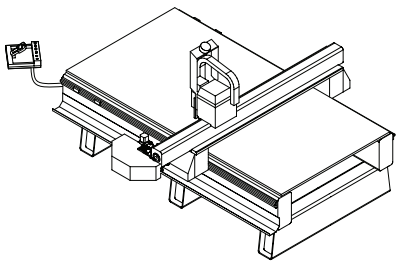
Multicam 5000 CNC Router

Manual

The
Creative
School

Design +
Technology LAB



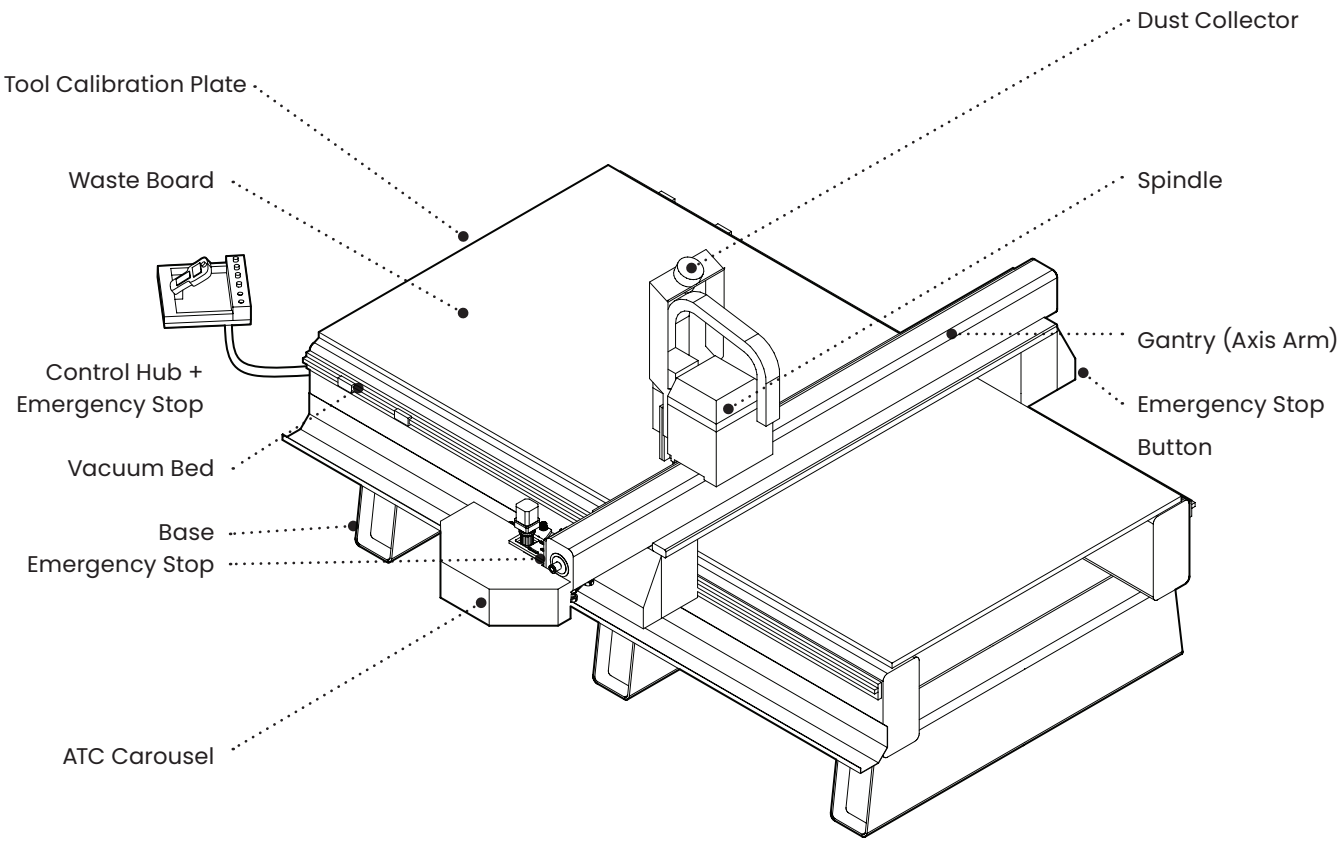
01:

The Multicam 5000 Series CNC Router

This manual presents fundamental information about the Multicam 5000 Series CNC Router.

Please read this manual before booking a CNC appointment with the workshop.

Questions and Concerns: If you have any trouble or questions regarding the CNC router email us at: Designtechnologylab@ryerson.ca.

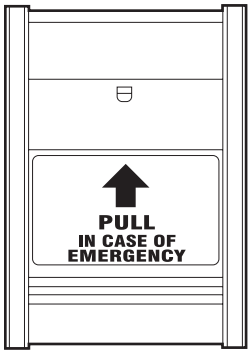


D+TL

General Workshop Safety

Required Personal Protective Equipment (PPE)

Recommended Personal Protective



Emergency Pull Station

Workshop Safety Practices

1. No person will work alone in any areas of the machine room. Design + Technology LAB members working after hours alone must get permission from the lead technician and notify Ryerson security.
2. Safety glasses must be worn while operating any power machinery/tools.
3. Distracting conversations and use of electronics (including cellphones and earbuds) can be dangerous in work areas and must not take place. Please take phone calls in the hall outside the lab.

Clothing and PPE

Before entering the Design + Technology LAB: Please ensure that you have read and adhere to the workshop general safety guidelines and are wearing all required PPE. The following rules concern appropriate Workshop apparel:

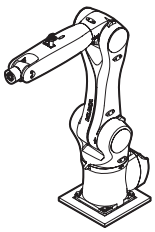
- Long sleeves must be rolled up.
- Loose scarves or clothing must be secured
- Heavy or loose jewelry must be removed (i.e., rings, watches, bracelets, necklaces, etc.)
- Long hair must be tied back.
- Non-slip, hard-toe, flat shoes must be worn.

What to do in the case of an accident.

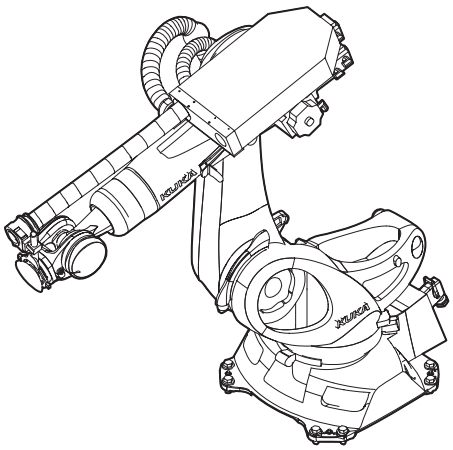
In the case of an accident notify a technician immediately and pull the EPS (Emergency Pull Station) switch to notify security. If possible have the injured person sit down to prevent fainting, collapsing, or further injury.

D+TL

General Workshop Safety



KUKA KR10 Robotic Arm



KUKA KR150 Robotic Arm

Robotic Operation Safety

There are a number of robotic arms located in the Design + Technology LAB. There is 1 large Kuka KR150 in the High Bay (B33A) and 2 Kuka KR10's in the Workroom (B31). Be mindful of robotic operation at the Design + Technology LAB. Do not enter rooms when the following signage is posted.

This sign is most commonly posted on the entrance to the High Bay. Interlocks are installed on doors to the Kuka Cage and High Bay. Do not open doors during Robot use or it will trigger the workshop's Emergency Stop system. If you require access to the Print Lab when a robot in the High Bay is in operation, ask the Design + Technology LAB staff for assistance.

Noise and Air Quality

The Design + Technology LAB is a new facility with a strong interior ventilation system and various dust and fume collectors. Air quality in the facility can be affected by the machining of various materials. The dust and fume collectors work to reduce your risk of exposure to off-gassing materials and fine dusts. In special circumstances, we recommend you wear an N95 mask as an extra precaution.

The dust collectors and fume collectors in the metal and woodworking shop are noisy. Please use earplugs if the noise bothers you. The Design + Technology LAB has disposable ear plugs available.

03:

CNC Router Safety

1: Pre Inspection Checklist

Are the power cords free of frays and damage?

Do you know where the emergency stop feature is located (button/switch/cord)?

Are guards in place and in good working order?

Is the machine/equipment secure and level?

Is the area around the machine/equipment free of slip/trip hazards?

Are flammable/combustible materials removed from the immediate work area?

Has the debris/material from previous operations been removed?

Are all tools/wrenches removed from the work area?

Is the machinery/equipment free of defects?

Are all your measurements and layouts complete before using machinery/equipment?

Are you wearing the required PPE (safety glasses, closed toed shoes)?

Is loose clothing secured and hair tied back?

2: Post - Operation Checklist

1. Turn the machine off. Do not walk away until it comes to a complete stop.
2. Clean up the machine and the surrounding area.

The Red Stop buttons will shutdown all power to the CNC Router.

03:

CNC Router Safety

The following outlines work safely with the CNC router machine. Please read this before operating the CNC router.

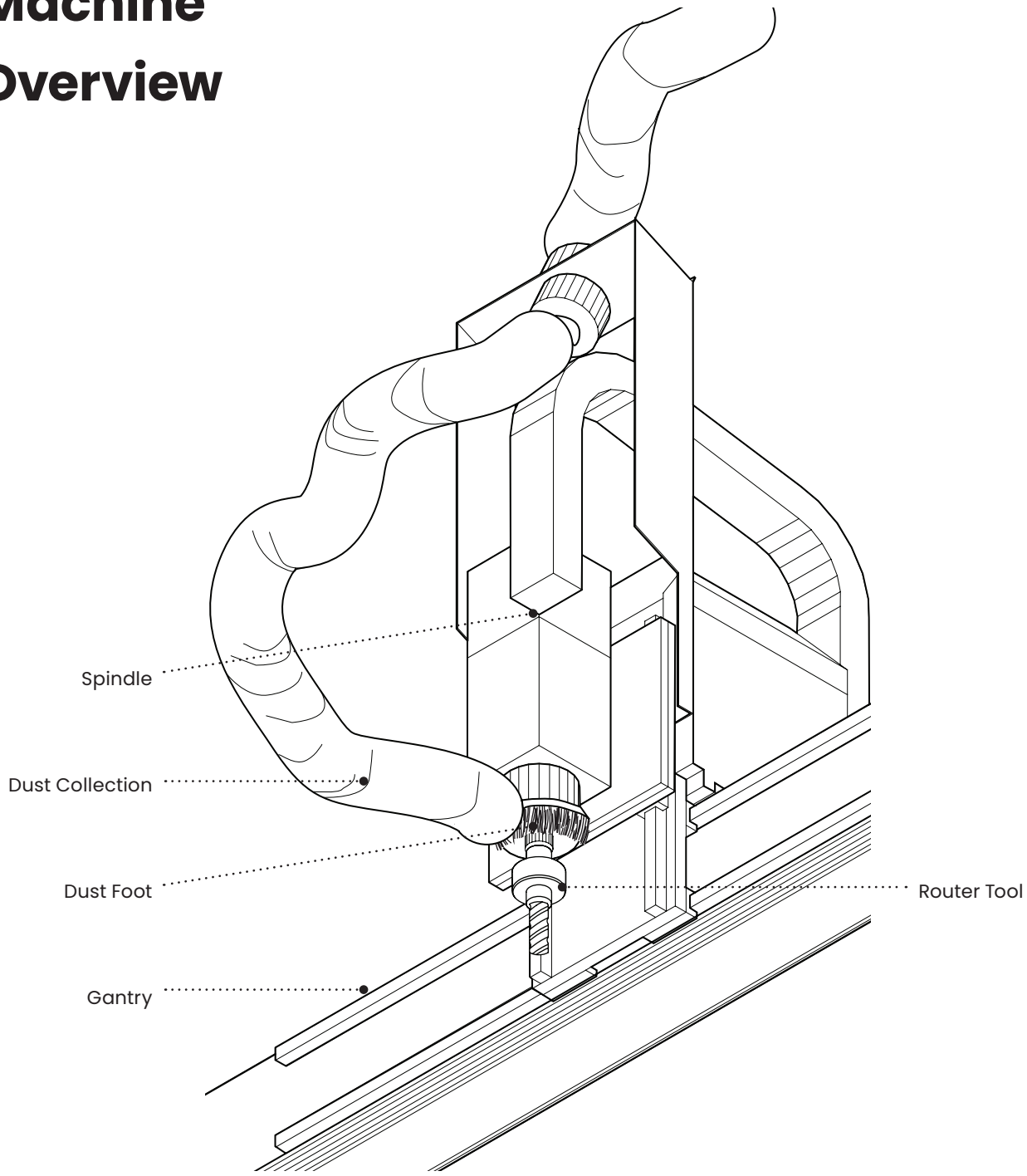
1. **Complete the pre-inspection checklist.**
2. **Never operate the CNC Router without the presence and/or approval of the Technician.**
3. **Limit wearing gloves to when necessary.** Wear gloves only when handling sharp or rough parts.
4. **Ensure that all work areas are free of hazardous obstructions.** Tools and other equipment must be returned to their proper storage place when not in use. Refer to the pre inspection checklist to the left.
5. **Keep the work area clean and uncluttered.** Oil, debris, or water on the floor/work area can cause unsafe conditions. DO NOT obstruct the light curtain path marked by red and black floor tape.
6. **Observe and follow all safety signs on the machine and in the surrounding areas.**
7. **Never operate the CNC Router if under the influence of alcohol, drugs, or any other substance or condition that may alter judgment or alertness.**
8. **Keep away from the tabletop when the spindle is turned on.**
9. **Be mindful of the CNC Router Gantry.** Stand behind the safety tape when the CNC Router is running.
10. **Ensure fire extinguishers are readily accessible.**

03:

CNC Router Safety

11. **Router tools are very sharp and should be handled accordingly.**
12. **Never leave the machine unattended during the cutting sequence.**
13. **When you are finished with the machine, complete the after-use checklist.**

04: Machine Overview



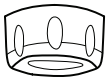
Multicam 5000 Series CNC Router Spindle

04: Machine Overview

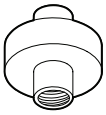
The CNC Router is capable of performing a variety of different milling operations on a variety of different materials. Before creating your 3D Machining File, it is important to consider the **type of tool** your are going to use on the CNC Router, and the **type of milling operation** your want the tool to perform. CNC Router Tools are made up of 4 main parts:

1. Collet

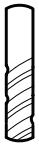
2. Collet Nut



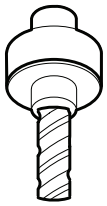
3. Tool Holder



4. Router Bit

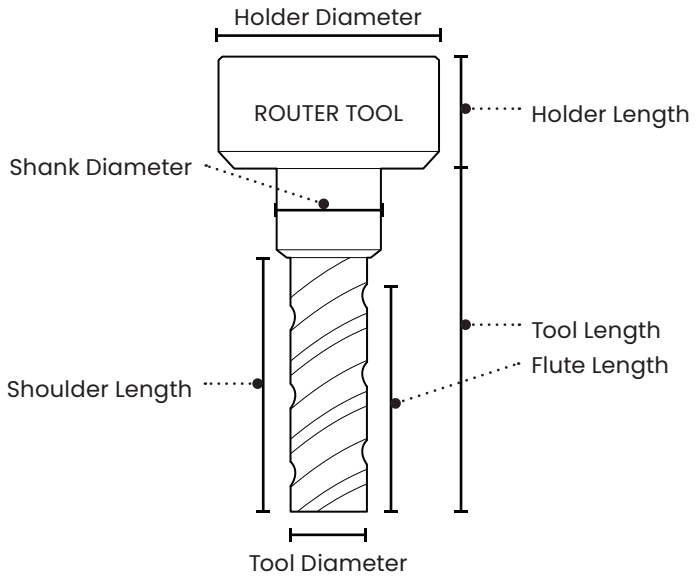


Router Tool Assembled



04: Machine Overview

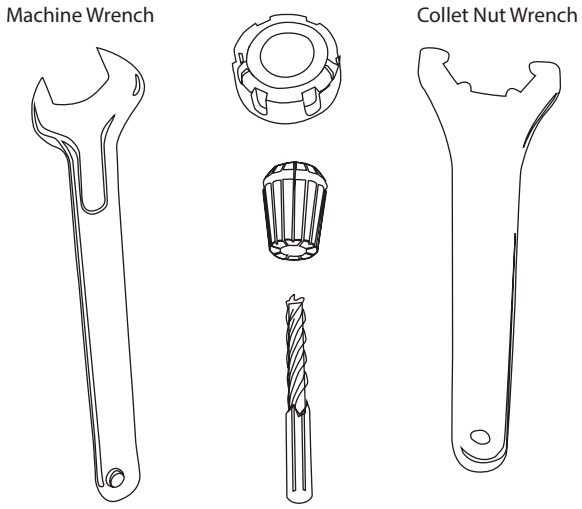
It is important to familiarize yourself with different router tools and their respective qualities when developing a machining file for your project. Router tools and bits come in a variety of sizes, and it is important to understand what these sizes mean. Below is a diagram outlining the dimensional qualities of router tools.



05:

Removing Router Tools

The CNC router is already equipped with set of tools on the automatic tool changer (ATC) carousel. However, router tools may be added or exchanged upon request. The technician will preform this task if necessary.



Important: ALWAYS calibrate the tool on the CNC and update the tool properties on RhinoCAM after changing or loading tool bits. A technician will primarily be responsible for performing this task. Refer to the *Multicam 5000 Series Operating Handbook* for tool loading procedures.

06:

Coordinates and the CNC

Rhino and RhinoCAM produce digital representations of the physical CNC and its operations. It is important to understand how space is represented digitally and how it relates to the physical CNC router machine.

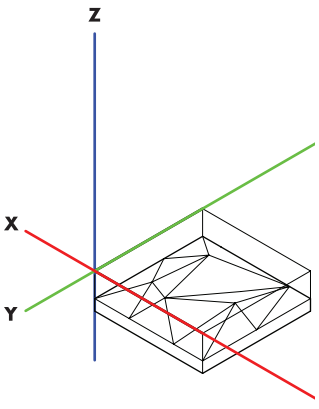
The X-Axis runs along the long edge of the CNC Router bed.

The Y-Axis runs along the short edge on the CNC Router bed.

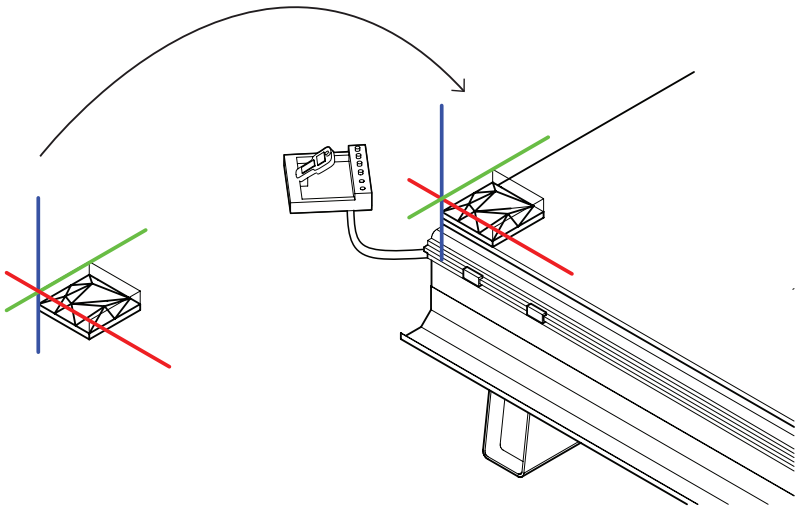
The Z-Axis runs vertical from the CNC Router bed.

The origin point (0,0,0) is the intersection of all three axes. The origin point is located at the TOP CORNER of your stock material CLOSEST to the control hub (illustrated below).

When mounting your stock material onto the CNC bed, it's LONG edge must align with the X-Axis and it's SHORT edge should align with the Y-axis. When you are developing your digital files it is important to model your stock material to correspond to its physical location on the CNC bed.



Origin in Rhino 3D Model Space



Digital Space (Rhino)

Physical Space (CNC Router Bed)

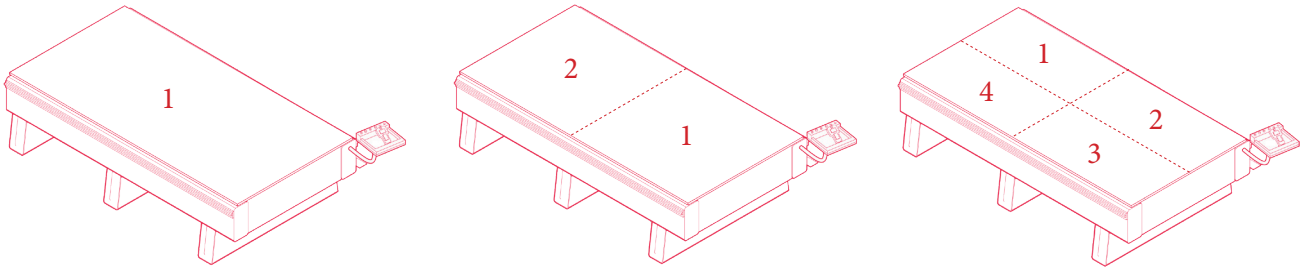
07:
Work-holding

You must ensure that your stock material is secured to the bed before you begin milling on the CNC Router.

The CNC Router bed is equipped with a vacuum hold-down mechanism to secure your stock to prevent it from shifting and lifting. An MDF spoil board is mounted on the CNC bed to prevent the router tool from cutting into the machine.

Sheet material will be slightly cupped or warped. Always place sheets on the CNC bed with the cup facing down. Tape the edges of warped or cupped boards to strengthen the vacuum force on the sheet.

You must activate each vacuum zone to correspond to the size and shape of your stock material. For example, a full 4'x8' sheet of plywood will require pressure from the entire CNC bed - activate zone 1; a 4'x4' sheet will only require pressure from half of the CNC bed - activate zone 1 + 2; and so on. Refer to the diagram below.

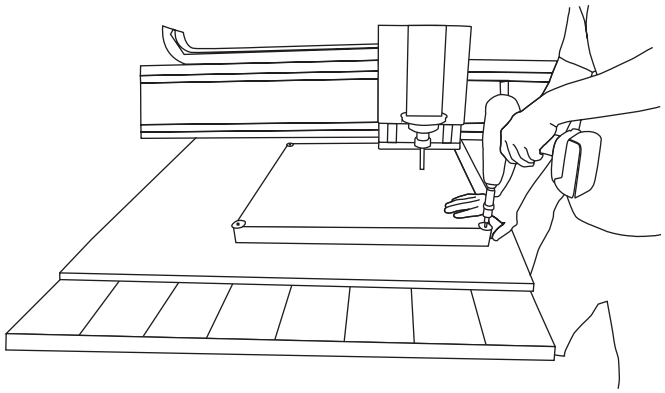


07:
Work-holding

The vacuum hold-down mechanism will be sufficient for holding down most materials with a flat bottom face. If necessary, you may secure your stock to the waste board with tapes, screws, clamps or jigs. The technician will assist you with determining the appropriate method:

Screws can be used on irregularly shaped, hard material. They should penetrate the spoil board by NO MORE than 1/2".

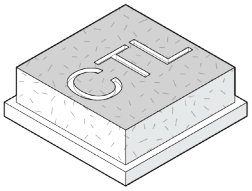
Clamps or Jigs can be used for repetitive cuts on small, irregularly shaped material



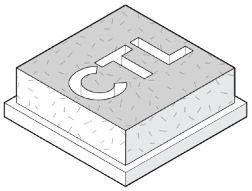
08:
Milling Operations

There are numerous methods of CNC milling operations that can be performed by the Multicam 5000 Series Router. These operations can be broken down into two subcategories: **2D milling** and **3D milling**

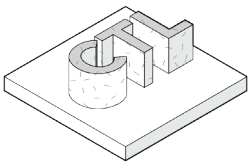
2D milling operations involve cuts along the X and Y axis. These operations remove material at a set depth as the Z-axis is fixed at every **pass**. Engraving, pocketing, and profiling are 2D milling operations that are commonly used at the Design + Technology Lab.



Engraving can be used to etch labels onto each profile for identification during assembly. Conventionally, labels are strategically placed on components that are hidden or have a minimal presence in the final project.



Pockets and other internal details that are milled partway through the stock material. This can include functional and aesthetic details such as slots, dados, and rabbets. These details will be explained further in this manual.

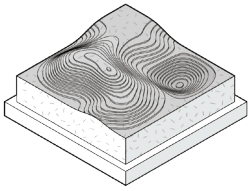


Profiling removes material from the outside perimeter of each profile.

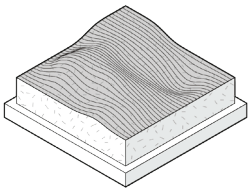
A **Pass** refers to one full movement about the x,y axis that the spindle makes at any given depth. Multiple passes may be required to complete a cut to relieve stress on the router toolbit.

08:
Milling Operations

3D milling operations involve cuts along the X,Y and Z axis. These operations remove material by simultaneously moving the tool in all directions across the surface of your stock material. Horizontal roughing and parallel finishing are the two milling operations commonly used at the Design + Technology Lab.



Horizontal Roughing removes the bulk of your stock material and creates stepped down contours following the curvature of your modeled surface. It removes material incrementally in layers to relieve stress on the tool bit during the horizontal finishing pass.

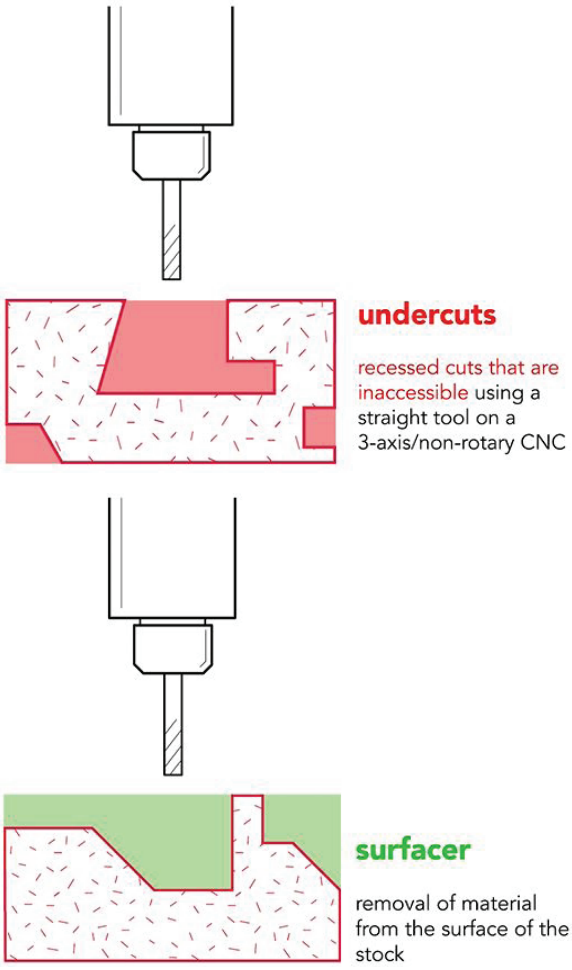


Parallel Finishing removes material continuously along the curvature of your modeled surface. It creates a smooth surface depending on how close together each pass is from the next.

08:
Milling Operations

Just like any other tool in the lab, the CNC has its own unique limitations. You will need to take this into consideration while developing your project for fabrication. These limitations are inherent to the 3-axis milling process and result from the available tools and its single plane movement.

Undercuts cannot be achieved on the 3-axis CNC as the spindle is fixed and cannot reach recessed areas under the surface of the stock material. Rather, the machine can only remove material across the surface of the stock.



It is also important to consider that the tools on the CNC mill will always have a radius meaning that the corners of internal cuts will always be rounded. The smaller the tool radius, the sharper the corners will appear.

09: Materials

The CNC Router is capable of milling a variety of different materials referred to as *stock.* Common materials milled on the Multicam 5000 Series Router at the Design + Technology Lab include:

Solid wood species - both hardwoods and softwoods.

Composites including, but not limited to: Medium Density Fibreboard (MDF), High Density Fibreboard (HDF), Plywood, Oriented Strand Board (OSB), etc.

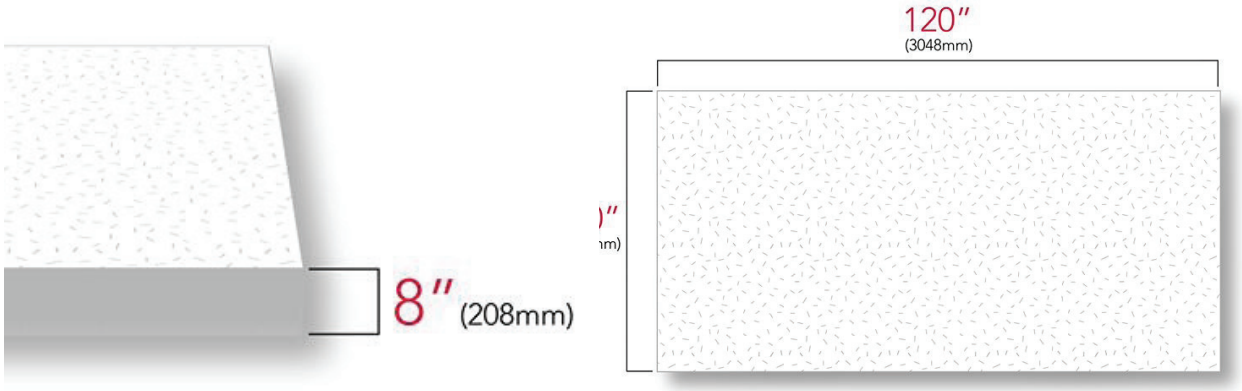
Sheet plastics including, but not limited to: acrylic, Polyethylene Terephthalate Glycol (PETG), Styrene, etc.

Rigid Foam products including, but not limited to: Expanded Polystyrene (EPS) and Extruded Polystyerene (XPS).

Aluminum

Consult a Technician if you wish to mill a material that is not mentioned above. It is important to know that your material will affect milling time.

The maximum dimensions of your stock material must not exceed 120”x 60”x 8” (3048mm x 524mm x 208mm)



10: Machine Appointment Procedure

The Design + Technology Lab is here to assist you in all of your CNC projects! Technicians are happy to consult with you regarding CNC file setup strategies and identify areas of potential exploration and development to improve the quality of your milling project. Consult a technician when booking your CNC appointment to ensure you are prepared for your appointment date.

Appointment Policy: Be sure to give yourself adequate time to consult a technician about your project before booking your CNC appointment. YOU ARE RESPONSIBLE FOR:

- Paying for your appointment.
- Ordering, purchasing, and arranging storage for your materials.
- Developing a digital file.

The operating lab technician MAY DECLINE your file from being processed if it does not meet the level of development outlined in this manual.

11:

The CNC Process

CNC stands for **C**omputer **N**umerically **C**ontrolled processes. The Multicam 5000 Series machine is a 3-axis CNC router mill. This means that the machine controls the movement of a high speed rotary spindle is in three axes: x,y, and z. In other words, the spindle traverses along a gantry and rail system to move up, down, left, right, forward, and backward. These movements are determined by a set of G-code generated from a digital file.

Whether you wish to 2D or 3D mill your project, you must follow the procedures outlined below. These steps will be slightly different for each milling operation. They will be expanded further in this manual.

CAD

Computer Aided Design

a program used to create precision drawings, technical illustrations, and 3D models.

CAM

Computer Aided Manufacturing

a program that uses numerical control to create G-code which acts as detailed instructions that drive the CNC router

CNC

Computer Numeric Control

uses the provided G-code to process a piece of material to meet the instructed specifications

11:
The CNC Process

STEP 1: Develop your CAD model. Rhinoceros is the primary 3D modeling software for creating milling files for the CNC at the Design + Technology Lab. If you modeled your design on other programs such as: AutoCAD, Maya, 3D Studio Max, Revit, Solidworks, Blender, Adobe Illustrator, etc. you may import your model into Rhino as a .3dm file.

STEP 2: Process your model through CAM. RhinoCAM is a CAM plug-in for Rhino that enables you to generate machining instructions for the CNC router from your digital model. These instructions are processed as G-code language for the CNC router to follow. RhinoCAM also allows you to preview and simulate your machining operations. A Technician must approve of the operations generated in this step before proceeding to step 3.

STEP 3: Mill your project. This stage involves the physical milling of your machining process. You will work through this stage with the Technician.

You are expected to prep your file to a caliber appropriate for coordinating next steps with a Design + Technology Lab Technician! Failure to do so effects the efficiency on the CNC appointment. Your supporting technician reserve the right to decline your files if they are not complete.

Step 1:

Develop your CAD model

File Etiquette is an important aspect to maintain throughout the CNC process. This ensures a smooth transition between steps and an ease of collaboration between you and your supporting technician. Your file should be organized into layers and sub layers that consolidate necessary information needed for the CNC process. DO NOT include process work in this file as it will clutter the Rhino workspace and produce errors.

ALWAYS model in INCHES at 1:1 scale! The Multicam 5000 Series CNC Router is programmed to read all units in inches; therefore, if your file was modeled in millimetres the machine will automatically read those units as inches. To change your file's units to inches without affecting the scale of your design open:

[Rhino Options > Document Properties > Units > Units and Tolerances > set Model Units to Inches]

When you **import** your model into Rhino from another modeling software be sure to take extra precautionary steps to fix any **geometry errors** including, but not limited to, the following:

Unintentional Open Curves *[Rhino command: join]*

Non-manifold errors *[Rhino command: analyze > edge tools > show edges > edge analysis > non-manifold edges]*

Overlapping geometry *[Rhino command: SelDup > delete]*

Text must be converted to VECTOR OUTLINES prior to import; otherwise, it may be lost in conversion. Text can also be modeled or converted in Rhino.

[Rhino command: SelText > explode OR utilizing TextObject to create vector text]

Step 1:

Develop your CAD model

File Etiquette

Units & Scale

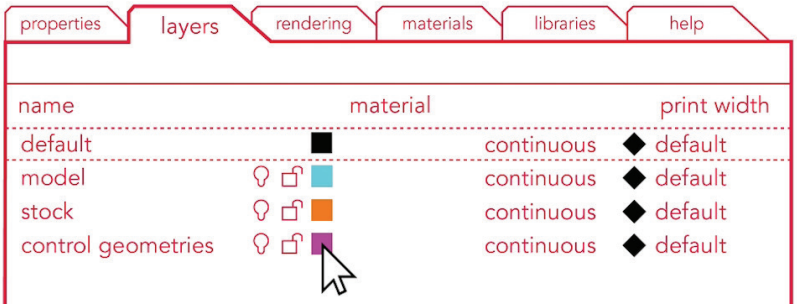
Import Settings

2D Milling

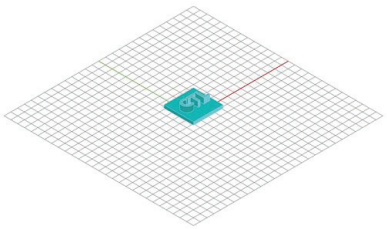
- **Model**
- Stock
- Control Geometries
- Joinery techniques

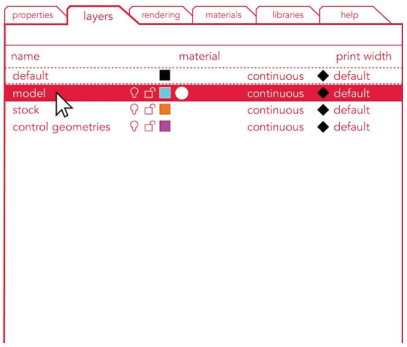
3D Milling

Begin by creating layers for your model, stock, and control geometries as shown above. Differentiate your layers by colour for clarity.



Import or model your design at 1:1 on the model layer. This model will be used for extracting control geometries while serving as reference for you and your supporting technician. It will provide a better understanding of what you wish to achieve from your CNC appointment.





Step 1:

Develop your CAD model

File Etiquette

Units & Scale

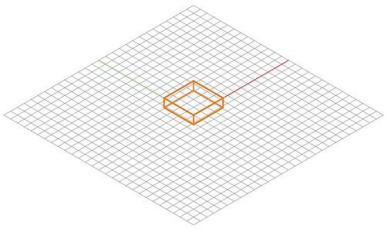
Import Settings

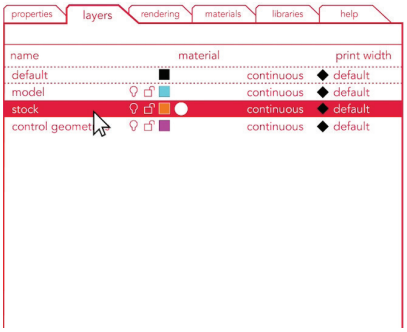
2D Milling

- •Model
- **•Stock**
- •Control Geometries
- •Joinery techniques

3D Milling

On the stock layer, accurately model your stock material to represent its actual dimensions and thickness. Use a caliper to measure thickness (ex. A sheet of ¾" Baltic birch is not exactly ¾" thick; rather, approx. 15/16" thick). When milling from an irregularly sized sheet of material, model the length and width accordingly. Position the top surface of your stock below the x,y world plane. Make sure one vertex is at the origin and the longer edge is parallel to the x axis as shown below:





Step 1:

Develop your CAD model

File Etiquette

Units & Scale

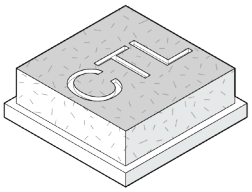
Import Settings

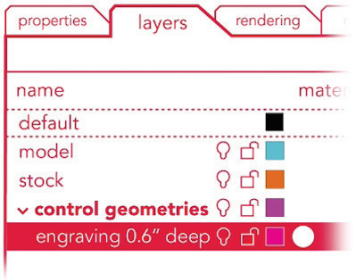
2D Milling

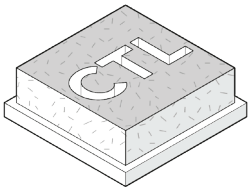
- •Model
- •Stock
- •**Control geometries**
- •Joinery techniques

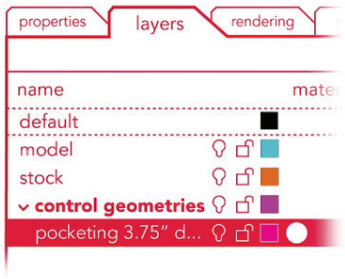
3D Milling

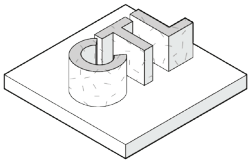
The **control geometries** layer is where all of your profiles are consolidated and organized into sub layers. Each sublayer should be labeled based on the milling operation and depth they represent (as shown above). The order in which they appear will only be important in Step 2.

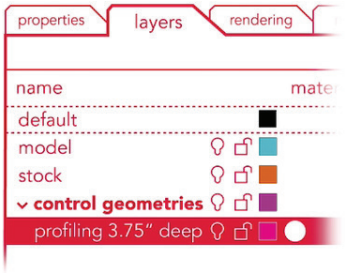












Profiles & Curve Regions are 2D planar curves in Rhino that represent the areas of material that will be removed by the CNC router. These curves (referred to as curve regions on RhinoCAM) are then translated into toolpaths.

Toolpaths are the X,Y,Z paths the CNC router tool will follow as it mills your project.

Step 1:

Develop your CAD model

File Etiquette

Units & Scale

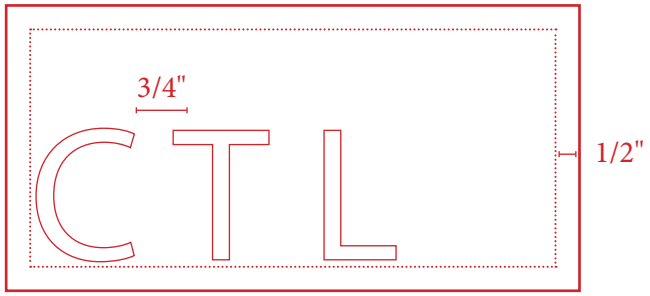
Import Settings

2D Milling

- •Model
- •Stock
- •**Control geometries**
- •Joinery techniques

3D Milling

Place your profiles to fit within a minimum of ½" margin from the EDGE of your stock. This margin provides perimeter support material and accounts for the position of any screws and fasteners you may need to use for securing your workpiece to the machine. Leave at least ¾" of material IN BETWEEN your curve regions. The reason is that a ¼" bit is commonly used for milling. If a project requires a larger bit, create a larger margin and space out your profiles accordingly. NEST AS TIGHT AS POSSIBLE to reduce the amount of waste material!



Step 1:

Develop your CAD model

File Etiquette

Units & Scale

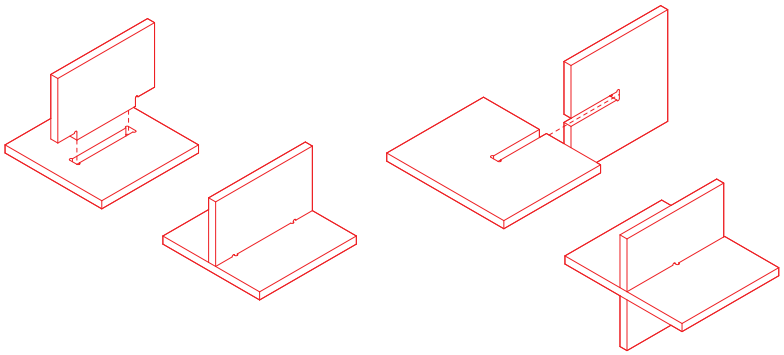
Import Settings

2D Milling

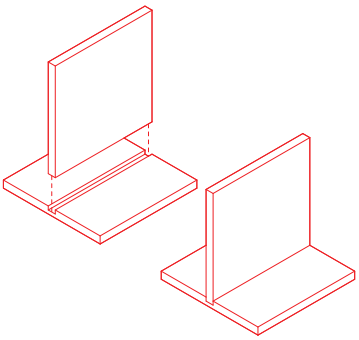
- •Model
- •Stock
- •Control Geometries
- •**Joinery techniques**

3D Milling

Fundamental **joinery details** can be milled directly into your profiles on the CNC. These are additional milling operations that must be isolated in their individual layers. It is important to give yourself time to prototype your joints well before milling your final piece. Mill samples of your joints to test how tight or loose they fit together. Some widely used CNC joinery techniques include but are NOT limited to the following:



Slots. When joining two perpendicular boards edge-to-face, slots allow them to intercept one another. Slots should be slightly larger than the width of your stock material.



Dados. An alternative edge-to-face detail can be created by milling a channel into the face of your profile. Be sure to extend the control curve past the edge of the profile for the bit to clear the perimeter of the board as shown above.

Step 1:

Develop your CAD model

File Etiquette

Units & Scale

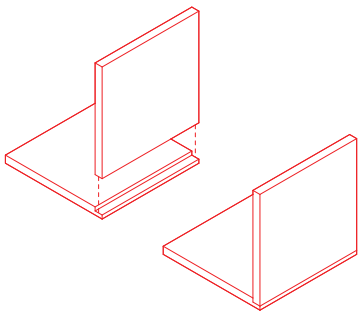
Import Settings

2D Milling

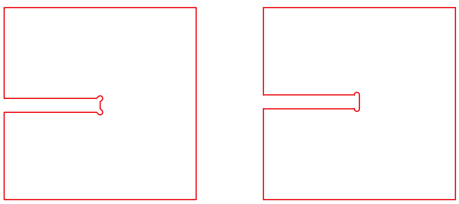
- •Model
- •Stock
- •Control Geometries
- •**Joinery techniques**

3D Milling

Rabbets. When joining two perpendicular boards edge-to-edge, rabbets help with location and alignment during glue-up. Rabbet joints are created by milling a channel into your stock at the edge of your profile in the same way as a dado.



Dog Bones. You must consider that every INTERNAL cut will have a radius; therefore, any connecting pieces will not have an exact fit. Dog Bone details remove material from the corners of an internal corner so that two boards can join together. There are many ways to create a dog Bone joint and the examples below are only two conventional dog Bone details.



A custom dog Bone joint (right) requires its own control curve or point that should be isolated in its own layer. Standard dog Bones (left) can be programmed into the toolpath on RhinoCAM and will be explained further in step 2.

Step 1:

Develop your CAD model

File Etiquette

Units & Scale

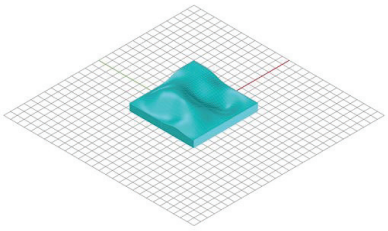
Import Settings

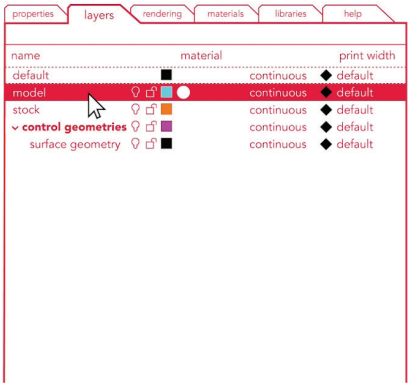
2D Milling

3D Milling

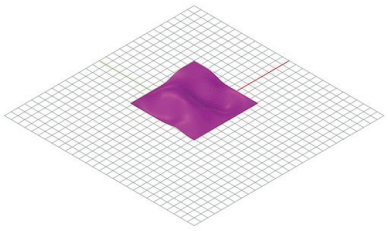
- **Model**
- **Control geometries**
- Stock

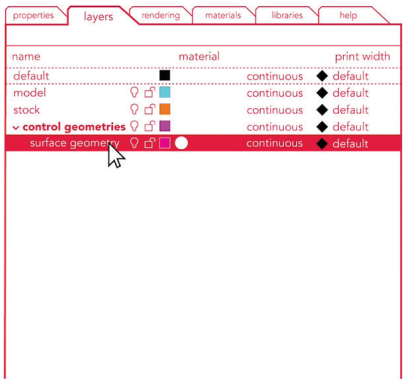
All file preparation conventions for 2D Milling process apply to the 3D Milling process as well. Begin by importing or modeling your design at 1:1 in the **model layer**





Next, copy and paste the TOP SURFACE of your model into the **surface geometry** sublayer. This layer should only contain one continuous NURBS SURFACE and not a MESH. NURBs Surface stands for Non-uniform Rational Basis Spline. It is an accurate representation of curves and surfaces.





Step 1:

Develop your CAD model

File Etiquette

Units & Scale

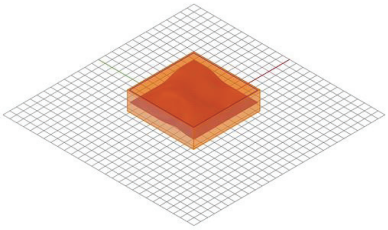
Import Settings

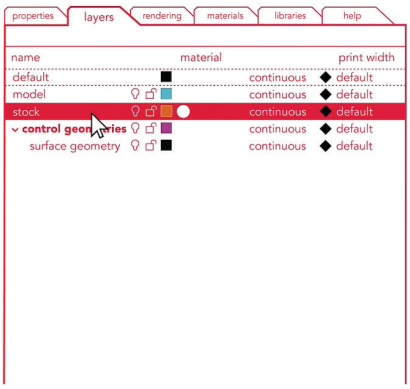
2D Milling

3D Milling

- •Model
- •Control geometries
- •**Stock**

Your **stock** material should encapsulate the entire surface geometry. Keep in mind that every material has varying hardnesses and densities; therefore, some material will take longer to mill than others. If you plan on laminating stock material be sure to allow it to cure completely prior to your CNC appointment.





Step 2:

Process your model through CAM

RhinoCAM Setup

2D Milling

3D Milling

Post

Once you have completed an organized and cohesive CAD file, you are ready to move on to the CAM portion of the CNC process. This step must be done IN LAB on the Design + Technology Lab's dedicated computer.

Before inputting any data into **RhinoCAM** be sure to check that the program is set to MILL, the machine is set to 3-AXIS, and the post settings are set to MULTICAM-INCH-REV. (figure 1)

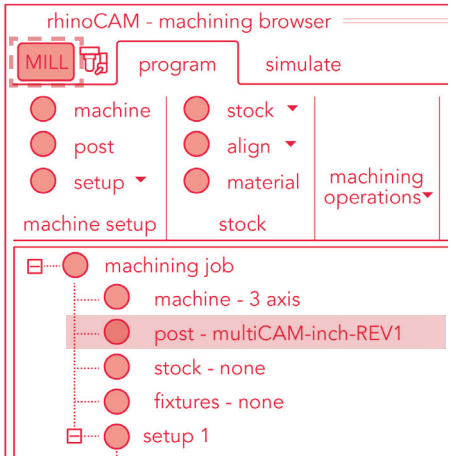


figure 1

The first geometry to input into RhinoCAM is the stock material. Select the geomtry representing your stock material, click STOCK and set STOCK FROM SELECTION. (figure 2)

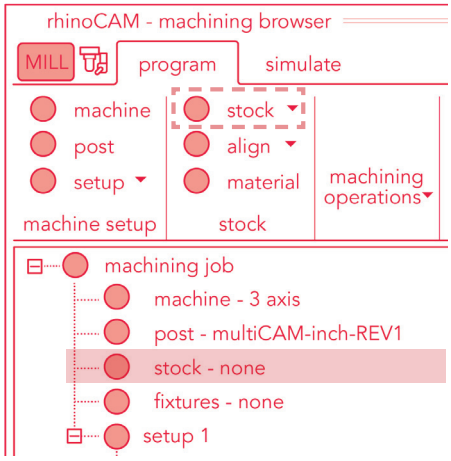


figure 2

Step 2:

Process your model through CAM

RhinoCAM Setup

2D Milling

3D Milling

Post

At this stage you may start inputting the curve regions into RhinoCAM. You will work with the operating lab technician to:

Select machining tool bits

Generate toolpaths

Simulate toolpaths

Post g-code instructions to the CNC router

Step 2:

Process your model through CAM

RhinoCAM Setup

2D Milling

- **Machine tool bits**
- Generate toolpaths
- Simulate

3D Milling

Post

A wide selection of machine tool bits are available; however, this section will cover the most commonly used tool bits at the Design + Technology Lab for **2D milling**. Each tool will have different lengths, diameter, cut direction, tool number, and FLUTE count. If your desired tool is not available, please make arrangements with a technician in advance.

Vee mill: a "V" shaped end mill used for engraving.

Flat mill up-cut: a spiral end mill with a flat tip used for pocketing and profiling. This tool forces chips UP as it rotates CLOCKWISE. It will create a clean edge at the BOTTOM surface.

Flat mill down-cut: a spiral end mill with a flat tip used for pocketing and profiling. This forces chips DOWN as it rotates CLOCKWISE. It will create a clean edge at the TOP surface.

Flat mill compression bit: a spiral end mill with a flat tip used for pocketing and profiling. It has cutting edges, known as FLUTES, that cut in BOTH directions. It will create a clean edge at the TOP and BOTTOM surfaces.

Flute: refers to each cutting spiral on

Step 2:

Process your model through CAM

RhinoCAM Setup

2D Milling

- **Machine tool bits**
- Generate toolpaths
- Simulate

3D Milling

Post

Single flute flat mill: a spiral end mill with ONE FLUTE and a flat tip used for milling SHEET PLASTICS.

Step 2:

Process your model through CAM

RhinoCAM Setup

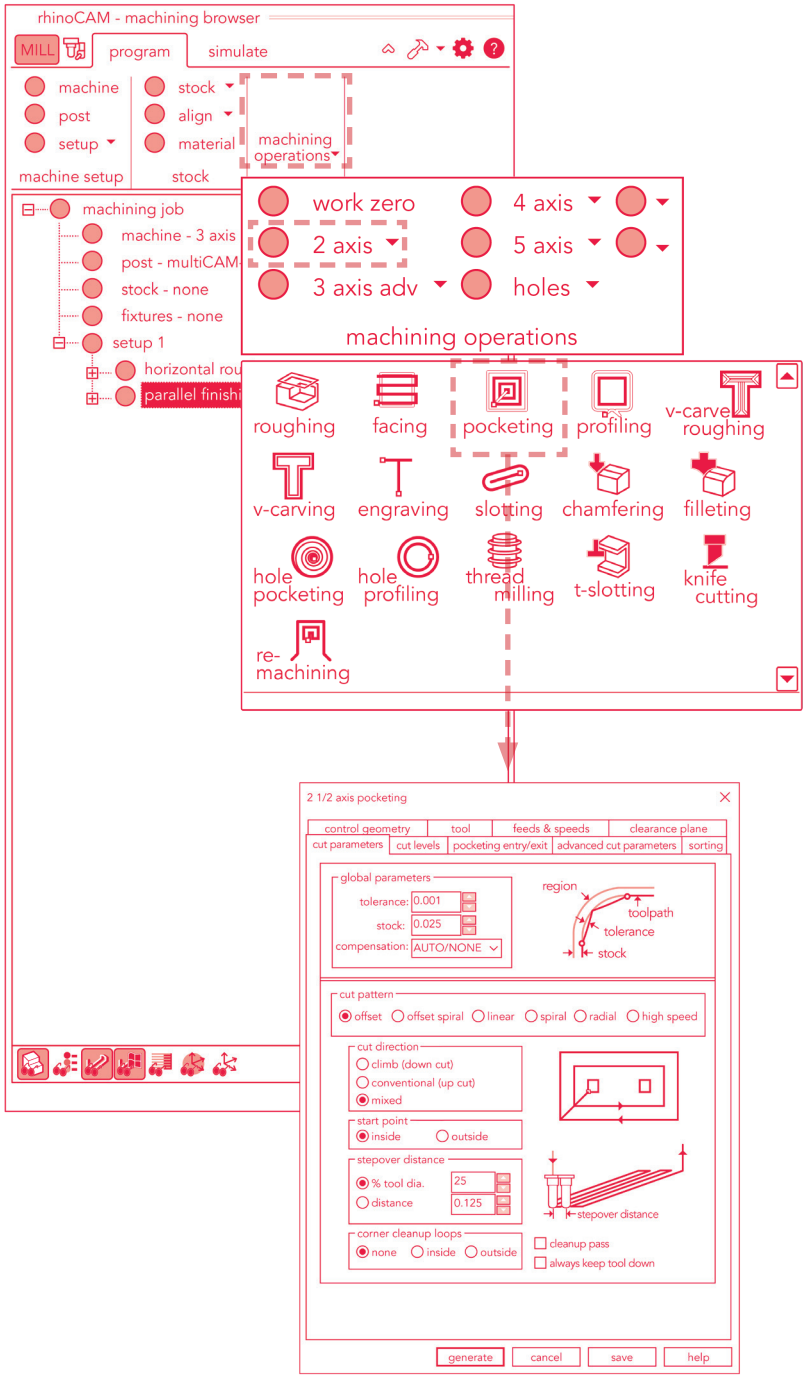
2D Milling

- •Machine tool bits
- **•Generate toolpaths**
- •Simulate

3D Milling

Post

A wide selection of 2-axis milling operations can be used when **generating toolpaths**. This section of Step 2 will briefly explain the essential milling parameters that are common to engraving, pocketing, and profiling. Open your desired machine operation as shown in the example below:



Step 2:

Process your model through CAM

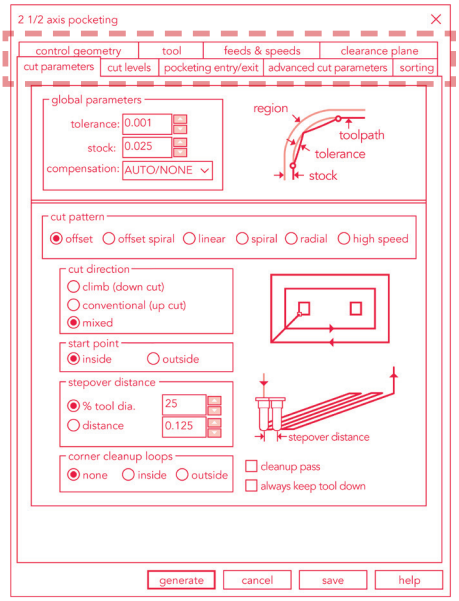
RhinoCAM Setup

2D Milling

- •Machine tool bits
- **•Generate toolpaths**
- •Simulate

3D Milling

Post



Control geometry: Rhino curves are input into RhinoCAM under the control geometry tab. RhinoCAM translates these curves into toolpaths. If the curve is moved in the Rhino workspace, the toolpath and its properties will not update.

Tool: select the proper tool for the operation under this tab. You can preview the properties of each tool by double clicking on the tool. DO NOT change these properties.

Feeds & speeds: These settings control the spindle rotation and the speed at which it travels in each direction. Feed and speed rates vary depending on your stock material and the properties of the tool bit. They are preset to mill composites such as plywood and MDF. Work with a technician to adjust these settings according to your material.

Entry/exit: This determines where the bit enters and exits the stock. This is usually set to NONE. If you are concerned about burn marks and/or indents appearing on your finish edge, set to LINES & ARCS. Note that this may require you to create more space in between your profiles.

Step 2:

Process your model through CAM

RhinoCAM Setup

2D Milling

- •Machine tool bits
- **•Generate toolpaths**
- •Simulate

3D Milling

Post

Cut levels: The depth at which the tool will travel at each pass is programmed in this tab. You can also program the number of passes by adjusting the *Rough Depth/Cut* value. Set the *Total Cut Depth* to a value that is approximately 0.05" deeper than the actual thickness for the tool bit to mill completely through your stock material.

Advanced Cut Parameters: If you are milling a hole or a profile that is SMALLER than 6", be sure to program TABS to the operation here. Tabs are small pieces of material that remain in between your profile and the stock material to hold it steady. Tabs prevent material from shifting and lifting from the machine bed. You can adjust the size and number of tabs that remain on your material.

Cornering Parameters: an optional setting if you require Dog Bones or T-bones to be milled in certain pockets and slots.

Clearance Plane: the maximum height above your stock material at which the spindle will retract. It should appear as a red plane hovering DIRECTLY ABOVE your stock material. If the clearance plane extends past your stock material, it is detecting another geometry and will produce errors.

Sorting: This determines where the machine will start and finish cutting. Set to minimum distance sort.

Step 2:

Process your model through CAM

RhinoCAM Setup

2D Milling

- •Machine tool bits
- **•Generate toolpaths**
- •Simulate

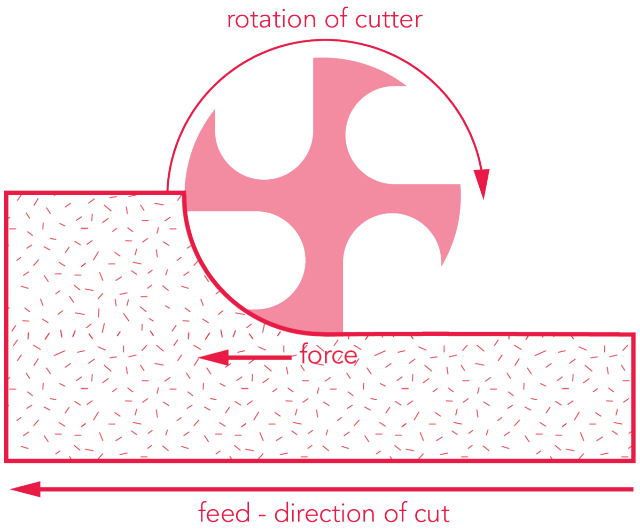
3D Milling

Post

Cut Parameters: there are a few parameters you need to consider when programming your toolpaths listed below:

- STOCK TOLERANCE is the amount of material left on the stock after the final pass to. Set to 0 to mill completely to the edge. Note that the CNC will not mill to your modeled geometry with 100% accuracy. Tolerances account for this discrepancy.
- Cutting Side determines which side of your control curve that the toolpath will appear. You may need to use the *flip* command in Rhino to change the direction of your control curves if the toolpath appears on the wrong side of your profile.
- Cut direction refers to the direction that the tool will travel along the toolpath. INSIDE and OUTSIDE cuts should be done in separate operations; therefore, they should also be isolated in their own layers.There are three options:

1. CLIMB CUTS push chips AWAY from the direction of travel of the router tool and produces less grain tear-out. This results in a cleaner edge.



Step 2:

Process your model through CAM

RhinoCAM Setup

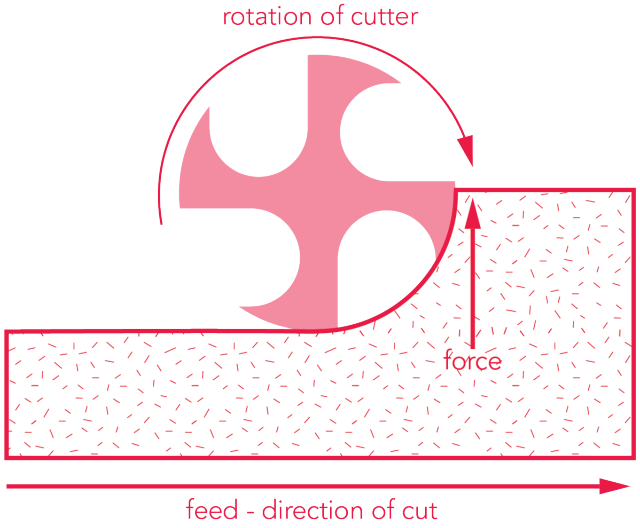
2D Milling

- •Machine tool bits
- **•Generate toolpaths**
- •Simulate

3D Milling

Post

2. CONVENTIONAL CUTS push chips IN FRONT of the tool bit as it moves along the toolpath. This can create a marred edged finish and re-cutting of chips. Conventional cuts have a higher risk of lifting the stock if it is not secured properly to the CNC bed.



3. MIXED CUTS allow the tool to alternate between climb and conventional cuts between passes.

Step 2:

Process your model through CAM

RhinoCAM Setup

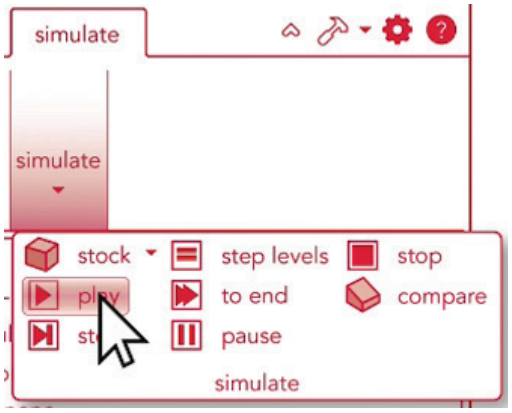
2D Milling

- •Machine tool bits
- •Generate toolpaths
- **•Simulate**

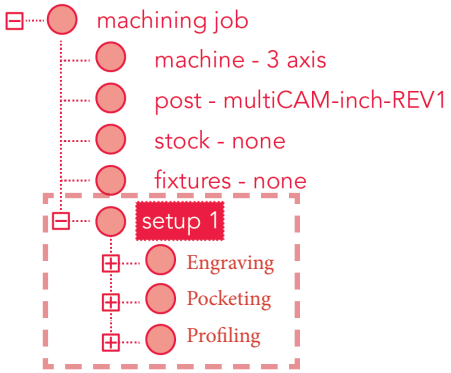
3D Milling

Post

Simulate your milling operation once your toolpaths are generated. Simulations serve as an animated representation of the milling process. It will show the movements of the tool and where it will remove material from your stock. Open the *simulate* tab as follows:



It is important to consider the order of each milling operation in the *setup* drop down sequence below. Shallow, internal cuts must be milled first followed by deeper, perimeter cuts.



Step 2:

Process your model through CAM

RhinoCAM Setup

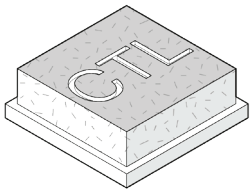
2D Milling

- •Machine tool bits
- •Generate toolpaths
- **•Simulate**

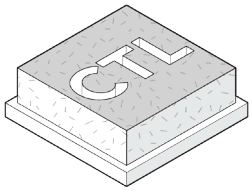
3D Milling

Post

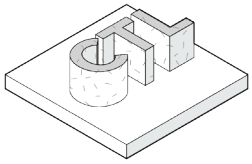
1. **Engraving** is the most shallow pass and will be milled first.



2. **Pockets** and other internal details that are milled partway through the stock material come after engraving in the sequence. This can include any holes, internal slots, dados, rabbets, and dog Bones.



3. **Profiling** should be the last operation in the sequence. At this stage, the outside perimeter of each profile is milled.



Step 2:

Process your model through CAM

RhinoCAM Setup

2D Milling

3D Milling

- **Machine tool bits**
- Generate toolpaths
- Simulate

Post

The **primary machine tool bit** Used for **3D milling** at the Design + Technology Lab is known as a BALL MILL. It is characterized as a spiral end mill with a ROUNDED tip. It is available in diameters of 3/4", 1/2", and 1/4". Each diameter will determine the resolution and smoothness of your final surface. The maximum depth to which you can mill depends on the tool length of the longest ball mill on the CNC Carousel. If your project requires a ball mill tool with a different diameter and length than what is available, please make arrangements with a technician.

Step 2:

Process your model through CAM

RhinoCAM Setup

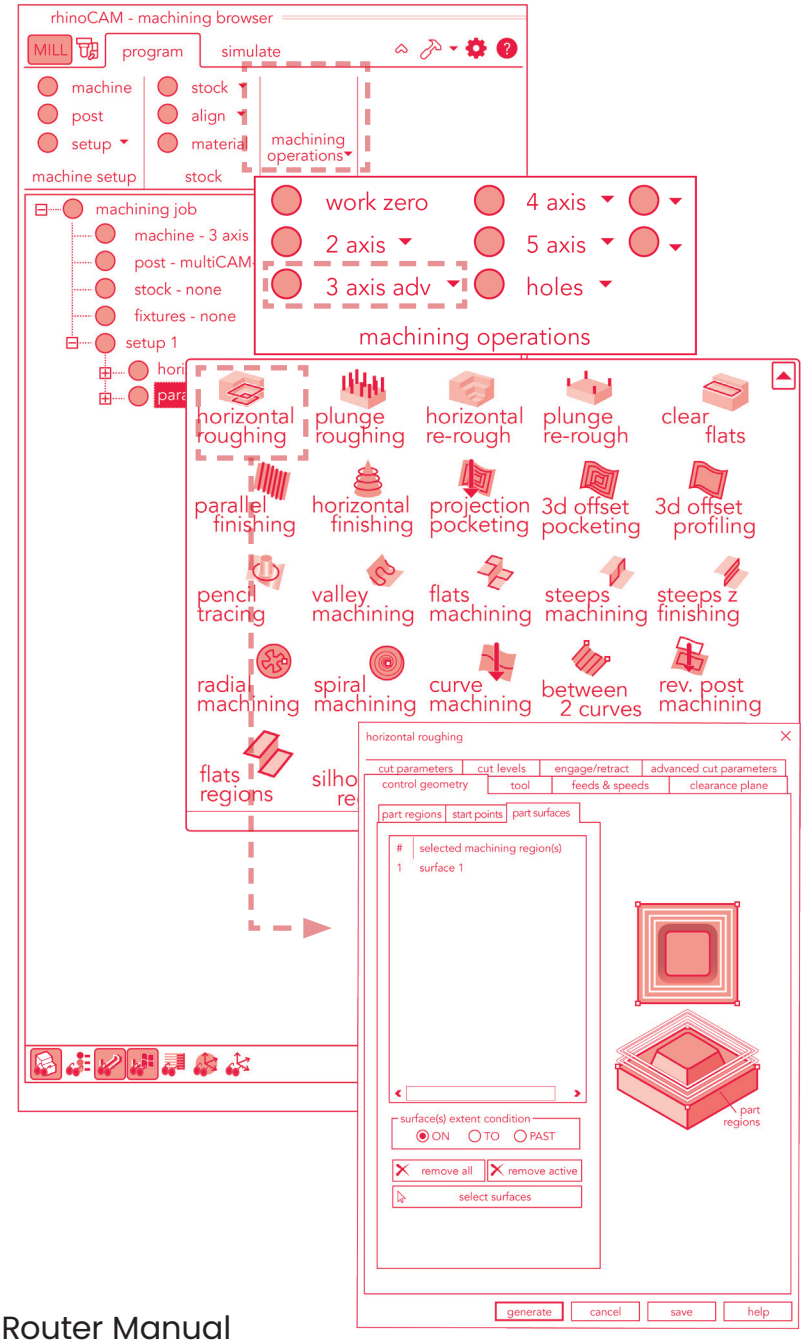
2D Milling

3D Milling

- •Machine tool bits
- **•Generate toolpaths**
- •Simulate

Post

A wide selection of 3-axis milling operations can be used when **generating toolpaths.** However, the primary machine operations used at the Design + Technology Lab are HORIZONTAL ROUGHING and PARALLEL FINISHING. This section of Step 2 will briefly explain the essential milling parameters that are common to both operations (highlighted below). Open your desired *machine operation* as shown in the example below:



Step 2:

Process your model through CAM

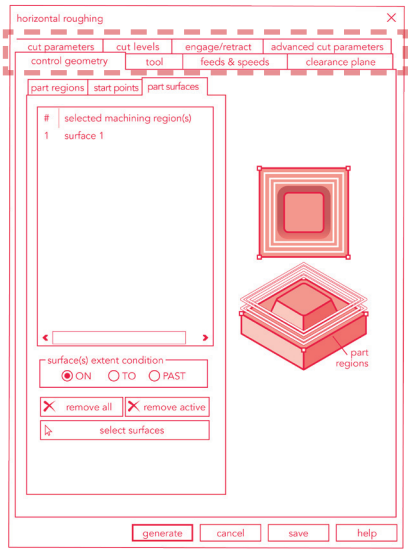
RhinoCAM Setup

2D Milling

3D Milling

- Machine tool bits
- **Generate toolpaths**
- Simulate

Post



Control geometry: Rhino surfaces are input into RhinoCAM under the control geometry tab. RhinoCAM translates these surfaces into toolpaths. If the surface is moved in the Rhino workspace, the toolpath and its properties will not update.

Tool: select the proper tool for the operation under this tab. You can preview the properties of each tool by double clicking on the tool. DO NOT change these properties.

Feeds & speeds: These settings control the spindle rotation and the speed at which it travels in each direction. Feed and speed rates vary depending on your stock material and the properties of the tool bit. They are preset to mill composites such as plywood and MDF. Work with a technician to adjust these settings according to your material.

Cut levels: these parameters only apply to horizontal roughing. It determines the depth at which the tool cut at each pass and is based on a percentage of the tool diameter. It also determines how many stepped-down contours appear after the horizontal roughing operation

Step 2:

Process your model through CAM

RhinoCAM Setup

2D Milling

3D Milling

- •Machine tool bits
- **•Generate toolpaths**
- •Simulate

Post

Cut Parameters allow you to modify the step-over distance based off of a percentage of the tool diameter. This will affect the resolution of your final surface: a smaller step-over distance will result in a smoother finish.

Clearance Plane is the maximum height above your stock material at which the spindle will retract. It should appear as a red plane hovering DIRECTLYABOVE your stock material. If the clearance plane extends past your stock material, it is detecting another geometry and will produce errors.

Step 2:

Process your model through CAM

RhinoCAM Setup

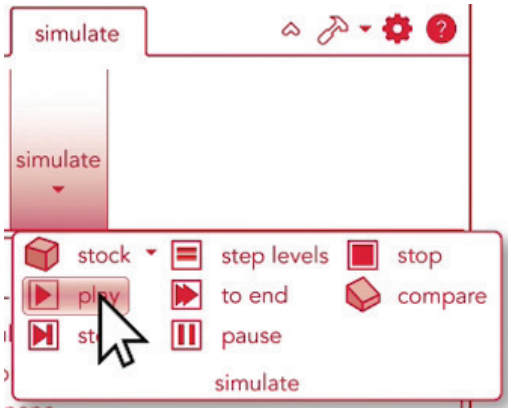
2D Milling

3D Milling

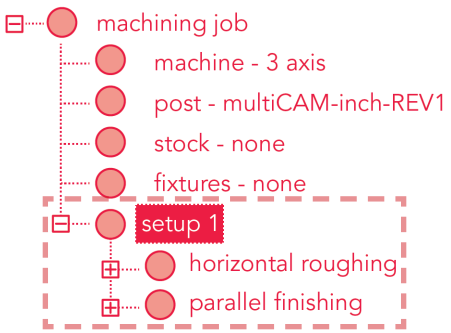
- •Machine tool bits
- •Generate toolpaths
- **•Simulate**

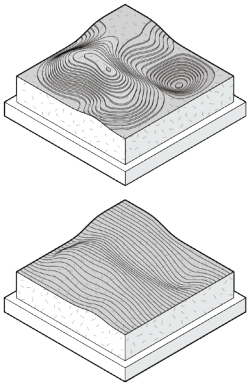
Post

Simulate your milling operation once your toolpaths are generated. Simulations serve as an animated representation of the milling process. It will show the movements of the tool and how smooth your final surface may appear. Open the *simulate* tab as follows:



It is important to consider the order of each milling operation in the *setup* drop down sequence below. While milling hard materials, HORIZONTAL ROUGHING must be completed before PARALLEL FINISHING. Failure to do so will result in faulty cuts, broken tool bits, and the stock being flung from the CNC bed. For rapid prototyping with foam, you may skip horizontal roughing. Keep in mind that this will result in a rough finish surface.





Step 2:

Process your model through CAM

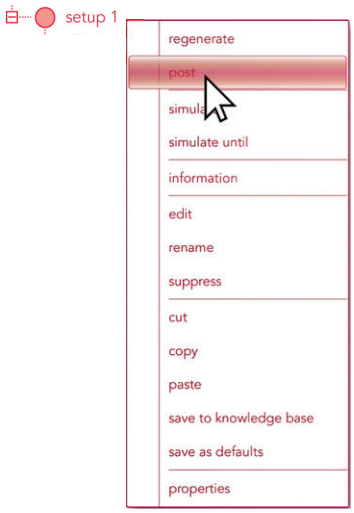
RhinoCAM Setup

2D Milling

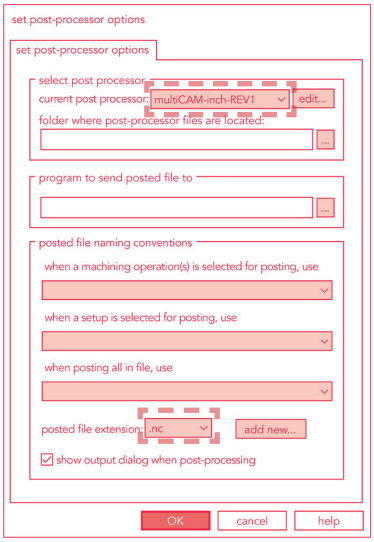
3D Milling

Post

When you and the technician are satisfied with your RhinoCAM settings, you may **post** your file as G-code. Right click on your *setup* and select *post.*



Be sure that the *current post processor* is set to *multiCAM-inch-REV1* and the posted file extension is *.nc* as highlighted below. Double check with the technician to make sure that the file is posted to the correct file location.



Step 2:

Process your model through CAM

RhinoCAM Setup

2D Milling

3D Milling

Post

Name your file according to the following naming conventions:

Your Initials_Project Name_Post Attempt #.nc

(ex. *DT_Lab Furniture_Shelf 01.nc*)

Note: there may be instances where you must re-post your file. Including a *Post Attempt* will help you and the technician locate your file on the Multicam CNC controller in Step 3.

Step 3:

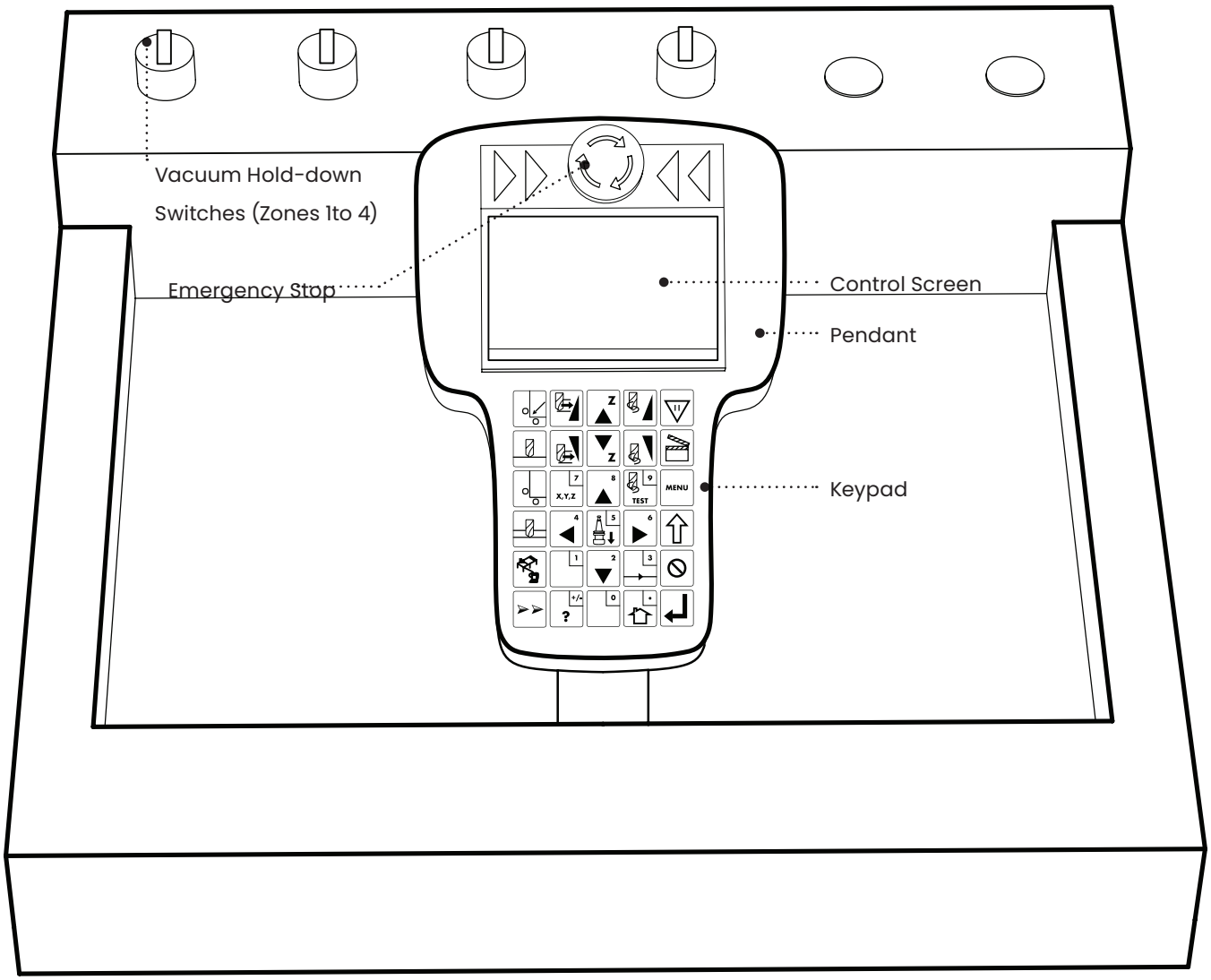
Mill your Project

Control Hub

Pendant Keypad

Milling procedure

Troubleshooting



Multicam 5000 Series CNC Router Control Hub

Step 3:

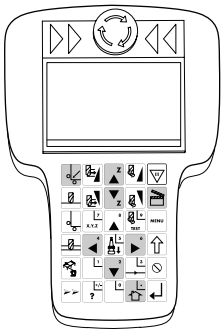
Mill your Project

Control Hub

Pendant Keypad

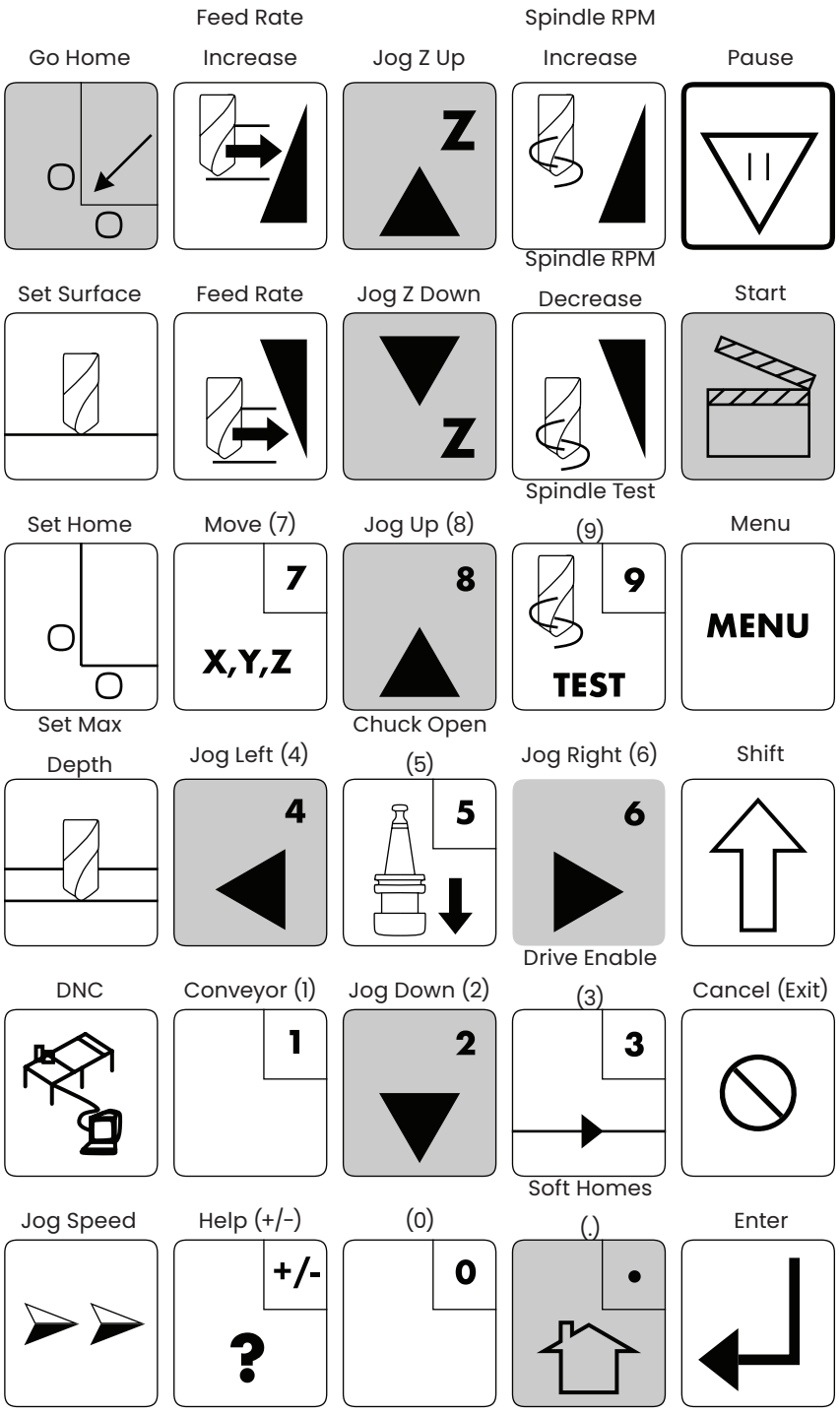
Milling procedure

Troubleshooting



Pendant

SAFETY NOTE: The CNC is equipped with a safety hold light curtain. It pauses all machine operations when a person or object crosses the marked safety border. If you must cross the border while the spindle is off, be sure to press the green reset button on the light curtain control box to proceed.



Step 3:

Mill your Project

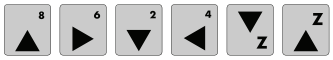
Control Hub

Pendant Keypad

Milling procedure

Troubleshooting

Jog means to move the spindle along the x,y,z axis. The jog keys are as follows:



Jog speed  can be adjusted to slow, medium, and fast.

The following outlines the general procedure for setting up and running the Multicam 5000 Series CNC Router. This process will always be performed with a technician.

1. **Turn on the CNC.** Locate the main power switch on the CNC power control box. The machine will automatically run a start up sequence and then display this message at the bottom of the screen: "*MACHINE NOT HOMED!"*

2. **Commence the homing sequence** by pressing  +  This allows the CNC to locate a hard home: the origin point in the x,y,z axis as well as on the tool carousel and knife cutting tools. This home position will always be set as home 1 and can be accessed with  +  Refer to chapter *09: Coordinates and the CNC Router f*or further details.

3. **The machine will then prompt a SPINDLE WARM UP.** A spindle warm up is recommended if you are milling multiple sheets of material for the entire day. If you choose to run a spindle warm up, JOG the spindle to a safe distance away from the control hub. Press  to start the warm up. This process will run the spindle at multiple speeds and should take approximately 15 minutes.

4. **Position your stock onto the CNC bed.** Align its long edge with the long spoil board edge closest to the control hub. Secure it with tape or screws if necessary. Refer to chapter *06 & 07 f*or further details.

5. **Turn on the green vacuum hold-down switch.** Depending on the size of your stock material, you may turn on or off vacuum hold-down zones 1-4. This step is necessary to ensure accurate measurements in the following steps.

6. **Calibrate the tools** that you have specified

Step 3:

Mill your Project

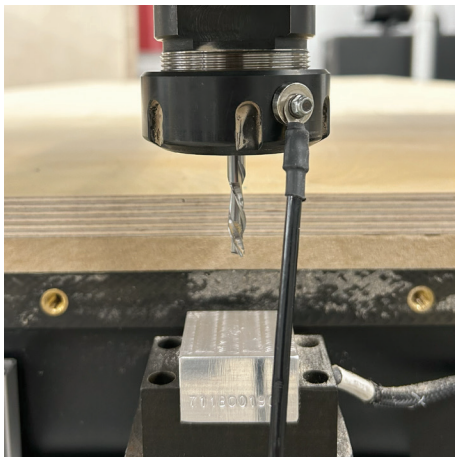


fig. 1

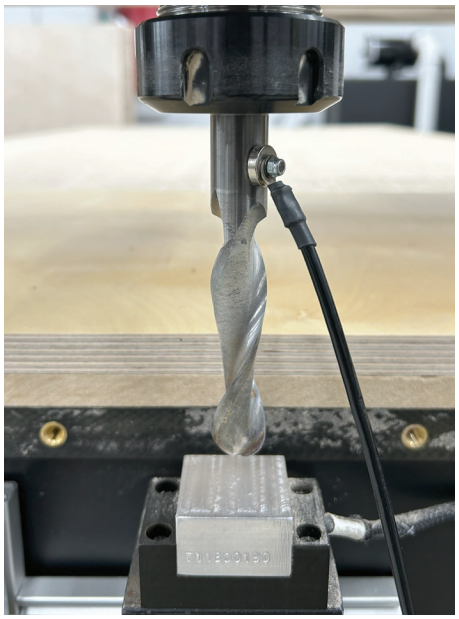


fig. 2

for your project. Open ATC in the menu [MENU] to access the tool calibration function. The spindle will automatically jog to the tool calibration plate. At this point, be sure to mount the grounding clip to the CHUCK (fig. 1). If the tool bit diameter is 1/2" or larger, mount the clip DIRECTLY ONTO THE TOOL (fig. 2). Slowly jog the spindle in the Z-axis until the tool is 1" away from the plate. Be sure NOT to hit the tool calibration plate. At this point, lower the spindle further at smaller increments by TAPPING the zero [0] key until the tool tip touches the plate. The spindle will automatically retract when the tool is calibrated.

7. **Set the TOP SURFACE** your stock with the surface key. While the grounding clip is attached to the tool or chuck, jog the spindle to any location above your STOCK MATERIAL. Place the touch pad directly below the tool and lower the spindle in the same manner as calibrating the tool (fig. 3). Once the spindle retracts, the CNC will recognize where the top surface of your stock material is on the Z-axis.

8. **Set the MAX DEPTH** with the max depth key. Position the touch pad on top of the SPOIL BOARD (fig. 4) and repeat the process outlined in the previous step. Once the spindle retracts, the CNC will recognize the exact depth of your stock material. To check the max depth value, open: *MENU > Params_2D > Max Depth.* This value should match the stock material thickness you measured with calipers. Hold [⇧] + [surface] and enter 0.03 to add 0.03" to the max depth. This ensures that the tool passes through the stock completely.

9. **Disconnect the grounding clip and remove the touch pad from the CNC bed** and move the spindle to the home position [home] + [1]

10. **Locate your g-code file under the DNC jobs menu.** Press start to preview your file on

Step 3:

Mill your Project

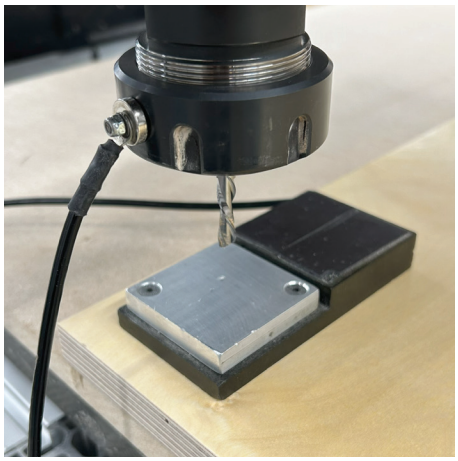


fig. 3

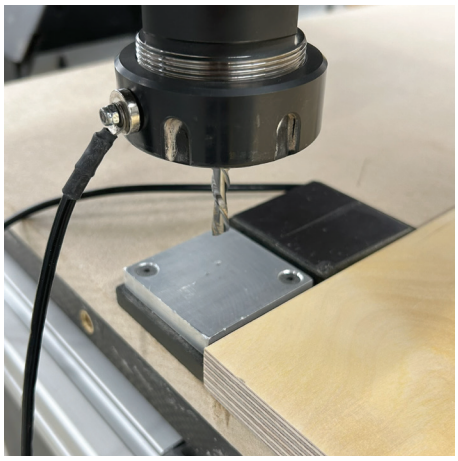


fig. 4

the controller display. Press ↵ to continue.

11. Perform a dry run. Scroll down ▼² and press enter ↵ . The spindle will follow the XY toolpath of your file without plunging into your stock. You may let the CNC complete the dry run or press pause to cancel.

12. **Turn on the dust extraction system.**

13. **Commence your CNC job.** Repeat Step 10 and press start 🎬 . Be ready to pause the operation should an error occur.

14. **Once your job is complete** wait for the tool to be returned, jog the gantry to the end of the machine bed, turn off the vacuum hold-down and remove your material from the CNC bed. Clear the bed, rails, spindle, tool carousel and the area around the CNC of any dust and chips.

15. **Shut down the CNC** unless you plan to continue milling the next day or otherwise directed by a technician. Scroll through the menu and locate the shutdown function. Wait for the CNC to complete a shutdown sequence before power down. Once prompted *"ok to power down*" you may now shut off the main power switch.

SAFETY REMINDERS

DO NOT leave the CNC unattended! If the bit breaks, the spindle plunges too deep, or an error occurs while the CNC is running press PAUSE ▽ and notify the technician. Refrain from using the emergency stop button unless it is necessary.

Step 3:

Mill your Project

Always account for errors and malfunctions when planning time and materials for your CNC appointment. The technician will be available to **troubleshoot** any problems should they interrupt the process.

 A **network disconnection** will be indicated by the network icon at the bottom of the CNC controller screen. This means that the CNC has lost connection with the host computer. To fix the connection, simply open *Coreo Command* application on the host computer. To review connection settings and status, open *Coreo Connect. Coreo* is a built-in CAM software for Multicam CNC routers.

When a **tool bit breaks**, pause  and cancel the milling operation and NOTIFY a technician. IMPORTANT: If the fracture caused sparks, TURN OFF the dust extractor and vacuum hold-down IMMEDIATELY! If not, leave the vacuum hold-down ON and follow the procedure below:

1. **Mark the spot where the tool bit broke** with tape or a marker BEFORE jogging the spindle.
2. **Carefully retrieve and dispose** of the broken tool bit. Be sure that the CNC bed is clear of any pieces of the tool.
3. **Jog the spindle** to an area above the CNC bed that you can reach with your arm if it has paused in an unaccessible spot. If the spindle has paused at an area you can reach, DO NOT jog the spindle.
4. **Open the chuck**  +  to unload the tool manually. Follow the on-screen prompts WHILE HOLDING the chuck with one hand. A pop-up warning will appear before you open the chuck. The spindle will release the chuck as you hold it steady. **Close the chuck** as soon as it releases.
5. **Replace the tool bit.** Refer to chapter *05: Removing Router bits.*

Step 3:

Mill your Project

6. **Re-install the chuck onto the spindle.** Open the chuck, place the chuck into the spindle, and close the chuck.
7. **Calibrate the new tool.** Update the corresponding tool settings on RhinoCAM.

The **proximity restart** function enables the CNC to continue milling right after replacing a broken tool bit without losing your progress and accuracy. To perform a proximity restart follow the procedure below:

1. Jog the spindle as close to the stopping point as possible.
2. Use the Shift  and Start  hotkeys on the control panel to access the proximity restart function.
3. Press the Enter  hotkey. The CNC Router Machine will look for a new starting point within a radius of 0.3” of the stopping point.
4. Press the Start  hotkey to restart the machining file from the new starting point.

12:

The Knife Cutting / Creasing Wheel Process

There are two pneumatically driven cutting and creasing attachments mounted to the left of the spindle. Each of these heads function independently with their own set of cutting or creasing capabilities. There are a number of knives and wheels that can be mounted on either of these heads. Please consult a technologist before switching out or loading a tool onto either attachment.

1. **Tool 31** is the **oscillating knife cutter**. It uses air to oscillate the knife at very high speeds to perform highly accurate curved cuts. It can hold knives with a length of 15-50mm meaning it can cut through a variety of materials including, but not limited to:

- Paper
- Styrene with a thickness of <1mm
- Corrugated cardboard
- Honeycomb Cardboard with a thickness <50mm
- Felt
- Rubber mat
- Foam mat with a thickness <50mm

2. **Tool 41** is the **interchangable drag knife or creasing wheel attachment**.

- Drag knives allow for a high degree of repeat-ability with straight cuts. It can perform 90 or 45 degree cuts using a standard box cutting blade.
- Creasing wheels allow folds to be rolled into the material

Step 1:

Create your CAD File

Similar to milling operations, the knife cutting and creasing attachments require control curves. Modelling your stock material is optional; however, it can serve as a reference.

1. **Draw or import** a vector drawing into Rhino.
2. **Organize** your file into separate layers. Make sure that curves that represent cut lines are on a separate layer from any curves that represent creasing lines.

Step 2:

Generate Toolpaths in CAM

Knife cutting and creasing wheel operations require control geometries, tool number inputs, cut depth, and feeds & speeds.

1. **Open** *Program > Machine Operations > 2 axis > Knife Cutting* to access the parameters. Note that creasing wheel toolpaths will be generated with these parameters as well.
2. Under **Control Geometries** select the curves that represent your cut lines or creases.
3. Under **Tool** select the appropriate cutting or creasing tool.

Tool 31 = oscillating knife
Tool 41 = creasing wheel / drag knives

4. Under **Feeds & Speeds** set your desired rates. These can vary depending on material.
5. Under **Cut Parameters** set the *tolerance to 0.001* and *cut depth* to a value slightly larger than the material thickness.
6. **Simulate** your setup to make sure that the tool follows your desired curves
7. **Post** your gcode to the CNC

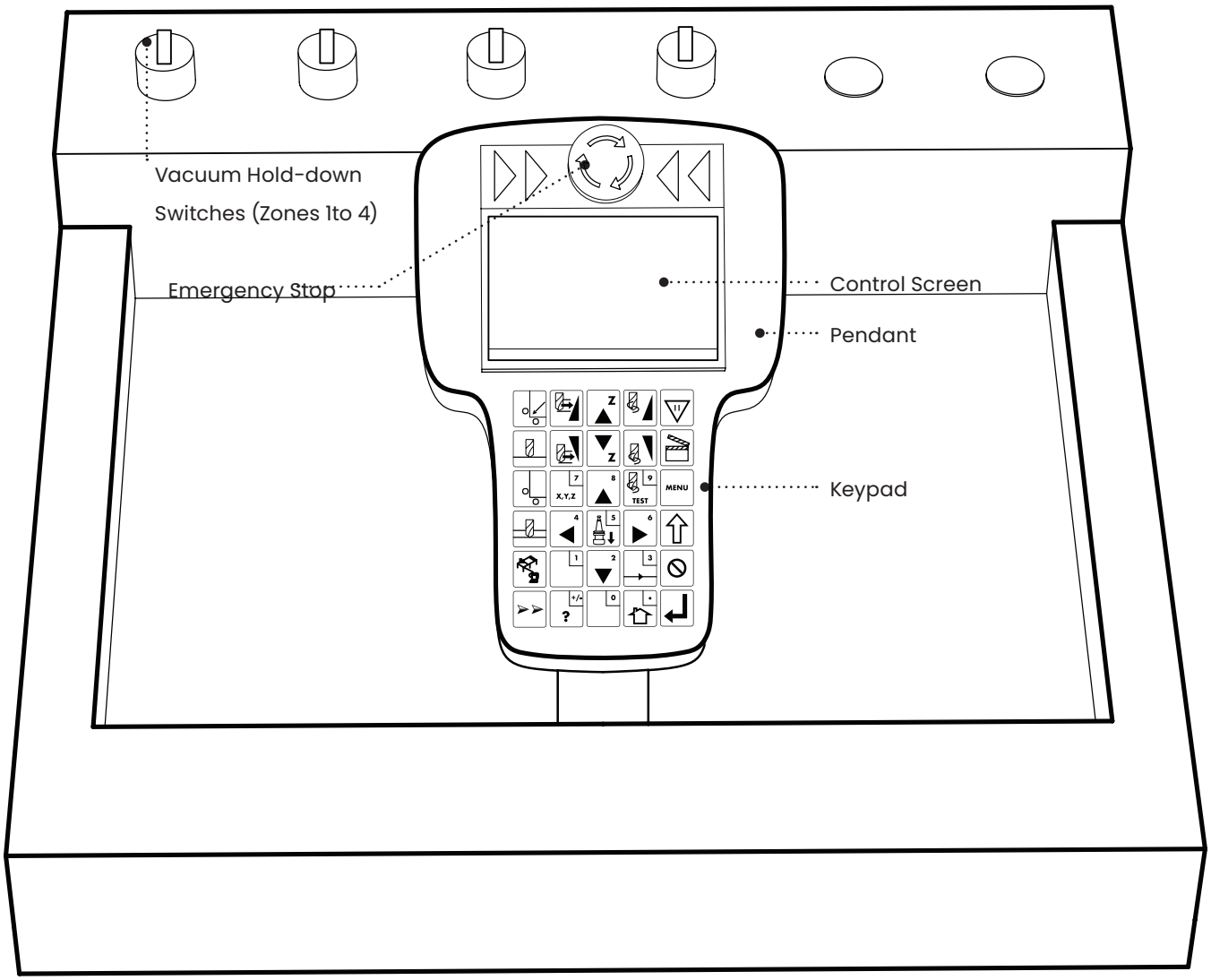
Step 3:

Cut / Crease your Project

Control Hub

Pendant Keypad

Cut / Crease procedure



Multicam 5000 Series CNC Router Control Hub

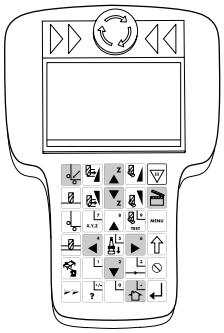
Step 3:

Cut / Crease your Project

Control Hub

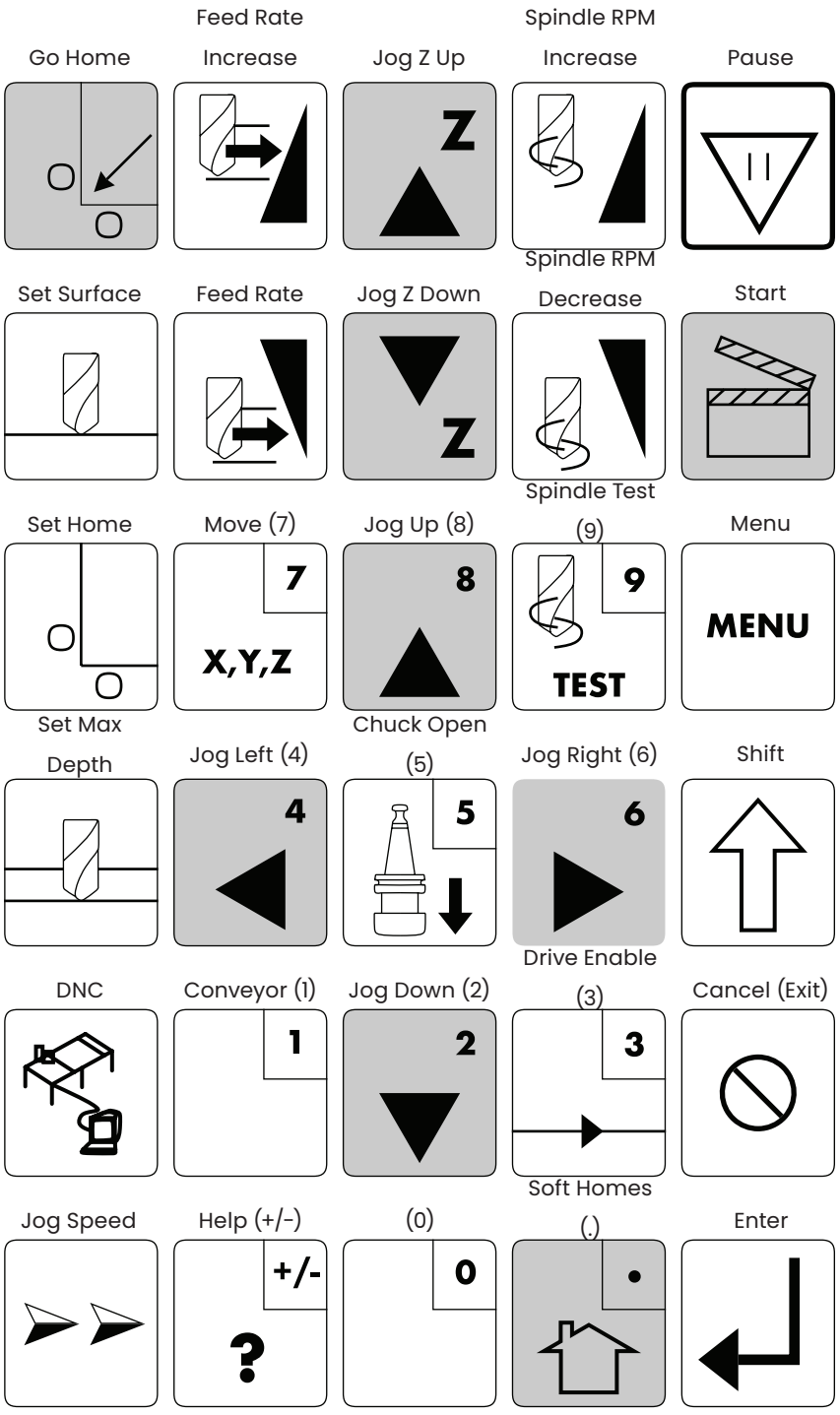
Pendant Keypad

Cut / Crease procedure



Pendant

SAFETY NOTE: The CNC is equipped with a safety hold light curtain. It pauses all machine operations when a person or object crosses the marked safety border. If you must cross the border while the spindle is off, be sure to press the green reset button on the light curtain control box to proceed.



Step 3:

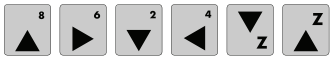
Cut / Crease your Project

Control Hub

Pendant Keypad

Cut / Crease procedure

Jog means to move the spindle along the x,y,z axis. The jog keys are as follows:



Jog speed  can be adjusted to slow, medium, and fast.

The following outlines the general procedure for setting up and running the Multicam Knife Cutting and Creasing wheel cartrige attachments. This process will always be performed with a technician.

1. **Turn on the CNC.** Locate the main power switch on the CNC power control box. The machine will automatically run a start up sequence and then display this message at the bottom of the screen: "*MACHINE NOT HOMED!"*

2. **Commence the homing sequence** by pressing  +  This allows the CNC to locate a hard home: the origin point in the x,y,z axis as well as on the tool carousel and dust collector foot. This home position will always be set as home 1 and can be accessed with  +  Refer to chapter *09: Coordinates and the CNC Router f*or further details.

3. **The machine will then prompt a SPINDLE WARM UP**. Press cancel to skip this step as the spindle will not be active for this procedure.

4. **Place the felt cutting mat on top of the CNC spoil board.** DO NOT operate the knife cutters or creasing wheels without it.

5. **Position your stock material anywhere on the mat** as long as it runs relatively square to the x and y axis of the cnc. Use tape to create a seal along the edges of your workpiece.

6. **Turn on the green vacuum hold-down switch.** Depending on the size of your workpiece, you may turn on or off certain vacuum hold-down zones. This step is necessary to ensure accurate measurements in the following steps.

7. **Install a knife blade or creasing wheel** into their proper adapter.

8. **Calibrate the tools** before starting a new job or when a new blade or wheel is installed. DO

Step 3:

Cut / Crease your Project

Control Hub

Pendant Keypad

Cut / Crease procedure

NOT use the grounding magnet to calibrate knife blades or creasing wheels. Open ATC in the menu 

to access the tool calibration function. At this point, lower the spindle further at smaller increments by TAPPING the zero 

key until the tool tip touches the plate. The spindle will automatically retract when the tool is calibrated.

9. **Set the TOP SURFACE** your stock with the surface key  . Jog the spindle to any location above your stock material. Place the touch pad directly below the tool and lower the spindle in the same manner as calibrating the tool. Once the spindle retracts, the CNC will recognize where the top surface of your stock material is on the Z-axis.

10. **Set CUT DEPTH** with the max depth under *MENU > UTILITY > ADJUST SURFACE*. Lower the surface in increments of one thousandths of an inch. For example, input 0.002" to lower the surface or -0.002" to raise it. Be sure to test these cutting depths before proceeding to your final cuts.

11. **Set the OSCILLATING STROKE** only when using the oscillating knife. Follow *MENU > KNIFE UTILITY > OSC. STROKE > ENTER > TOOL # OSC ON*. Place a sheet of standard weight paper on the bed spoil board and lower the tool (Jog Speed = MED.) until ½" above carpet surface. Switch the Jog Speed to SLOW and carefully lower the tool until you feel the paper vibrate and the knife begins to scratch the paper. The pendant will now prompt you to repeat this process with the oscillating stroke OFF. Oscillation Stroke value should range between 0.22 and 0.30.

12. **Set a SOFT HOME** at the corner of your material closest to the pendant. Follow *MENU > KNIFE > SET HOME WITH KNIFE*. Always set the soft home with Tool 31 or Tool 41.

13. **Home Knife** to reset the orientation of the knife

Step 3:

Cut / Crease your Project

Control Hub

Pendant Keypad

Cut / Crease procedure

or creasing wheel tool.

14. **Set the Knife Control to job** by navigating MENU > KNIFE > KNIFE CONTROL > JOB to ensure the cuts match your toolpaths

15. **Locate your g-code** file under the DNC jobs menu. 

16. **Perform a dry run.** This step is optional but recommended. Simply start  your job WITHOUT your material on the cutting mat. Observe the feedrate of the knife, its retracting movements, and the depth of cut. Make adjustments as necessary.

17. **Commence your file.** Access your file in the DNC jobs menu and press start  . Be ready to pause the operation should an error occur.

18. **DO NOT leave the knife exposed on the cutting head.** Remove the blade from the cartridge before shutting down the machine.

19. **Remove your material from the CNC bed.** Jog the gantry out of the way and clean the bed, rails, spindle, tool carousel and the area around the CNC of any dust and chips. Be mindful of the knife cutting blade.

20. **Shut down the CNC** unless you plan to continue milling the next day or otherwise directed by a technician. Scroll through MENU and locate the shutdown function. Wait for the CNC to complete a shutdown sequence before power down.

SAFETY REMINDERS

DO NOT leave the CNC unattended! If the bit breaks, the spindle plunges too deep, or an error occurs while the CNC is running press PAUSE  and notify the technician. Refrain from using the emergency stop button unless it is necessary.

13: Multicam Vision System

The Multicam 5000 CNC is equipped with a visual fiducial registration system that allows the machine to locate an image relative to your workpiece before cutting it out. It consists of a camera mounted on *cartridge 41* that connects directly to *Coreo Command* on the CNC Computer console as well as the pendant.

This system can be used in conjunction with the spindle, oscillating knife cartridge, and the creasing wheel cartridge.

Using the Multicam Vision System is a three-step process that can be integrated into milling, knife cutting, and creasing processes:

1. Print file preparation
2. Machine file preparation
3. Machine operation

Step 1:

Print File Preparation

Adobe Illustrator and **Rhino** are the primary softwares used for vision system applications at the Design + Technology Lab. You will be responsible for creating a graphic that follows the format described in this chapter. Inadequate files will not be processed at your scheduled appointment.

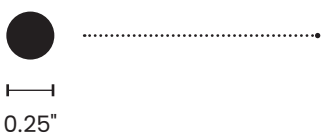
You will create two corresponding files from this drawing: a print file and a machine file. These two files MUST match each other as accurately as possible to ensure the best results.

PRINT FILE: This file will be printed on your workpiece material. It can be printed on a number of different materials including paper, cardboard, plywood, mdf, acrylic, polycarbonate, etc. Large format prints will be processed on the Ricoh printer at the Design + Technology Lab.

MACHINE FILE: This file will be used to generate toolpaths and coordinates for the camera to locate the fiducials.

Create an Adobe Illustrator document that contains three layers:

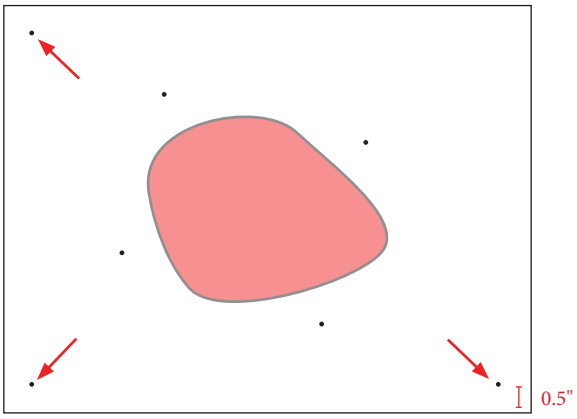
1. A layer for your graphic.
2. A layer for a vector cutline that follows the perimeter of your graphic. To cut on the line, position the cutline ON the outline of your graphic. You may offset the cutline outside of your graphic to create a border. You may also offset the cutline within the bleed line to create a borderless graphic
3. Fiducial points (also known as registration points): these are black dots with a diameter of 0.25”. There are two types of fiducial points that must appear on your file: global and local fiducials.



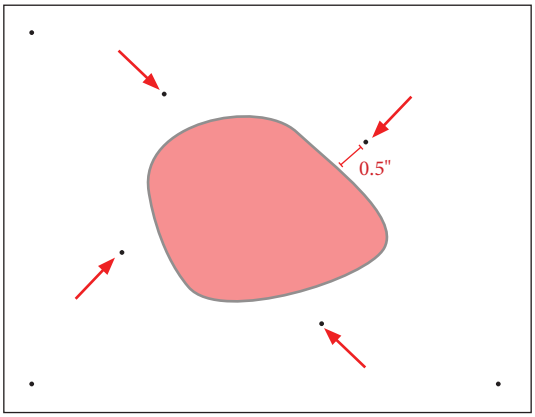
Step 1:

Print File Preparation

Global fiducials: three points that appear on your workpiece that will determine the orientation of your workpiece on the bed of the machine. Be sure to position them within a reasonable margin away from the edge of the workpiece (min1/2").



Local fiducials: points that appear around your graphic that will determine the skew and orientation of your graphic. There must be a minimum of three fiducials that are positioned around your graphic. Locate them at least ½" away from the cutline and a minimum of 1" away from neighboring fiducials.



Once your file has all the data mentioned above, save and export two files:

1. A .pdf file to be sent to the printer. Large files will be sent to the Ricoh large format printer at the Design + Technology Lab.
2. A .dwg or .dxf file to be imported into Rhino. Be sure to select all (ctrl+a) the components of your file before exporting.

Step 2:

Machine File Preparation

Open Rhino to generate toolpaths from your graphic file

Import your file into Rhino. Your graphic, cutline, and fiducials should appear.

Organize your graphic, cutline, and fiducials into separate layers. This will help especially when there are multiple graphics that are nested into your workpiece. Note: if your fiducials import without a vector outline, you may need to extract them manually. Use the SelHatch command > explode into surfaces > dupedge and select the fiducials to extract from

Open the RhinoCAM Plugin to begin programming toolpaths .

1. The FIRST operation to program in your *setup* are the fiducials. The vision system operates by finding the centres of the fiducials on your workpiece and comparing their location to the coordinates on your file. Use the *Drill* operation to program the fiducials. Follow: *Program > Machine Operations > Holes > Drill*
2. Under **Control Geometries** select the curves that represent fiducials
3. Under **Tool** select *Camera* (Tool 99)
4. Under **Feeds & Speeds**, make sure that the spindle speed is set to 0
5. Click generate
6. Program your milling, knife cutting, or creasing wheel operations (refer to chapters 11 & 12)
7. Simulate your setup. Set the simulation speed to slow and check if the Camera tool has jogged to each fiducial
8. Post the gcode to the CNC

Step 3:

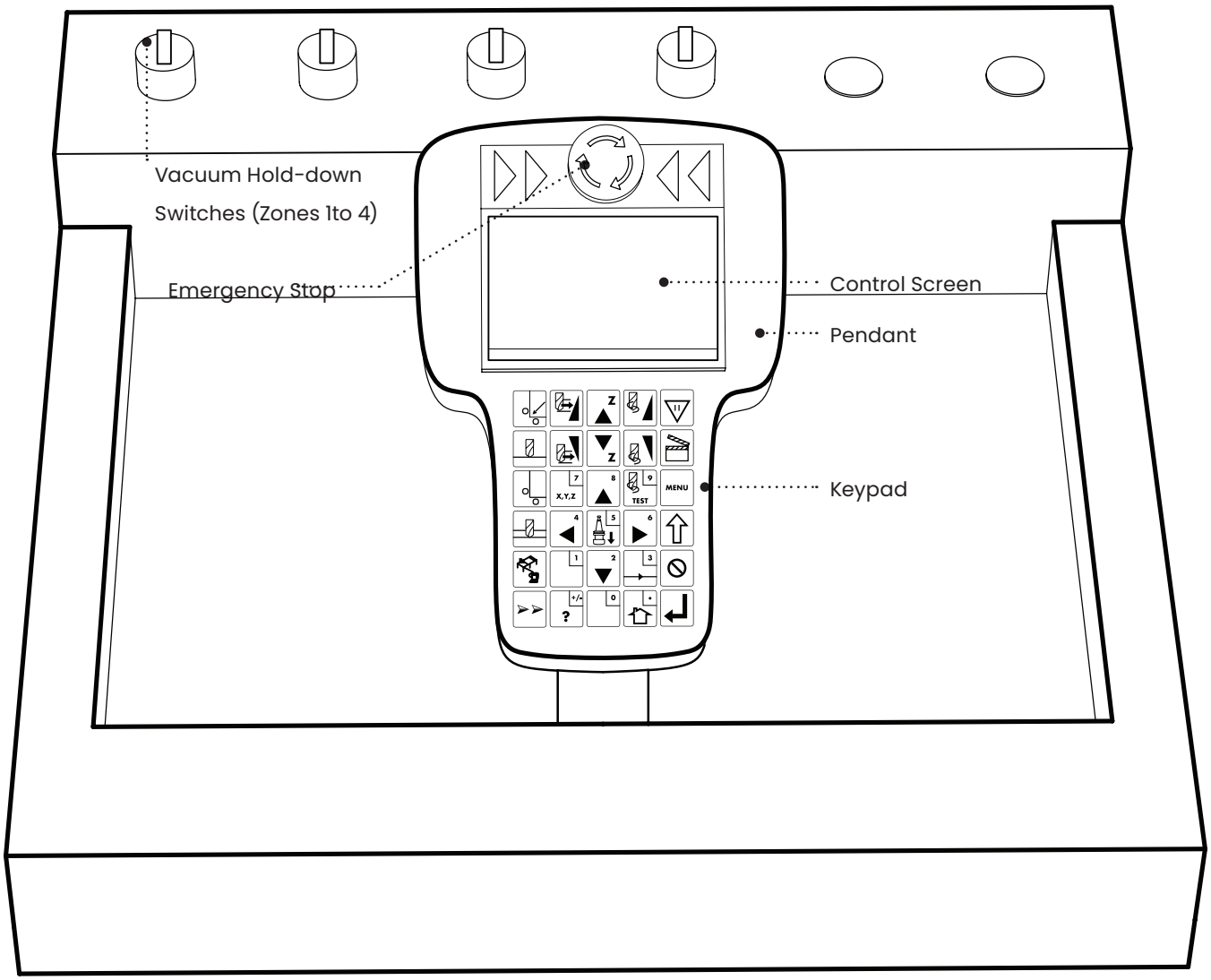
Machine Operation

Control Hub

Pendant Keypad

Vision System + Milling

Vision System + Knife cutter
/ creasing wheel



Multicam 5000 Series CNC Router Control Hub

Step 3:

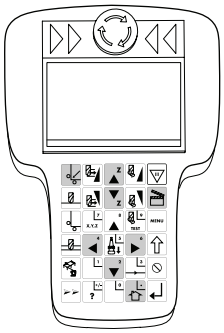
Machine Operation

Control Hub

Pendant Keypad

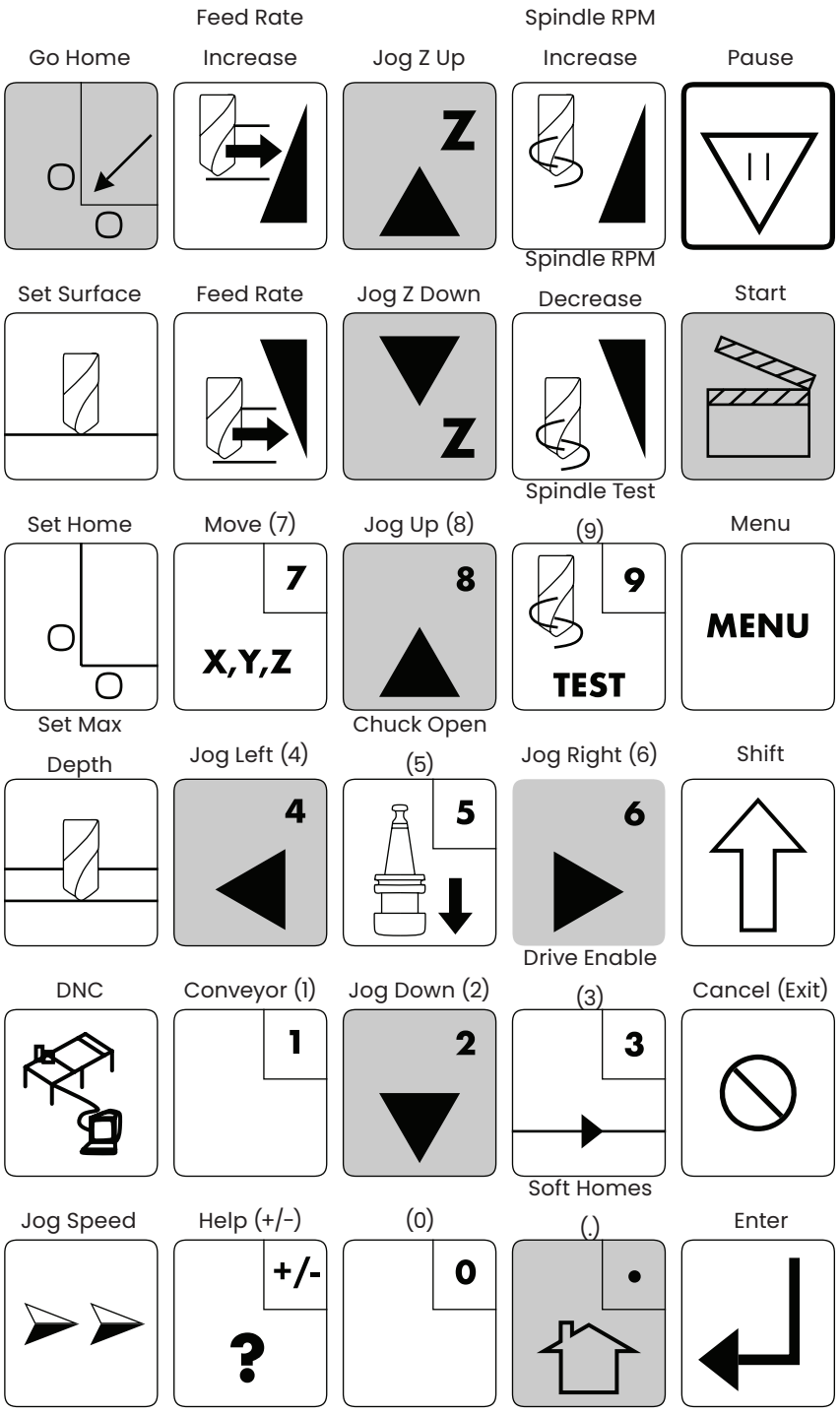
Vision System + Milling

Vision System + Knife cutter / creasing wheel



Pendant

SAFETY NOTE: The CNC is equipped with a safety hold light curtain. It pauses all machine operations when a person or object crosses the marked safety border. If you must cross the border while the spindle is off, be sure to press the green reset button on the light curtain control box to proceed.



Step 3:

Machine Operation

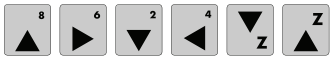
Control Hub

Pendant Keypad

Vision System + Milling

Vision System + Knife cutter / creasing wheel

Jog means to move the spindle along the x,y,z axis. The jog keys are as follows:



Jog speed  can be adjusted to slow, medium, and fast.

The following outlines the general procedure for setting up and running the Multicam Vision System for MILLING operations. This process will always be performed with a technician.

1. **Turn on the CNC.** Locate the main power switch on the CNC power control box. The machine will automatically run a start up sequence and then display this message at the bottom of the screen: "*MACHINE NOT HOMED!"*

2. **Commence the homing sequence** by pressing  +  This allows the CNC to locate a hard home: the origin point in the x,y,z axis as well as on the tool carousel and dust collector foot. This home position will always be set as home 1 and can be accessed with  +  Refer to chapter *09: Coordinates and the CNC Router f*or further details.

3. **The machine will then prompt a SPINDLE WARM UP.** A spindle warm up is recommended if you are milling multiple sheets of material for the entire day. If you choose to run a spindle warm up, JOG the spindle to a safe distance away from the control hub. Press  to start the warm up. This process will run the spindle at multiple speeds and should take approximately 15 minutes.

4. **Position your stock onto the CNC bed.** Align its long edge with the long spoil board edge closest to the control hub. Secure it with tape or screws if necessary. Refer to chapter *06 & 07 f*or further details.

5. **Turn on the green vacuum hold-down switch.** Depending on the size of your stock material, you may turn on or off certain vacuum hold-down zones. This step is necessary to ensure accurate measurements in the following steps.

6. **Calibrate the tools** that you have specified

Step 3:

Machine Operation

Control Hub

Pendant Keypad

Vision System + Milling

Vision System + Knife cutter / creasing wheel

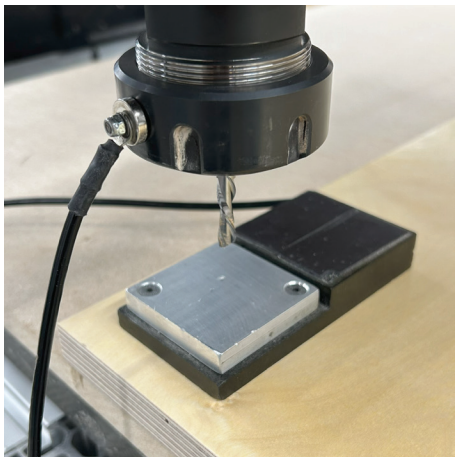


fig. 3

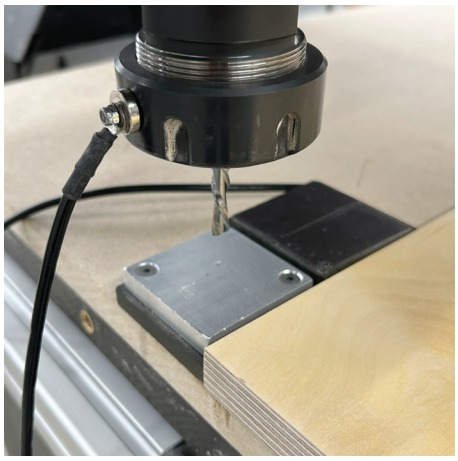


fig. 4

for your project. Open ATC in the menu MENU to access the tool calibration function. The spindle will automatically jog to the tool calibration plate. At this point, be sure to mount the grounding clip to the chuck (fig. 1). If the tool bit diameter is 1/2" or larger, mount the clip DIRECTLY ONTO THE TOOL (fig. 2). Slowly jog the spindle in the Z-axis until the tool is 1" away from the plate. Be sure NOT to hit the tool calibration plate. At this point, lower the spindle further at smaller increments by TAPPING the zero key until the tool tip touches the plate. The spindle will automatically retract when the tool is calibrated.

7. **Set the TOP SURFACE your stock** with the surface key. While the grounding clip is attached to the tool or chuck, jog the spindle to any location above your STOCK MATERIAL. Place the touch pad directly below the tool and lower the spindle in the same manner as calibrating the tool (fig. 3). Once the spindle retracts, the CNC will recognize where the top surface of your stock material is on the Z-axis.

8. **Set the MAX DEPTH** with the max depth key. Position the touch pad on top of the spoil board and repeat the process outlined in the previous step. Once the spindle retracts, the CNC will recognize the exact depth of your stock material. To check the max depth value, open: *MENU > PARAMS_2D > MAX DEPTH.* This value should match the stock material thickness you measured with calipers. Hold + and enter 0.03 to add 0.03" to the max depth. This ensures that the tool passes through the stock completely.

9. **Disconnect** the grounding clip and remove the touch pad from the CNC bed and move the spindle to the home position +

10. **Move the CNC computer cabinet** to a position where you can see the tool and the screen at the same time. Be sure that it is not in the way of the gantry. Open *Coreo Command*

Step 3:

Machine Operation

Control Hub

Pendant Keypad

Vision System + Milling

Vision System + Knife cutter / creasing wheel

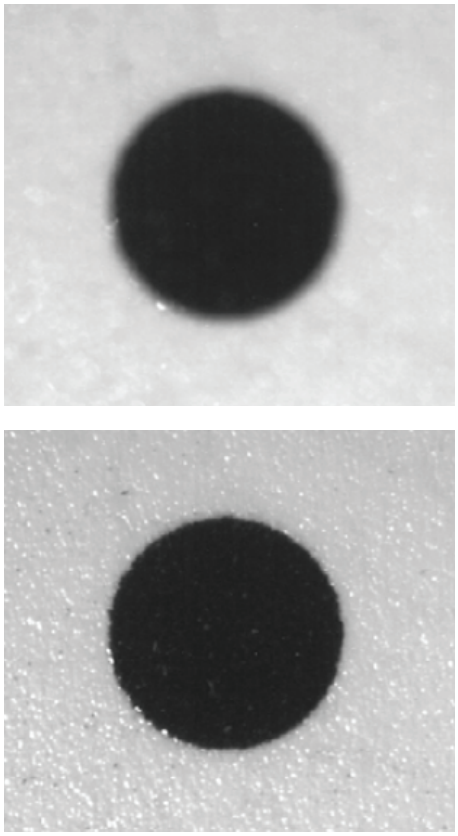


Fig. 1 Fiducial Focusing

source: Multicam Router User Manual v. 10 14 (Manufacturers manual)

and leave it visible on the screen.

11. **Focus the camera** before starting a new job and when working with another material with a different thickness. This should be done after setting the top surface and max depth of the material. Open *MENU > VISION > FOCUS CAMERA.* This will prompt you to remove the tool. Press enter to unload the tool. Make sure that the vacuum hold down is on to ensure an accurate reading.

12. **Jog the camera** to any one of the fiducials on your workpiece. It should be highlighted under the green light and appear in the centre of the *live view* window.

13. **Within the VISION tab in Coreo Command** click the icon to access a live view of the camera. Set the Jog speed to medium and jog the camera up and down with the Z-axis jog arrows until the fiducial appears as a sharp image. Refer to *Fig. 1 Fiducial Focusing.*

14. **Press enter when the fiducial is in focus.** The Vision System will scan the fiducial to determine its scale. The reading should result in a fiducial size of about 0.25". Press enter to accept.

15. **Access your job in the DNC menu.**

16. **Perform a dry run.** Scroll down and press enter . You may let the CNC complete the dry run or press pause to cancel. Make sure that the camera scans the right fiducials and the spindle follows your cutlines.

17. **Turn on the dust extraction system.**

18. **Commence your file.** Access your file in the DNC Menu and press . Watch the machine carefully and be ready to pause should an error occur.

19. **Remove your material from the CNC bed.** Jog the gantry out of the way and clean the bed,

Step 3:

Machine Operation

Control Hub

Pendant Keypad

Vision System + Milling

Vision System + Knife cutter / creasing wheel

rails, spindle, tool carousel and the area around the CNC of any dust and chips.

20. **Shut down the CNC** unless you plan to continue milling the next day or otherwise directed by a technician. Locate the *Shutdown* function under MENU. Wait for the CNC to complete a shutdown sequence before powering down.

SAFETY REMINDERS

DO NOT leave the CNC unattended! If the bit breaks, the spindle plunges too deep, or an error occurs while the CNC is running press PAUSE  and notify the technician. Refrain from using the emergency stop button unless it is necessary.

Step 3:

Machine Operation

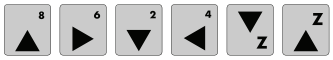
Control Hub

Pendant Keypad

Vision System + Milling

Vision System + Knife cutter / creasing wheel

Jog means to move the spindle along the x,y,z axis. The jog keys are as follows:



Jog speed can be adjusted to slow, medium, and fast.

The following outlines the general procedure for setting up and running the Multicam Knife Cutting and Creasing wheel cartrige attachments. This process will always be performed with a technician.

1. **Turn on the CNC.** Locate the main power switch on the CNC power control box. The machine will automatically run a start up sequence and then display this message at the bottom of the screen: "*MACHINE NOT HOMED!"*

2. **Commence the homing sequence** by pressing + This allows the CNC to locate a hard home: the origin point in the x,y,z axis as well as on the tool carousel and dust collector foot. This home position will always be set as home 1 and can be accessed with + Refer to chapter *09: Coordinates and the CNC Router f*or further details.

3. **The machine will then prompt a SPINDLE WARM UP.** Press cancel to skip this step as the spindle will not be active for this procedure.

4. **Place the felt cutting mat on top of the CNC spoil board.** DO NOT operate the knife cutters or creasing wheels without it.

5. **Position your stock material** anywhere on the mat as long as it runs relatively square to the x and y axis of the cnc. Use tape to create a seal along the edges of your workpiece.

6. **Turn on the green vacuum hold-down switch.** Depending on the size of your workpiece, you may turn on or off certain vacuum hold-down zones. This step is necessary to ensure accurate measurements in the following steps.

7. **Calibrate the tools** before starting a new job or when a new blade or wheel is installed. DO NOT use the grounding magnet to calibrate knife blades or creasing wheels. Open ATC in the menu MENU to access the tool calibration

Step 3:

Machine Operation

Control Hub

Pendant Keypad

Vision System + Milling

Vision System + Knife cutter / creasing wheel

function. At this point, lower the spindle further at smaller increments by TAPPING the zero key until the tool tip touches the plate. The spindle will automatically retract when the tool is calibrated.

8. **Set the TOP SURFACE** your stock with the surface key. Jog the spindle to any location above your stock material. Place the touch pad directly below the tool and lower the spindle in the same manner as calibrating the tool. Once the spindle retracts, the CNC will recognize where the top surface of your stock material is on the Z-axis.

9. **Set CUT DEPTH** with the max depth under *MENU > UTILITY > ADJUST SURFACE*. Lower the surface in increments of one thousandths of an inch. For example, input 0.002" to lower the surface or -0.002" to raise it. Be sure to test these cutting depths before proceeding to your final cuts.

10. **Set the OSCILLATING STROKE only when using the oscillating knife.** Follow *MENU > KNIFE UTILITY > OSC. STROKE > ENTER > TOOL # OSC ON*. Place a sheet of standard weight paper on the bed spoil board and lower the tool (Jog Speed = MED.) until ½" above carpet surface. Switch the Jog Speed to SLOW and carefully lower the tool until you feel the paper vibrate and the knife begins to scratch the paper. The pendant will now prompt you to repeat this process with the oscillating stroke OFF. Oscillation Stroke value should range between 0.22 and 0.30.

11. **Set a SOFT HOME** at any global fiducial. Follow *MENU > KNIFE > SET HOME WITH KNIFE.* Always set the soft home with Tool 31 or Tool 1 even if you need to use Tool 41.

12. **Move the CNC computer cabinet** to a position where you can see the tool and the screen at the same time. Be sure that it is not in

Step 3:

Machine Operation

Control Hub

Pendant Keypad

Vision System + Milling

Vision System + Knife cutter / creasing wheel

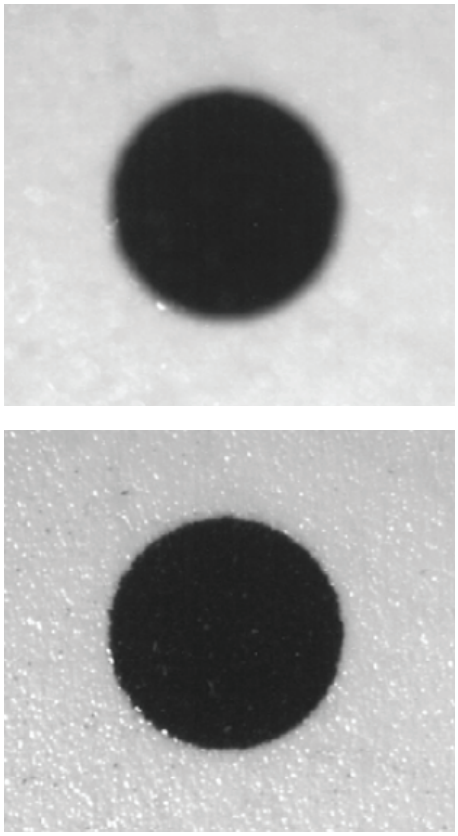


Fig. 1 Fiducial Focusing

source: Multicam Router User Manual v. 10 14 (Manufacturers manual)

the way of the gantry. Open *Coreo Command* and leave it visible on the screen.

13. **Focus the camera** before starting a new job and when working with another material with a different thickness. This should be done after setting the top surface and max depth of the material. Open *MENU > VISION > FOCUS CAMERA.* This will prompt you to remove the tool. Press enter to unload the tool. Make sure that the vacuum hold down is on to ensure an accurate reading.

14. **Jog the camera** to any one of the fiducials on your workpiece. It should be highlighted under the green light and appear in the centre of the *live view* window.

15. **Within the VISION tab in Coreo Command** click the  to access a live view of the camera. Set the Jog speed =MED and jog the camera up and down with the Z-axis jog arrows until the fiducial appears as a sharp image. Refer to *Fig 1. Fiducial Focusing.*

16. **Press enter when the fiducial is in focus.** The Vision System will scan the fiducial to determine its scale. The reading should result in a fiducial size of about 0.25". Press enter to accept.

17. **Perform a dry run.** S This step is optional but recommended. Simply start  your job WITHOUT your material on the cutting mat. Observe the feedrate of the knife, its retracting movements, and the depth of cut. Make adjustments as necessary.

18. **Set the Knife Control to job** by navigating MENU > KNIFE > KNIFE CONTROL > JOB to ensure the cuts match your toolpaths

19. **Locate your g-code file** under the DNC jobs menu. 

20. **Commence your file.** Access your file in the

Step 3:

Machine Operation

Control Hub

Pendant Keypad

Vision System + Milling

Vision System + Knife cutter / creasing wheel

DNC jobs menu and press start  . Be ready to pause  the operation should an error occur.

21. **DO NOT leave the knife exposed on the cutting head.** Cover the blade with the foam shoe while the machine is paused. Remove the blade from the cartridge before shutting down the machine.

22. **Remove your material from the CNC bed.** Jog the gantry out of the way and clean the bed, rails, spindle, tool carousel and the area around the CNC of any dust and chips.

23. **Shut down the CNC** unless you plan to continue milling the next day or otherwise directed by a technician. Scroll through MENU and locate the shutdown function. Wait for the CNC to complete a shutdown sequence before power down.

SAFETY REMINDERS

DO NOT leave the CNC unattended! If the bit breaks, the spindle plunges too deep, or an error occurs while the CNC is running press PAUSE  and notify the technician. Refrain from using the emergency stop button unless it is necessary.

Multicam 5000 CNC Router
Manual

The
Creative
School
Design +
Technology LAB

Multicam 5000 CNC Router
Manual

The
Creative
School
Design +
Technology LAB