CNC Milling

step-by-step guide to 3 axis CNC milling
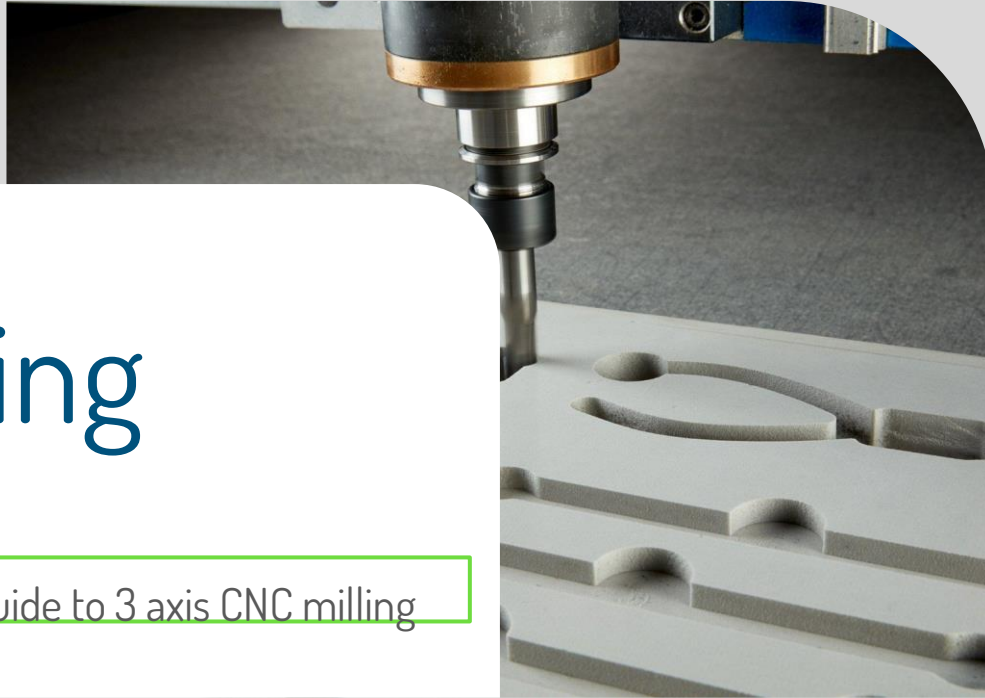
Intermediate Level

Table of Contents

The following tutorial is for operating a Techno CNC LC Series 4896 machine.

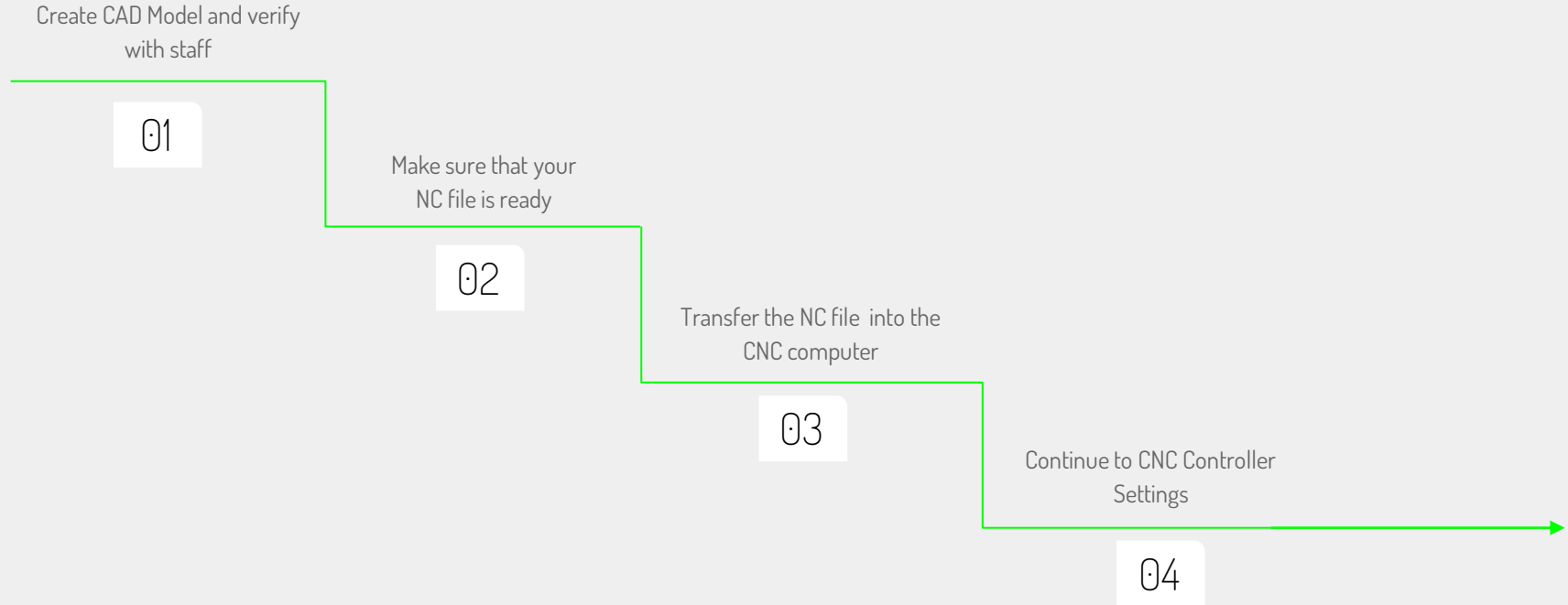It is a 3 axis, full sheet, air cooled CNC with vacuum hold down and a touch probe.

The CAM software is RhinoCAM 2014.

The CNC controller is using the final release version of Techno CNC's software.

Your organization may differ in software, hardware, and specific procedures.

Create CAD Model and verify
with staff

01

Make sure that your
NC file is ready

02

Transfer the NC file into the
CNC computer

03

Continue to CNC Controller
Settings

04

CNC Milling

Is your NC file uploaded to the CNC computer?

Yes	No
Skip to slide 7	Move to following slide with upload instructions

Techno CNC Interface

0%

Cut Speed 46.3
F1 F2

Plunge Speed 39.7
F3 F4

X 0.0000
Y 0.0000
Z 0.0000
A

Start	Pause	Stop

Spindle **Coolant**
- OFF   - OFF
- ON   - ON
- AUTO   - AUTO

| Preprocess | Edit |
| Skip | Preview |

| File | Setup | Help |

Y+
Z+   A+

X-   ← ✕ → X+

Y-
Z-   A-

Home
Jog Speed 205
F5 F6

Zero
Jog Step 0.005
F7 F8

Goto
F9 - Continuous

Tool
F8 - Step

CNC Milling
NC File Upload

To transfer the NC file to the CNC controller...

User mounts external storage or DAS.

User tranfers physical media to other user.

User mounts external storage or DAS to retrieve and copy media.
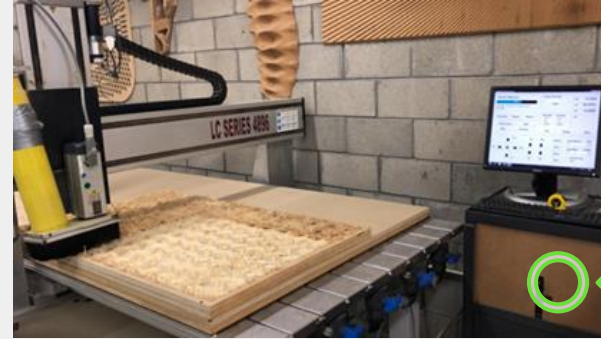
01
Find the NC file that you have post-processed.

02
Insert a flash drive (ask a fab lab employee if you do not have one.)

03
Copy the NC file to the flash drive.

04
Eject the drive and insert it on the front panel of the CNC controller.
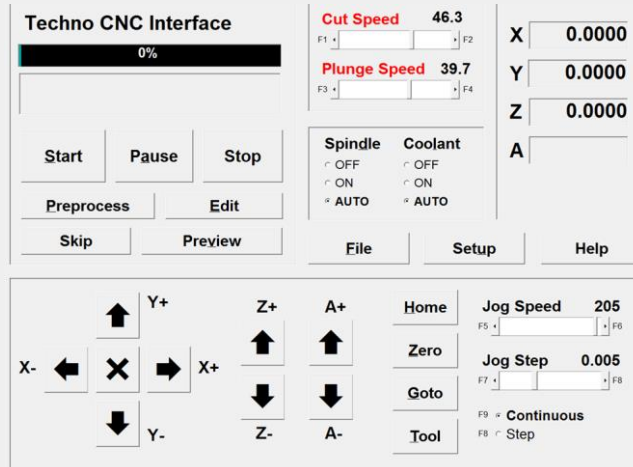
Picture Credit

03 CNC Controller

Techno CNC Interface — 0%

Cut Speed 46.3
Plunge Speed 39.7

Spindle: OFF / ON / AUTO
Coolant: OFF / ON / AUTO

Start | Pause | Stop
Preprocess | Edit
Skip | Preview
File | Setup | Help

X 0.0000
Y 0.0000
Z 0.0000
A

Y+ / Y- / X- / X+ / Z+ / Z- / A+ / A-

Home | Zero | Goto | Tool

Jog Speed 205
Jog Step 0.005
Continuous / Step

NOTE:

In certain cases it is of benefit to allow the **.nc file** to control certain **machine behaviors**.

In other cases it is of benefit to allow the **CNC controller software** to control (or override) **machine behaviors**.

01
Verify that the controller software standard presets are correctly configured.

02
Consult with CNC Operator to adjust the settings to best suit your project.

03
Use the PREVIEW option to check the spindle path and verify the correct order of operations.

04
In the PREVIEW window ZOOM, SIMULATE, and change viewports to check the spindle path.

[Picture Credit](#)

ROBOTICS ACADEMY | FIU | Communication, Architecture + The Arts | NSF
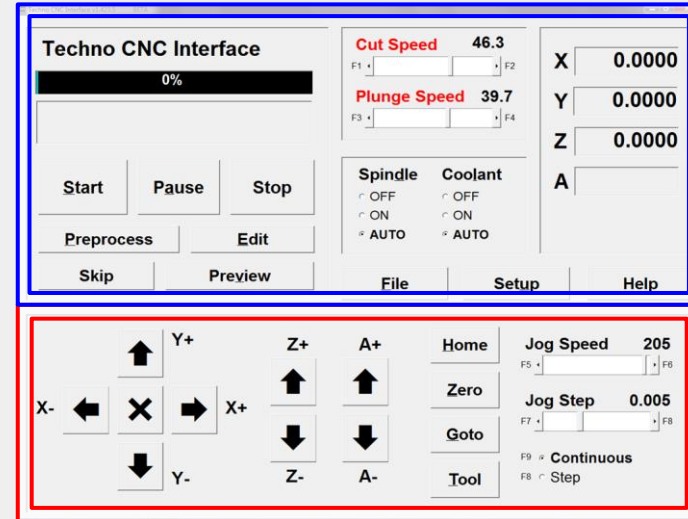
CNC Milling
CNC Interface

Examine the settings:
The Interface Section highlighted in blue pertains to an NC file.
The Interface Section highlighted in red establishes machine pre-sets and manually jogs the CNC machine servo motors..

Verify that the settings are appropriate for your project.

If unsure, consult with lab manager or CNC operator.

Yes

No

Move on to proceeding steps

Change the settings.

Techno CNC Interface

Cut Speed 46.3
0%
Plunge Speed 39.7

X 0.0000
Y 0.0000
Z 0.0000
A

Start   Pause   Stop

Spindle   Coolant
OFF   OFF
ON   ON
AUTO   AUTO

Preprocess   Edit
Skip   Preview   File   Setup   Help

Y+   Z+   A+   Home   Jog Speed   205
X-   X+   Zero   Jog Step   0.005
Y-   Z-   A-   Goto   Continuous
Tool   Step

Picture Credit

PREPROCESS: Calculates all program motions in their specified order including spindle and servo motor speeds, acceleration, deceleration, and bit change operations. All .nc files must be preprocessed before running for the first time.

HOME: Returns all of the servo motors to an operator defined pre-set location.

ZERO: Establishes a physical X, Y, Z position on the CNC machine corresponding to 0, 0, 0 in virtual space. The zero position must be verified relative to stock and tool tip before running a program.

PREVIEW: Generates a graphic simulation showing all program motions in their specified order. This is a useful step to verify the proper order of operations and to discover potential collisions, motion errors, or operational inefficiencies.
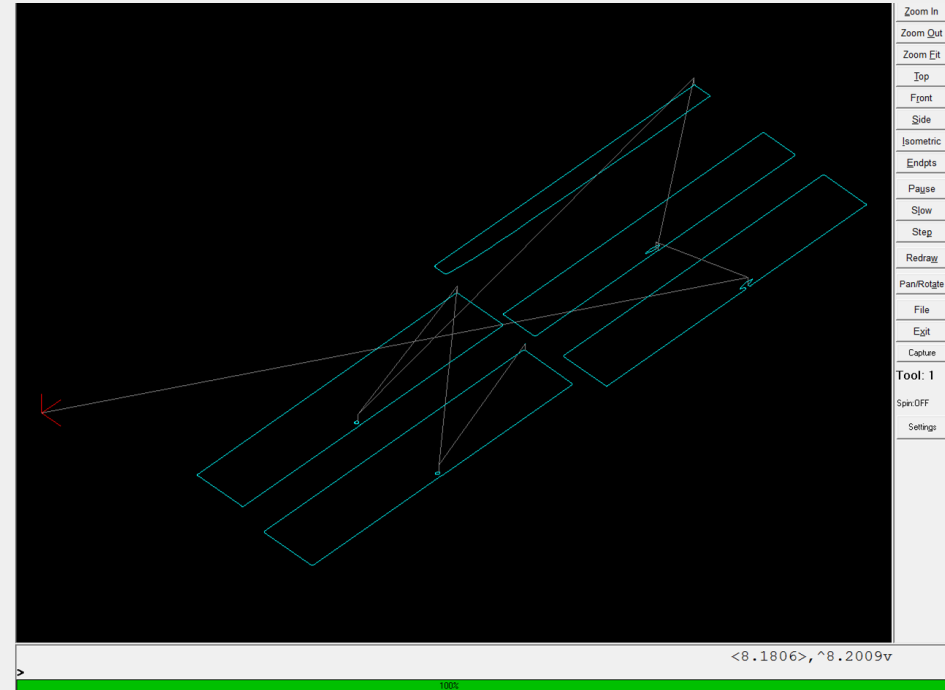

Picture Credit
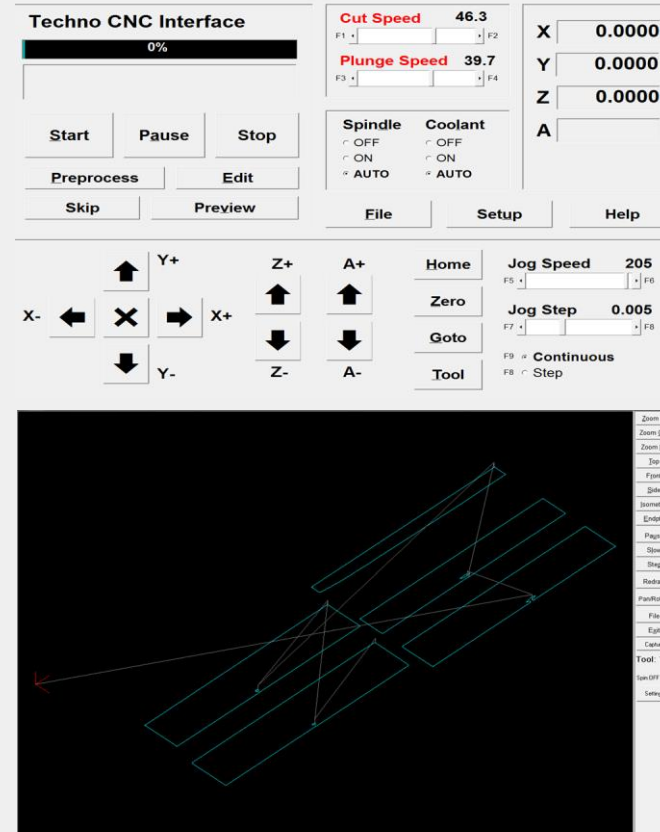
CNC Milling
Preview

Open the PREVIEW window by selecting the **Preview** button in the upper left portion of the interface.

The Preview pop out window will dynamically draw the path the milling tool will follow.

Verify that the procedures, parameters, and the order of operations are correct.

Use the options on the right side of the window to change view parameters such as view angle, zoom, and playback speed.


Zoom In
Zoom Out
Zoom Fit
Top
Front
Side
Isometric
Endpts
Pause
Slow
Step
Redraw
Pan/Rotate
File
Exit
Capture
Tool: 1
Spin:OFF
Settings

<8.1806>,^8.2009v

100%

Picture Credit

04 Procedure

CNC Milling
Preprocess

To preprocess an .NC file, simply click the **Preprocess** button on the top left of the interface.

Once preprocessing has completed, It will display:
 Minimum and maximums of each axis
 Estimated milling time

Check the minimum and maximum values for each axis. This is the best way to avoid potential damage to the CNC Machine.
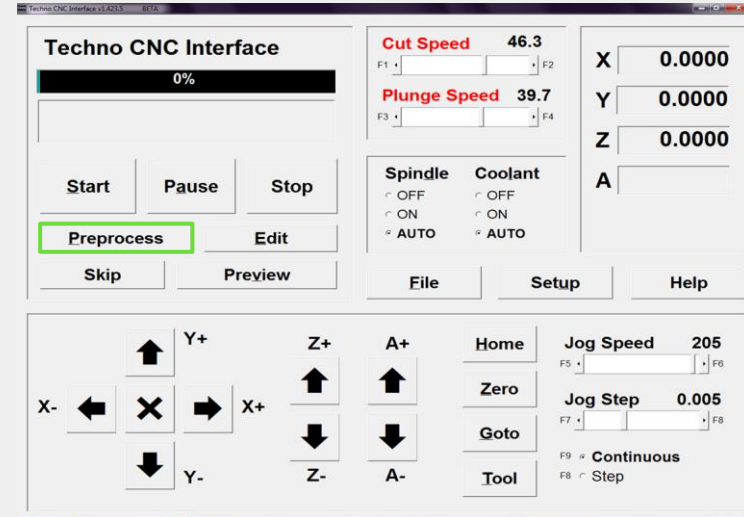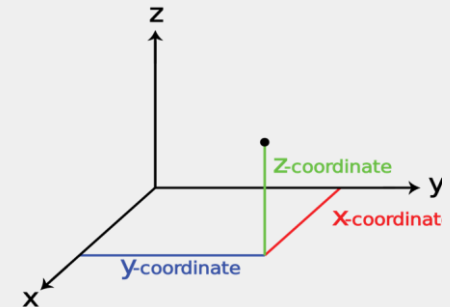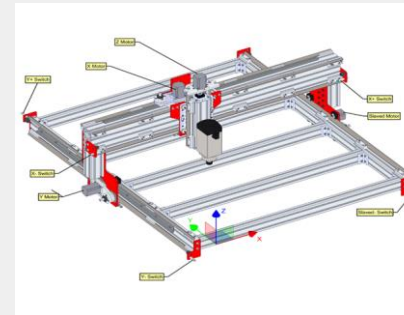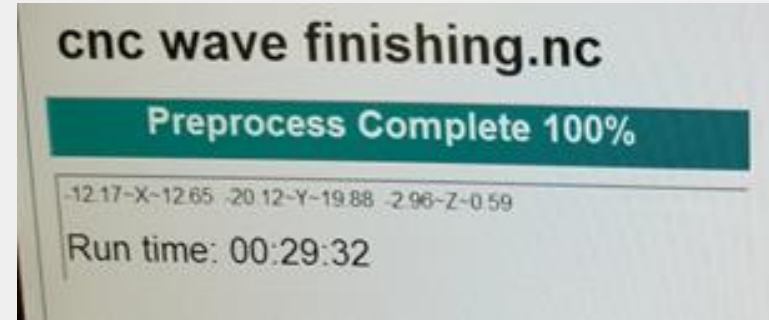
CNC Milling
Preprocess

The CNC Machine works along the X, Y and Z coordinates. Each one has a minimum and a maximum that the machines extend can reach.

Please check the minimum and maximum values for each axis. This is the best way to avoid machine crashes and damage!

cnc wave finishing.nc

Preprocess Complete 100%

-12.17~X~12.65 -20.12~Y~19.88 -2.96~Z~0.59

Run time: 00:29:32

X MIN	X MAX	Z MIN	Z MAX	Y MIN	Y MAX

https://socratic.org/questions/what-is-a-z-coordinate

Picture Credit

ROBOTICS CADEMY FIU | Communication, Architecture + The Arts NSF

CNC Milling
Work holding

Use a WORKHOLDING method best-suited for your CNC project.

Workholding is method for positioning and fastening material to the CNC table. Secure workholding is safer and less prone to potential accidents or errors.
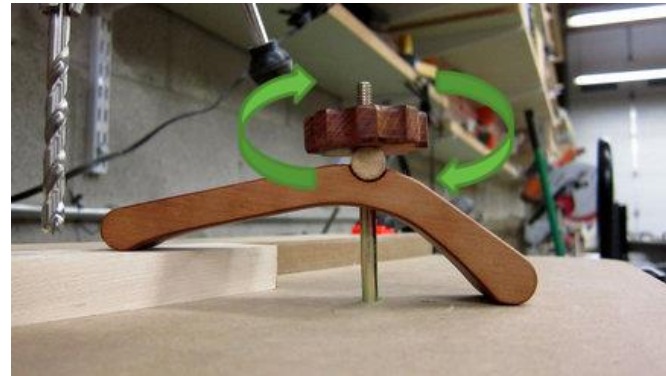
Review the example showing hold down clamps.

T – Slots

Hold Down Clamp

Insert the bolts in the T slots needed to hold down the project.

Tighten the knobs to secure the material to the CNC bed.

CNC Milling
Installing Cutting Tools

Cutting tools are attached to a rotating spindle using a collet and retainer nut assembly. While some CNC machines feature an automatic tool changer, others require bits to be manually changed using a pair of wrenches.

Collets of different sizes accommodate bits with various standard shaft diameters.

Once removed from the spindle, the collet can be detached from the retainer nut with modest lateral pressure.


Picture Credit

CNC Milling
Installing Cutting Tools

COLLETS
Collets MUST be properly inserted into the Collet Retainer Nut before being fastened to the spindle or tool holder. Improper collet installation may result in personal injury or permanent damage to equipment.

Attach: Insert the collet at a slight angle aligning the groove of the collet with the eccentric extractor ring on the inside surface of the collet retainer nut. Push the collet into the retainer nut until the groove and ring are positively engaged.

Detach: Apply firm lateral pressure to the collet while firmly grasping the collet retainer nut until the eccentric extractor ring disengages from the groove in the collet.

Collet groove

fig. 1

Eccentric extractor ring

fig. 2

Collet Retainer Nut

Collet

Collet Wrenches

Improper Fit

Proper Fit
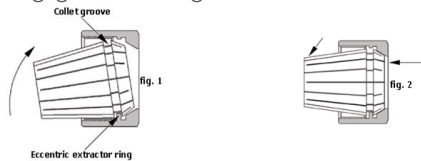
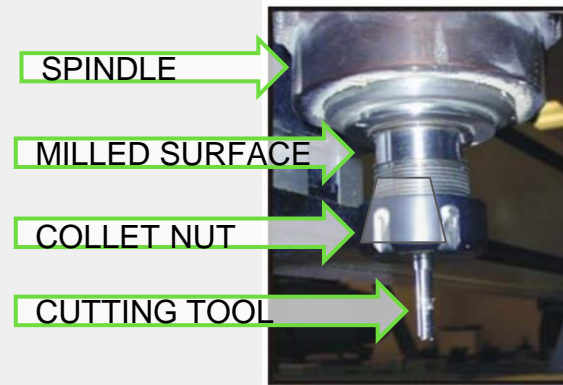# CNC Milling
Installing Cutting Tools

MANUAL BIT CHANGING
Insert a bit into the collet and hold it in place.

Slide the castle wrench into the slots on the collet retainer nut ensuring that the tangs on the wrench slide vertically into corresponding grooves in the collet retainer nut.

Use the crescent wrench to hold the spindle in place. Flat spots milled into the spindle axle directly above the threads for the collet retainer nut accomodate a crescent wrench.

Apply pressure to the castle wrench to tighten or loosen the collet retainer nut, as required.

Castle Wrench

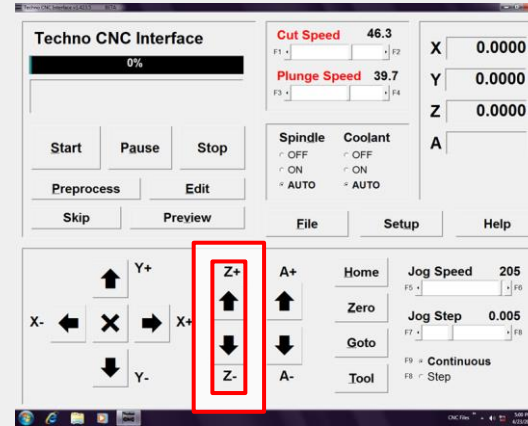SPINDLE

MILLED SURFACE

COLLET NUT

CUTTING TOOL

Picture Credit

07 Homing & Zeroing

CNC Milling
Process

Tool Selection

Homing

Establish Work Zero

Touch Pad Zeroing

Manual Zeroing

Picture Credit
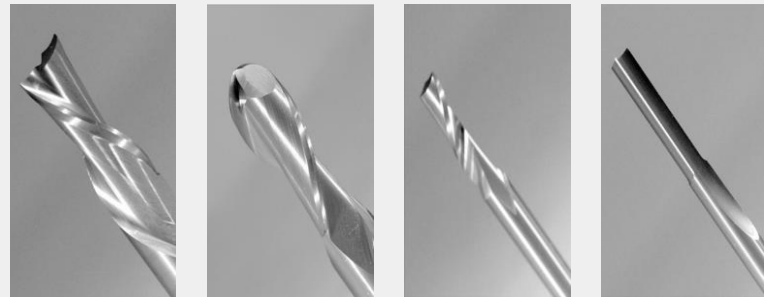
CNC Milling
Tool Selection

Each CNC project requires careful consideration for tooling selection. There are a wide variety of cutting tools designed for different materials and milling procedures.

Consult with an experienced fabricator and CNC operator to develop a tooling strategy optimized for the materials and cutting procedures.

Picture Credit

CNC Milling
Tool Selection

Up cut end mills feature two or more upward helix blades which help remove chips and dust from the cut while providing a smooth cut on the bottom surface of the stock.
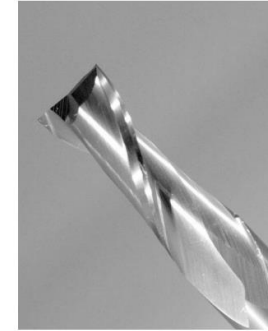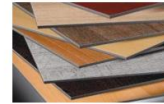
Upcut end mills are optimized for two dimensional curves or straight lines and right angled, stepped surfaces. They are not particularly suitable for organic compound curvatures.

UPCUT SPIRAL

The up cut spiral bit is used when getting the chips out of the cut is critically important, or when the material is laminated or coated on one side only, and is being machined face down. This tool geometry does present a risk that the top of the material being cut can chip or fray.

This tool is a good choice for mortise and tenon cutting needs and also provides an excellent option for short runs on Corian or phenolic. (longer runs should consider a PCD diamond bit as it is more cost-effective).

Up cut bits provide the ability to feed faster than down cut bits as the chips are pulled out of the cut by the upward shearing action and keep the tool running cooler.
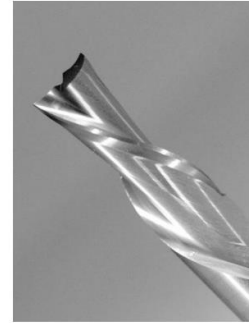
GDP GUHDO

© 2018

CNC Milling
Tool Selection

Down cut end mills feature two or more downward helix blades. They are a good choice for materials with a delicate top surface such as laminates or veneers.

Down cut end mills are optimized for two dimensional curves or straight lines and right angled, stepped surfaces. They are not particularly suitable for organic compound curvatures.

DOWN CUT BIT

The down cut spiral bit provides a superb top surface finish, but it does run the risk, depending on application, of pushing the chips into the cut and bogging down the bit. With good dust extraction and proper chip load, this should not be a problem. It is often selected for doing grooves, dados and rebate cuts. Important to remember, always use the shortest possible cut length for dados, grooves and rebates, as the longer tools will have more deflection and can break more easily or provide poor finish due to deflection when cut length is excessive.

If there is an ongoing dado or rebate cut requirement, an insert bit will decrease cost considerably within a short period of time if the tool is available in the diameter needed. (Image below)

© 2018

GDP GUHDO

Picture Credit

ROBOTICS ACADEMY   FIU Communication, Architecture + The Arts   NSF

CNC Milling
Tool Selection

Ball nose or round nose bits are excellent for organic or curved surfaces.

Their main drawback is that they cannot cut sharp corners or 90 degree stepped surfaces.

BALLNOSE UPCUT BIT

A ball nose tool is a great choice for cove and fluting operations, but is also the tool of choice when complex shapes have to be surfaced. A flat bottom bit would leave lines and a poor finish, but a ball nose bit, due to its rounded shape, provides seamless passes.

Another option for this tool is a carbide insert version (pictured below), which is always much more cost effective if the project is ongoing with a continuous tooling need.

GDP GUHDO

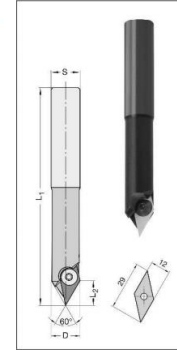© 2018

CNC Milling
Tool Selection

V-Groove or Chamfer bits are excellent for engraving and signmaking. They are also good for fine details and text.

As the name implies, they carve V-shaped grooves in the surface or outside edge of material. Common angles include 30, 45, and 60 degrees.

V-GROOVE AND ANGLE CUTS

For mitre folds, insert V-Groove Bits are available as standards for 45, 60 and 90 degree (included angle) cuts. Other angles can be produced as custom tools. The insert knives are double sided and provide two life cycles each.

Insert V-Groove bits are also a great solution for lettering, engraving and decorative cuts as well as beveling the inside edges of shaker doors and square corner cut-outs.

GDP GUHDO

© 2018

CNC Milling
Tool Selection

O-Flute bits are purpose built for soft plastic. They use a single non-helical straight cutting edge.

These bits cannot be successfully used for any other material.

O-FLUTE BIT (SOLID CARBIDE)

This image shows a solid carbide, straight O-Flute. The flute shape is ground into what looks like a half-circle.

This particular geometry is used for cutting flexible plastics. The flute is straight and helps keep a light weight flexible plastic from moving or lifting on the router table during machining.

Use this tool on materials such as:
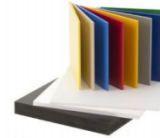
Polycarbonate
ABS
Polystyrene
PVC
and other flexible plastics

GDP GUHDO

© 2018

Picture Credit

CNC Milling
Tool Selection

O-Flute up cut bits are optimized for harder plastics. They feature a wide single aggressive helical blade optimized for chip extraction.

These bits cannot be successfully used for any other material.

O-FLUTE UPCUT

The O-Flute upcut bit features the same O-Flute geometry needed for efficient chip removal when routing plastic, and also has an upward shearing angle to bring the chips out of the cut and provide cleanest possible edge finish.
This tool is the preferred router bit for hard and rigid plastic materials, such as:

Acrylics
Nylons
Plexiglass
and other rigid plastics.
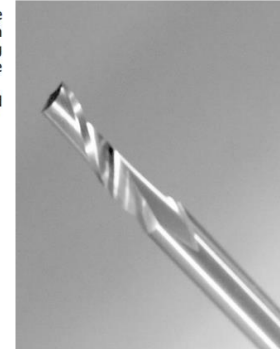
GDP GUHDO

© 2018

CNC Milling
Homing

Homing is a procedure for calibrating a CNC machine. Homing should be performed before each milling job to ensure that the servo motors do not inadvertently exceed their limits of movement.

During homing, the servo motors travel all the way to the end of each axis engaging the limit switches, then reversing and returning to the pre-established home position.

Picture Credit
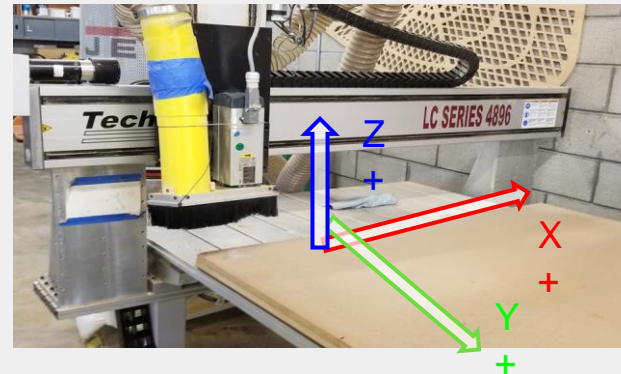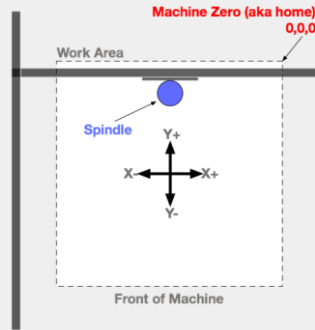
CNC Milling
Homing

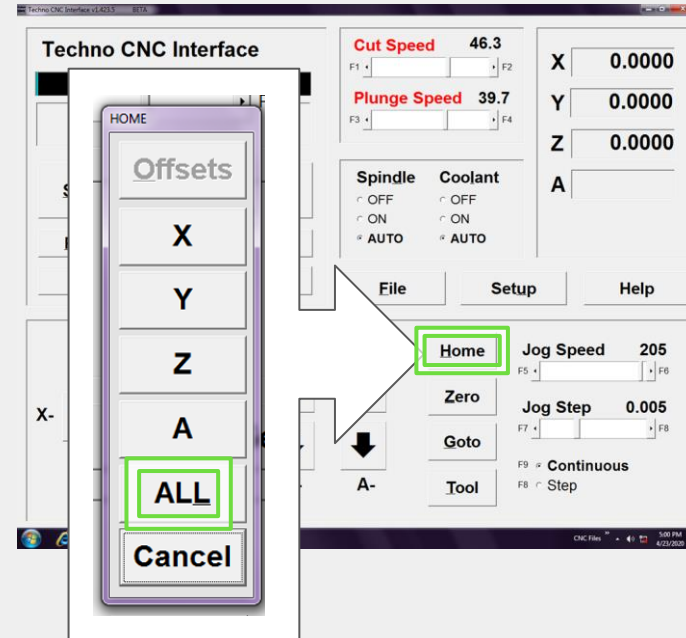Select the HOME button in the lower portion of the menu.

01

Select ALL to home all the machine axes.

02

To accelerate the homing process, select the "5" key

03

CNC Milling
Zeroing

Zeroing is a procedure for establishing the X, Y, Z coordinates of milling stock on the CNC table. To Zero a piece of stock, choose a datum point (either the top or bottom of stock), then jog the various axes of the CNC machine until the bit rests at the lower left corner of the stock.
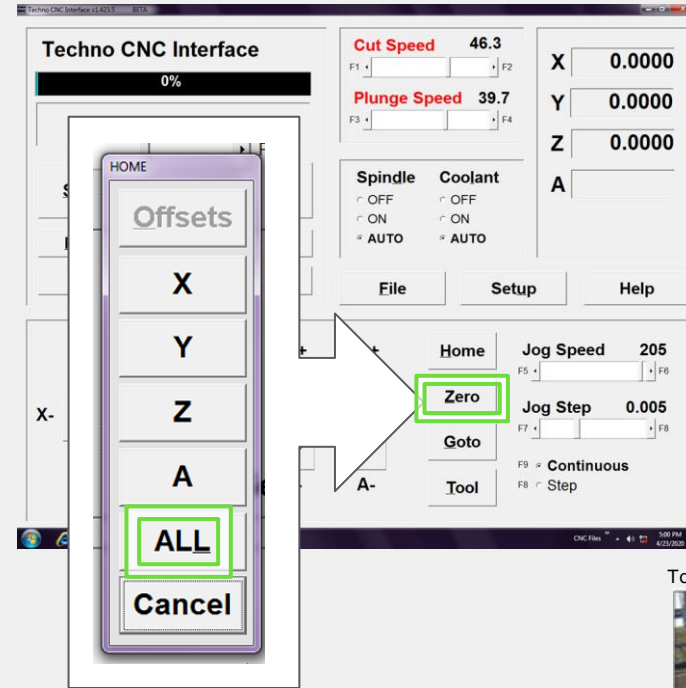
Manual Methods

Use your eye. Sight along one side of a board and center the bit.

Using a known diameter bit, bring it to the edge of your material until it just touches. Then raise the bit and move it one-half the diameter of the bit to find the center.

Touch Pad Zeroing

Prop

CNC Milling
Safety Check

Final check:
Is the material securely clamped?
Is the proper tool loaded?
Is the collet fastened tightly?
Is the dust collector on?
Are the overhead dust filters on?
Are there any objects on the work area that need to be removed?
Are you wearing proper safety equipment?

Select START to begin milling.

Pause or stop the job if something abnormal happens (material comes loose, tool breakage, smoke etc)

Do not leave the machine unattended while in operation!

09 Milling Monitoring

CNC Milling
Check Milling While in Progress

Monitor your project while the CNC Machine is in operation to ensure everything is in order. Pause the program periodically to maintain a clean and safe work environment.

Thank you!