the material normally saves a lot of process time.

Router Materials

Section 5

Router Materials

Foam

Foam is the best option for milling a large surface out of a solid block. Jobs created out of foam are much quicker to mill than wood and put less wear on the bits. They can also be coated for a more durable surface. We ask that you use polyurethane foam, polystyrene foam (pink and blue), or some other foam with approval by the lab. Foam available from Larry in the shop includes, High to Medium density polystyrene (white) various thicknesses and Grey isolative foam. High density prototyping foam must be sourced from outside.

Plastic

Various types of plastic can be cut with the CNC, but it is in general more difficult than wood and may require special tooling to get good results.

Wood Sheets

In general, plywoods and wood composites (MDF,OSB, fiberboard, etc.) are all acceptable materials for cutting out profiles. If you are planning on using a nonstandard material or finish, please check with a lab coordinator before placing your order. Special bits or tools might be required for example, to cut drywall).

Wood Solids

If something more durable than foam is necessary, you can create a solid block of material from sheet material. However, the faculty only provides bits for solid surfacing in foam.

Solid Block Preparation

Before gluing your sheets together, remove as much material as possible with standard woodworking tools or through 2D profile cuts with the router. Large blocks of foam are also readily available. When gluing up your block, apply thick even layers of glue. For best results use the portable vacuum bag available from Larry in the shop. Allow the block to set at least overnight. If this is not followed through, the layers of the block could delaminate, ruining your job. Please also avoid using toxic glues such as construction adhesive as this material will become airborne.