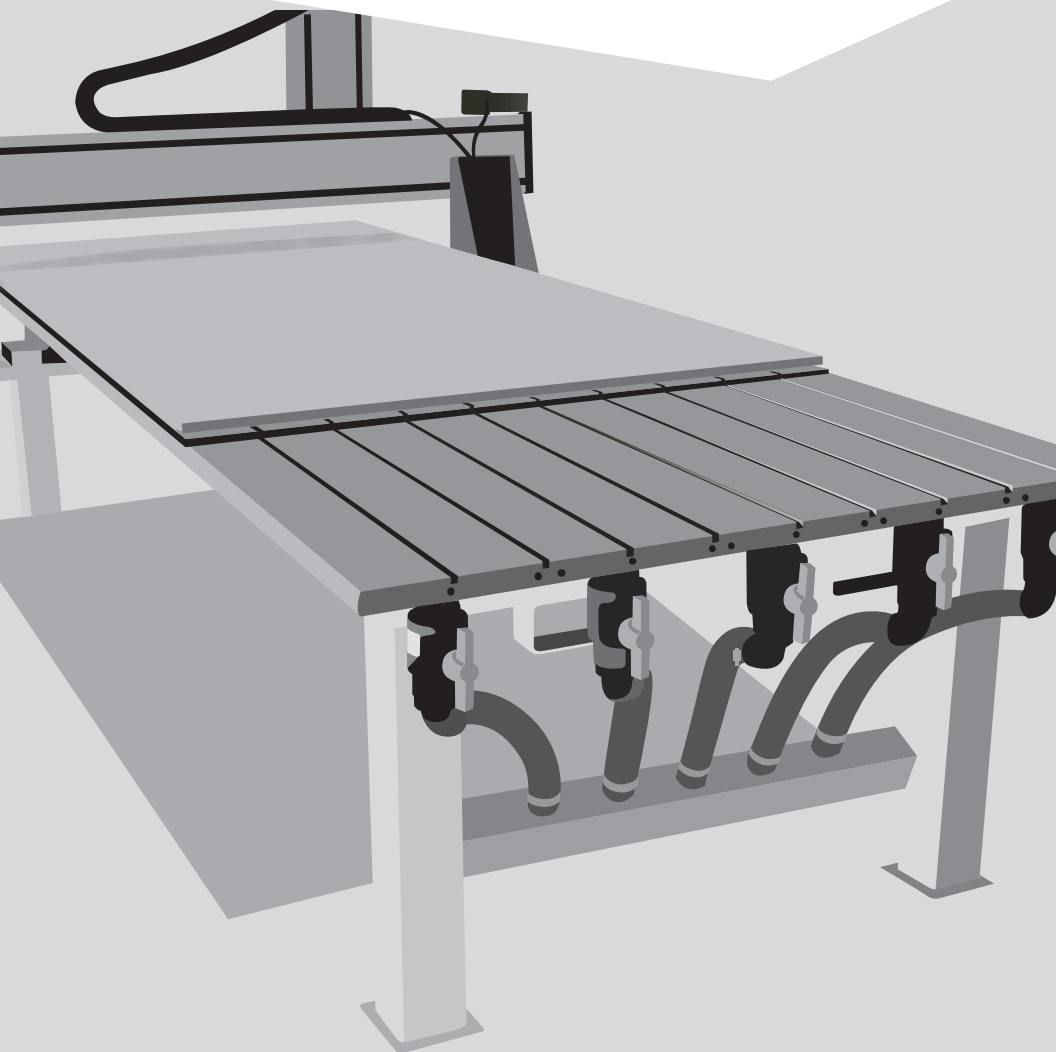


# Using the Techno CNC Router



## Table of Contents

Introduction: About the CNC Router	4
Introduction: Rhino and RhinoCAM	5
The Router	6
The Techno CNC Interface	9
Loading Your File	10
Previewing Your File	13
Setting Up the Router	14
Starting your Job	18
Pause & Emergency Stop	20
Safety	21
Glossary of CNC Terms	22



Instructional Fabrication's mission is to serve the contemporary art making needs of the School of the Art Institute of Chicago by providing instruction and technical assistance to students and faculty using the School's shops and fabrication facilities. Our staff work collaboratively across artistic and academic disciplines to support all manners of fabricating art objects.

*Instructional Fabrication is a division within the department of Instructional Resources & Facilities Management (IRFM).*

[www.saic.edu/instructionalfabrication](http://www.saic.edu/instructionalfabrication)

## About the CNC Router

The CNC (Computer Numerically Controlled) Router carves objects using a rotating bit on an overhead gantry system. The gantry carries the bit across the X, Y, and Z axis<sup>3</sup>.

Once a model is created using a CAD or 3-D application such as Maya, Cobalt, Form Z, or Rhino, the file can be imported into Rhino/Rhino Cam, to generate tool paths. Tool paths are then processed to generate code that can be translated by the CNC Router.

- The CNC Router can be used to machine foam, wood, plastics and some non-ferrous metals.
- Models should be created without under cuts, or surfaces more than 2” deep.
- Models taller than 2” should be cut into slices and machined as separate parts to be assembled later.
- Parts can be created using either 3-D models or 2-D drawings.
- 2 1/2 axis milling is used to cut flat parts out of sheets of material.
- 3 axis milling is used to “carve” objects modeled in 3-D applications.

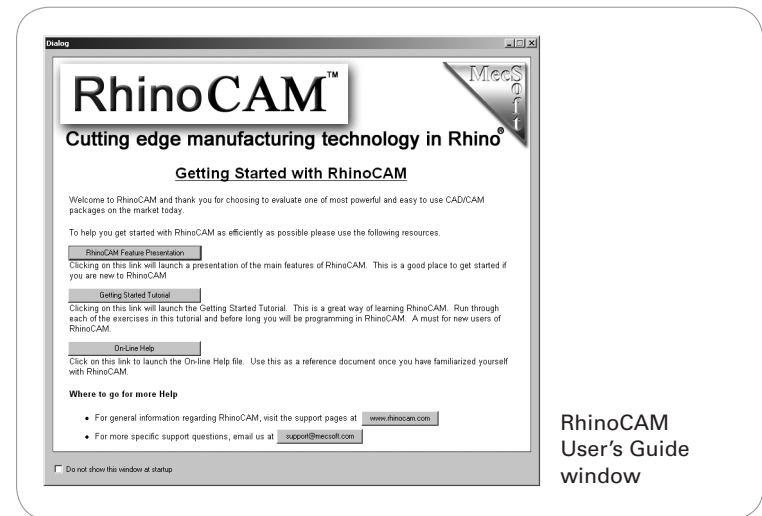
There are many kinds of toolpaths you can create using RhinoCam. This section will lead you through a few of the most common types. It is important to explore and preview different types of cutting. Seek a Studio Manager for advice on which toolpath to use for your application.

## Rhino and RhinoCAM

RhinoCAM is a toolpath creation plugin for Rhinoceros NURBS Modeling for Windows. Files modeled in other applications are imported into Rhino, and machining operations are created using the RhinoCAM plug in. If you are unfamiliar with “Rhino” you may want to explore the Help and Tutorial exercises included in the software.

Help > Learn Rhino > Getting Started

Rhinocam > Getting Started

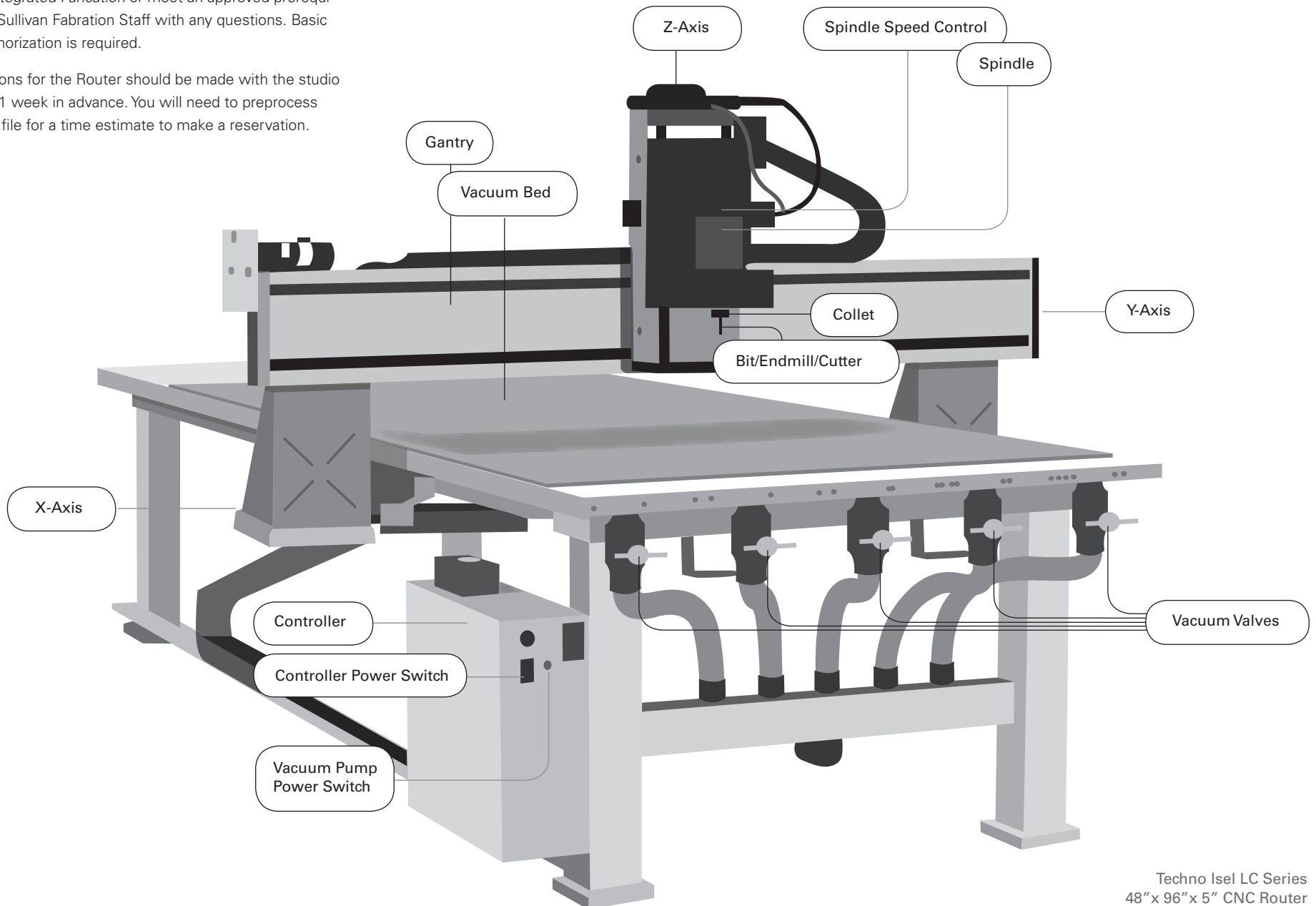


RhinoCAM  
User's Guide  
window

## Techno CNC Router

Before you can use the CNC Router, all users must enroll in *CNC Integrated Fabrication* or meet an approved prerequisite, see Sullivan Fabrication Staff with any questions. Basic Shop Authorization is required.

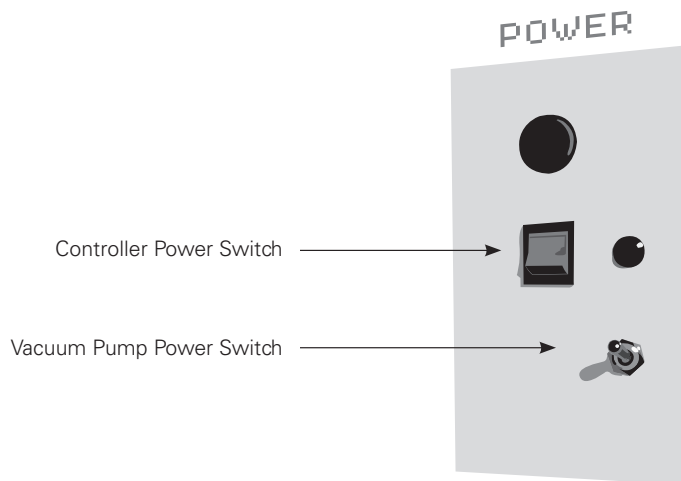
Reservations for the Router should be made with the studio manager 1 week in advance. You will need to preprocess your .ncd file for a time estimate to make a reservation.



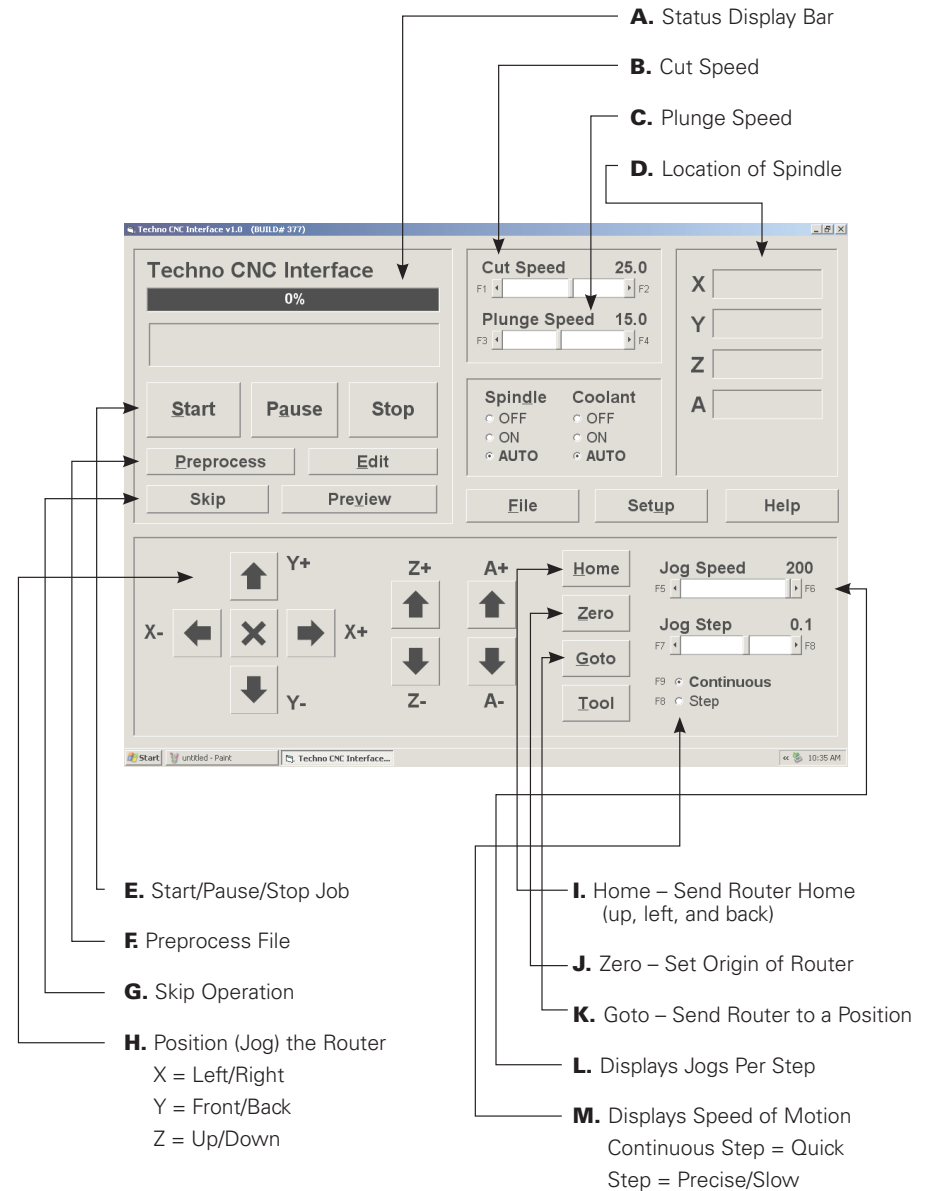
Techno IseI LC Series  
48" x 96" x 5" CNC Router

## To Begin Using the CNC Router

- To begin using the CNC Router, turn on the power to the Controller and Vacuum Pump.
- Open the TECHNO CNC Interface. A short cut to Techno is found on The desk top or in the Start > Programs menu.

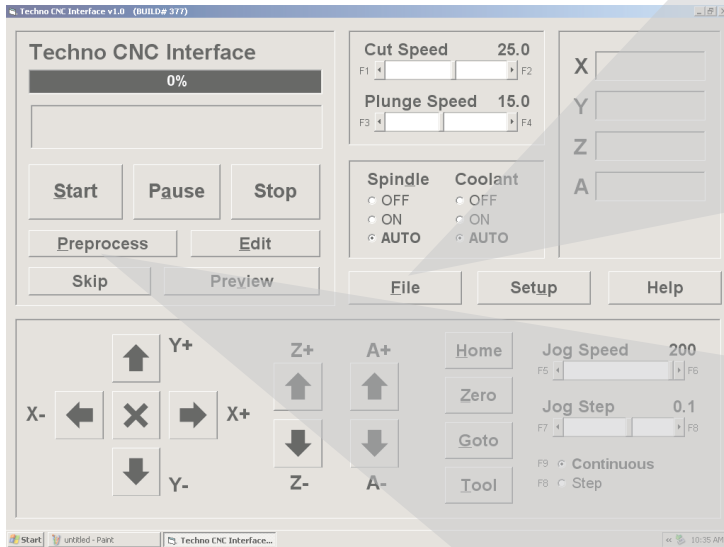
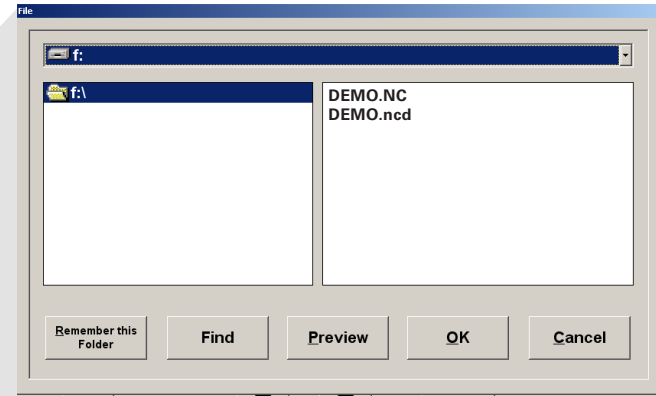


## CNC Interface



## A. To Import NC/NCD File

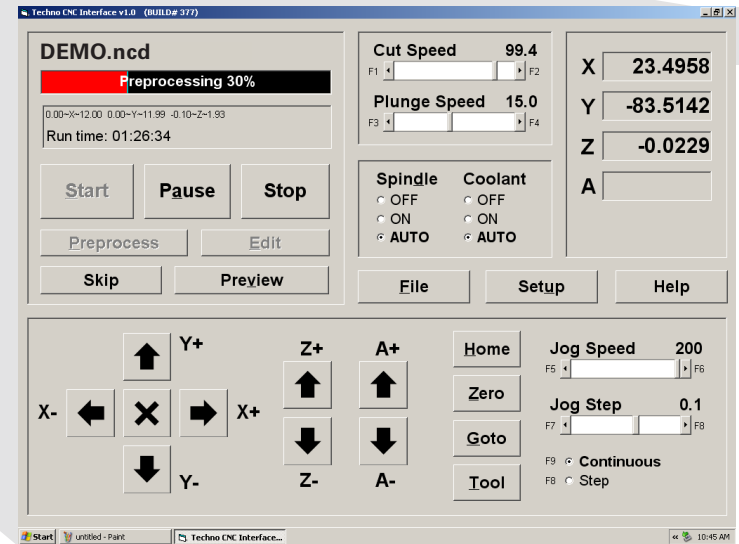
- Click File button. A file finder will appear. Navigate to find the appropriate drive location and pick your file.
- Click OK.



## B. To Preprocess NC/NCD File

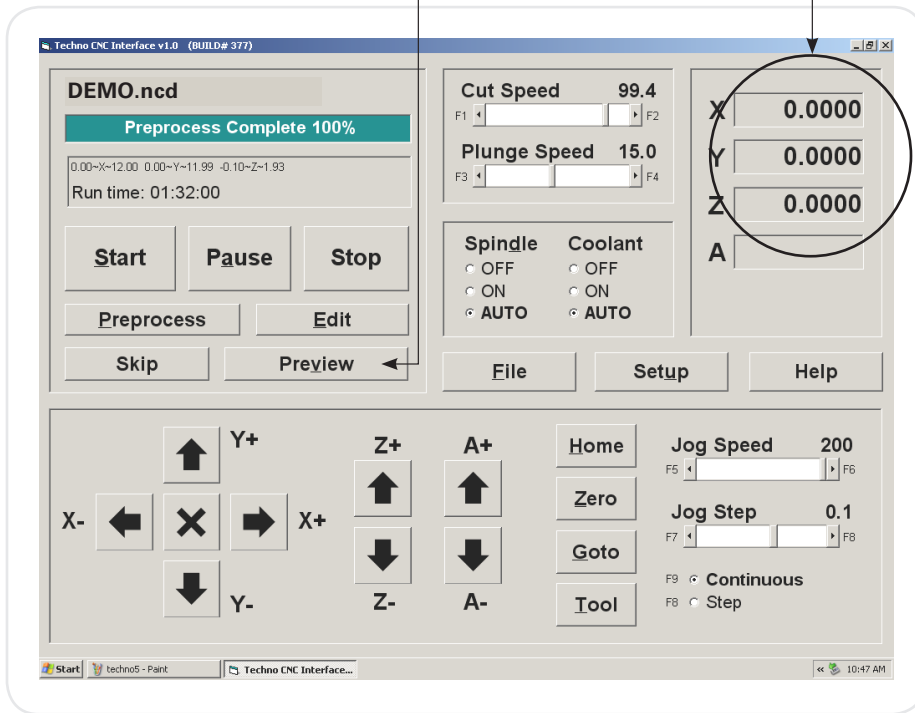
- Click Preprocess button.

Preprocessing analyzes your file, and calculates the job run time based on the cut speed and plunge speeds you set. After you start the job, this will change. The progress window will display an end time.



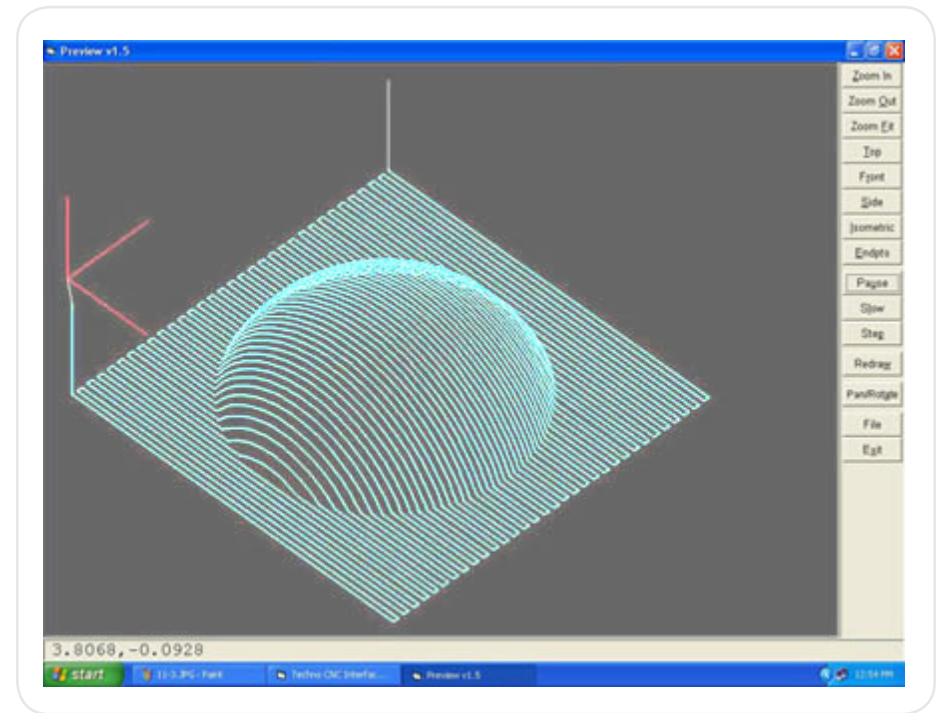
## CNC Interface (con't.)

- The start position of the mill end is located at the origin 0, 0, 0 coordinate.
- After preprocessing is complete, preview the file.



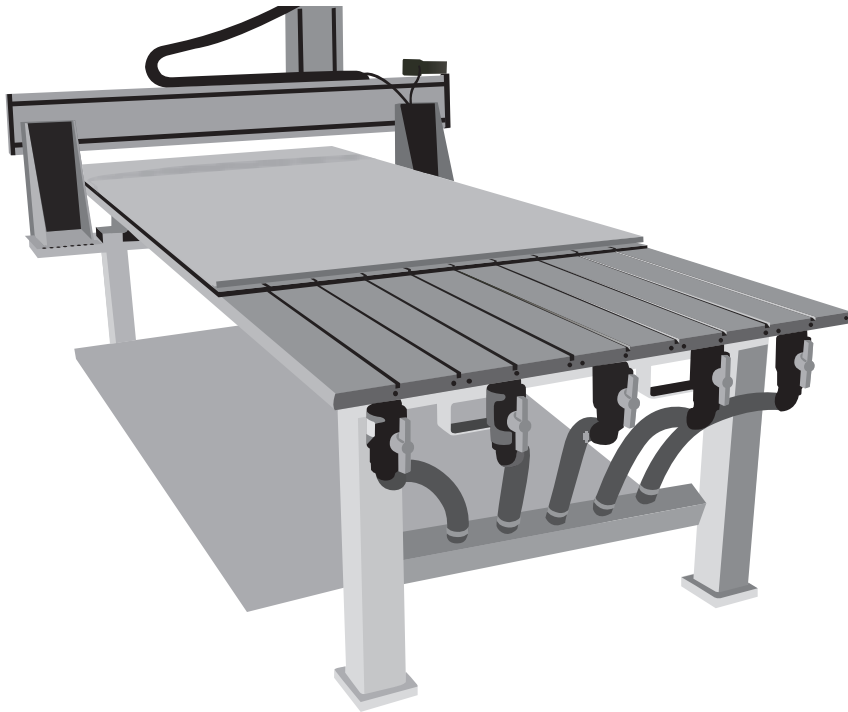
## Previewing File

- Once Preview loads, navigate using the buttons on the right. You may need to **Zoom Fit** to view the whole file.
- Verify that it is the right file and toolpath.
- Close **Preview** to return to the interface.



## Open Table Vacuum Valves

The table vacuum valves should be closed when you open the main valve. Open the table valves one at a time and wait for the table to grip the spoiler board before opening the next. The valves should be opened from one side to the other.

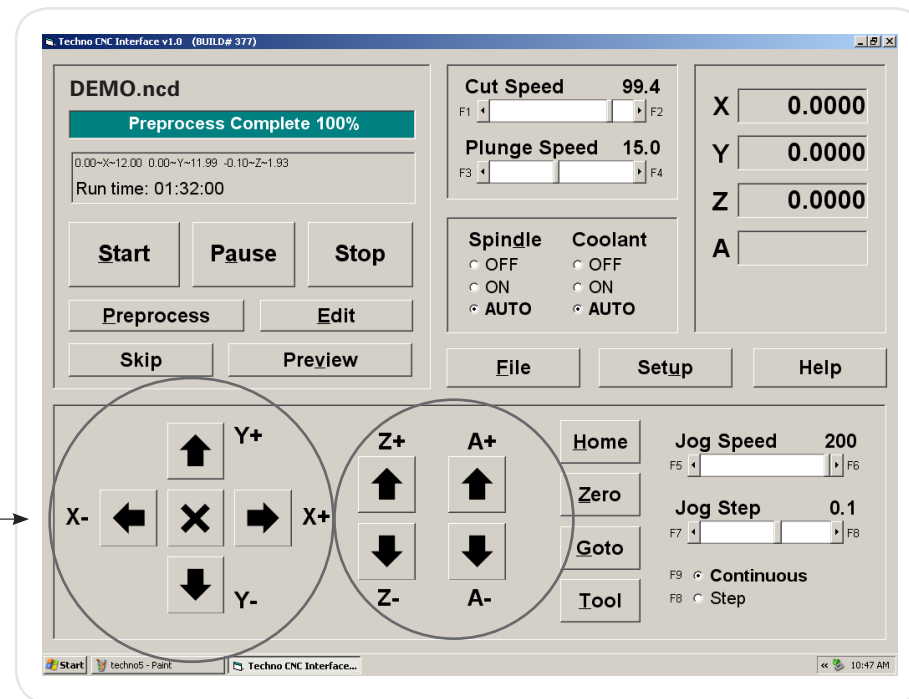


The CNC Router is equipped with a vacuum hold down table. A spoiler board is used on the table surface to protect the table from the cutting tool. This board should be in place already. Air drawn through the board creates a vacuum strong enough to hold some materials in place.

## Positioning the Router

Using the arrows and jog settings in the Techno CNC interface on the PC, *Jog* the router to position. You can also use hot keys, and the arrow and + - keys on the keyboard. You will need to position the router with the Z Axis all the way up, and clear of the table to position material and set up the Router.

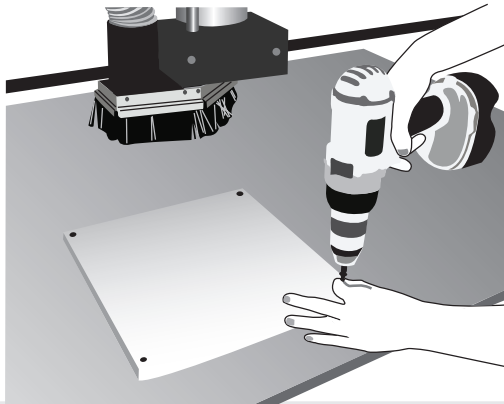
Set the Jog to Continuous for rapid movement or Step for precise positioning.





## Mounting Material to Table

The CNC Router is equipped with a vacuum hold down table. A spoiler board is used on the table surface to protect the table from the cutting tool. Air drawn through the board creates a vacuum strong enough to hold some materials in place. This may be the only hold down you need for large sheets of smooth surfaced material. Textured materials and smaller work pieces require mechanical attachments. Various methods including adhesives, screws (below, fig. 18.1), clamps and jigs may be used. The studio staff will assist you with determining the appropriate method.



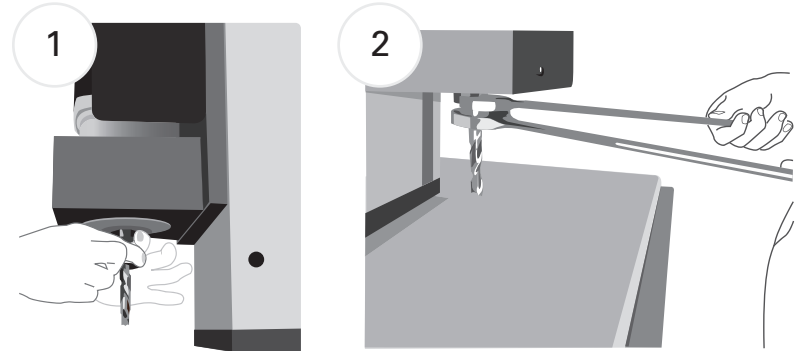
## Install Router Bit

After selecting the appropriate bit and collet for the application, install it in the spindle. Handle the tool carefully as it is very sharp. The bit should be seated fully in the collet. Make sure the power is turned off on the router before you start.

Hand Tighten the collet so that it does not slip out (facing page). The Bit is very sharp and should be handled with care. Always handle it by the shank. Tighten the collet by holding the top spindle nut, and turning the bottom collet nut. **Do not turn or pull on the bit.**

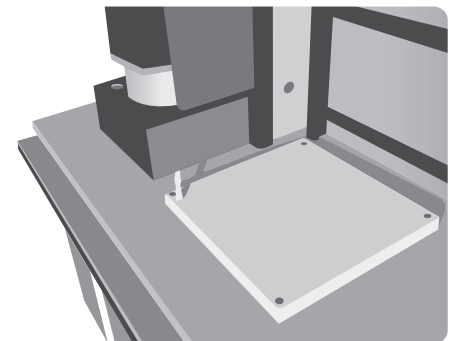
## Tighten Collet

Using the two large wrenches, Tighten the collet. Hold the top wrench on the spindle nut, and turn the bottom wrench. Always position the wrenches so that you are pulling them together. The bit must be secure or it could come loss during cutting.



## Set Origin (0,0,0)

The tip of the bit should be positioned on the surface of the material at the left front corner. "0" is the center of the bit. In the CNC Interface, select **Zero > All** to synchronize the origin in your model with the origin of the router.



## Install Dust Collection Shroud

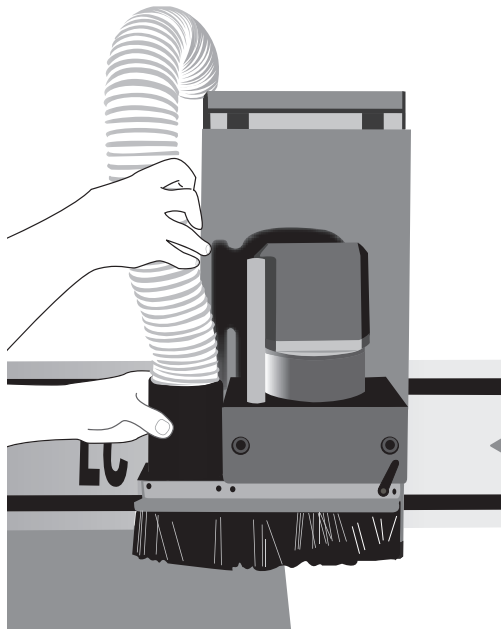
Jog the spindle all the way up, and away from your material. Slide the dust collection shroud into the spindle assembly from below. Be careful not to hit the bit with your hand or the shroud.

## Lock Shroud

Tighten the set screw in the spindle assembly to secure the shroud.

## Install Dust Collection Hose

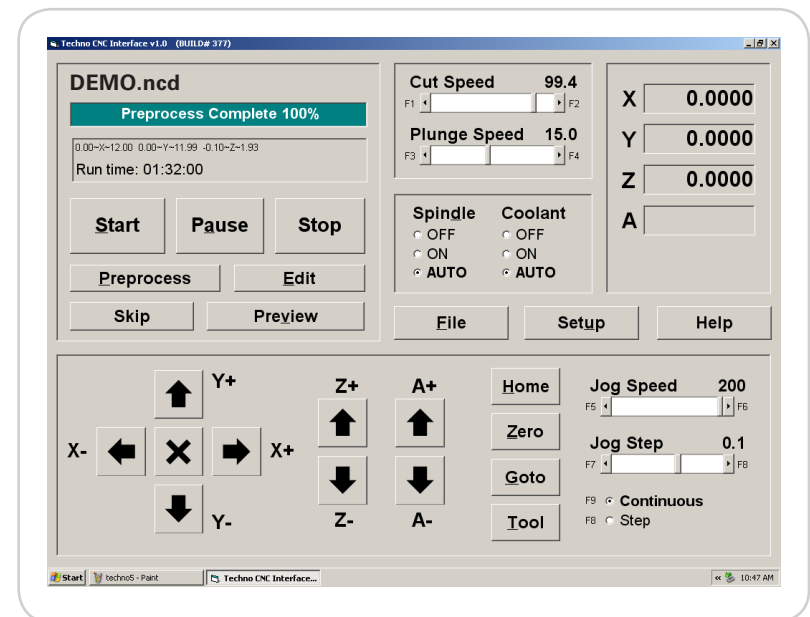
The dust collection hose should be used whenever possible. Seat the hose connector securely in the shroud. You may also need to turn on the dust collection system.



## Start Your Job!

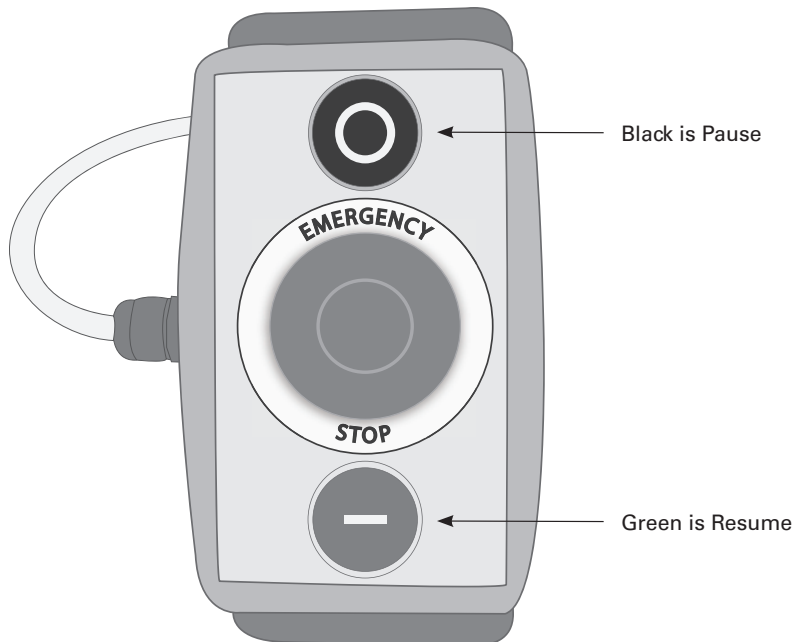
Make one last check to make sure everything is installed properly, before you start. Turn the power and set the spindle speed on on the router. Set the spindle to auto in the cnc interface.

Press **Start**. The interface will now display a resume button, and a load tool flashing prompt. Press **Resume** to start your job.



## Pause & Emergency Stop

There is a remote pause/emergency stop control box located next to the PC. If at any time during the job, something goes wrong, or seems awry, press and hold the pause button until the CNC router stops. After pausing, the CNC Router can be jogged to other positions for adjustments, and the job resumed. If there is smoke, safety hazard or injury, press the Emergency Stop button. The Emergency stop will lock the Router off until it is reset.



## CNC Router Safety

- Wear eye and ear protection at all times. The Shop will supply you with both goggles and earplugs.



- Keep the table clean and free of tools and loose materials.
- Never leave the router running unattended. If you need to leave, please ask someone to watch it for you or pause the job.
- Never lean on the table or rest hands or feet on the router while it is running.
- Do not attempt to make adjustments or reposition the material while the router is running.
- Use the dust collector or vacuum while the router is cutting.
- Do not exceed recommended feed speeds or spindle speeds.

## Bit Safety

- Handle bits with care, they are very sharp. Grip them by the unfluted shaft only.
- Never pull a bit out of a collet - push them out from the back.
- Store bits in the drawer or in plastic cases.
- Do not use damaged, dull, or burnt bits.

# GLOSSARY OF CNC TERMS

**Climb Cut (aka downcut):** In a climb cut, the cutter motion is downriver, opposite its rotation—the cutter rotates clockwise but is fed counter-clockwise and away from the rotation. The resulting cut can produce a finer surface edge but climb cutting is extremely hard on bits and can be easily overdone. Climb cuts should only be made with extreme caution and in special situations.

**CNC:** Computer Numerically Controlled

**Collet:** The holder that locks cutting tools into the spindle of the CNC router.

**Conventional Cut (aka upcut):** In a conventional cut, the cutter rotation and tool motion are in parity, the cutter is fed upstream into the material, producing a robust cut but with a tendency to tearout. Conventional cutting is the standard cutting direction and should be used in most situations.

**Cutting Edge:** The sharpened edge of the tool used for cutting.

**Downcut Bit:** The flutes and cutting edge of a downcutting bit are spiraled downward pushing chips and grain down into the material.

**Feed Rate:** The rate at which the cutter moves from point A to point B. Expressed in inches/minute.

**Flute:** The grooves cut into the tool to allow chip removal during cutting.

**Fixture:** A template or clamping device which helps to either locate or affix your material to the machine. If, for example, a student is milling many parts from identical sized stock, it would make the process go faster to affix a location template on the spoil board so each new piece of stock would not need to be re-zeroed and could instead use the same origin point as the last milled piece.

**Gantry:** The horizontal support member of the CNC, on which the Spindle is mounted. The gantry moves along the X-axis. The router moves laterally along the gantry to describe the Y-axis.

**G-Code:** A system of numeric code and machine instructions. When you *Post* a file, Rhinocam will produce G-Code. Once the file is posted a plain text file will open in a new notepad window. This is the G-Code. Do not alter this code! You will notice the file type is .nc, this stands for numeric code.

**Generate:** The command in Rhinocam which tells Rhinocam to interpret the information you have provided in the MOp and develop toolpaths.

**Home:** An absolute coordinate position programmed into the CNC machine. When you send the machine *Home* you are sending it to this absolute coordinate position. If you are running many files with a consistent origin, it is useful to know where that origin is in relation to the Home coordinates. In the event of a CNC shutdown or some other loss of your origin coordinates, the coordinates can always be reproduced from this Home position.

**Knowledge Base:** A group of basic tool-paths with speeds and other parameters already defined. Most often, when a student begins developing a toolpath they should start with one of the toolpaths found in the *Knowledge Base* file so that some options are already established.

**Machining Operations (MOps):** The term used to describe the tool path and settings created for machining.

**Origin:** In Rhino, this is the world 0,0,0 coordinate. On the CNC machine you establish this point when you zero out the bit on the material. When you do this you must zero out all three coordinates. Once you have established this origin, the CNC will interpret that origin to be the position of the 0,0,0 origin in your Rino file.

## GLOSSARY OF CNC TERMS *continued*

**Post (or postprocess):** The process by which Rhinocam converts tool paths into numeric code which are interpretable by the CNC machine.

**Post Processor:** In the Rhinocam setup tab, there is an option to define the *Post processor*. When you define this you are telling Rhinocam what kind of CNC machine you are using. Every CNC speaks a slightly different form of numeric code (.nc) so it is necessary to tell Rhinocam what kind of machine it is going to be communicating with so that it posts the .nc file in the correct format. In SFS, we only have two machines, Techonsel and Onsrud TM3. If your post processor is setup for any other machine, your file will not run correctly.

**Region:** The area in Rhinocam you use to designate a milling area. Generally, regions are closed curves on or around the geometry you wish to mill. Think of a region as a way to tell Rhinocam where to look for milling geometry.

**Spindle:** The rotating motor that spins the cutting tool.

**Spoil Board:** Sheet of underlying material, beneath your working stock, which serves as both a place to affix your stock as well as a sacrificial cutting surface. While it is allowable to set your toolpaths such that they cut into the spoil board, be sure they do not cut through it!

**Stock:** The material that will be machined on the CNC Router.

**Straight Flute:** Straight flute bits are used in applications where the material may flex or lift during cutting, when for instance the material is thin and light.

**Support Material:** The material outside of the part you are milling. When you are milling, there must generally be some sort of support material in

addition to the part being milled so as to secure the parts to the machine as they are being milled.

**Tool:** The rotary cutter used in the CNC machine to cut the material. Tools are made from a variety of materials and in a wide range of sizes and geometry.

**Toolpath:** A definition of the motion of the tool as it makes a given cut. In Rhinocam, the toolpath itself is expressed by a light green line. Each green line represents one linear motion of the tool and defines the center location of the bit at any one time.

**Upcut Bit:** The flutes and cutting edge are spiraled upward pulling chips and grain up out of the material.