



HAAS CNC MILL

VERSION 3.1



AUTODESK
PIER 9

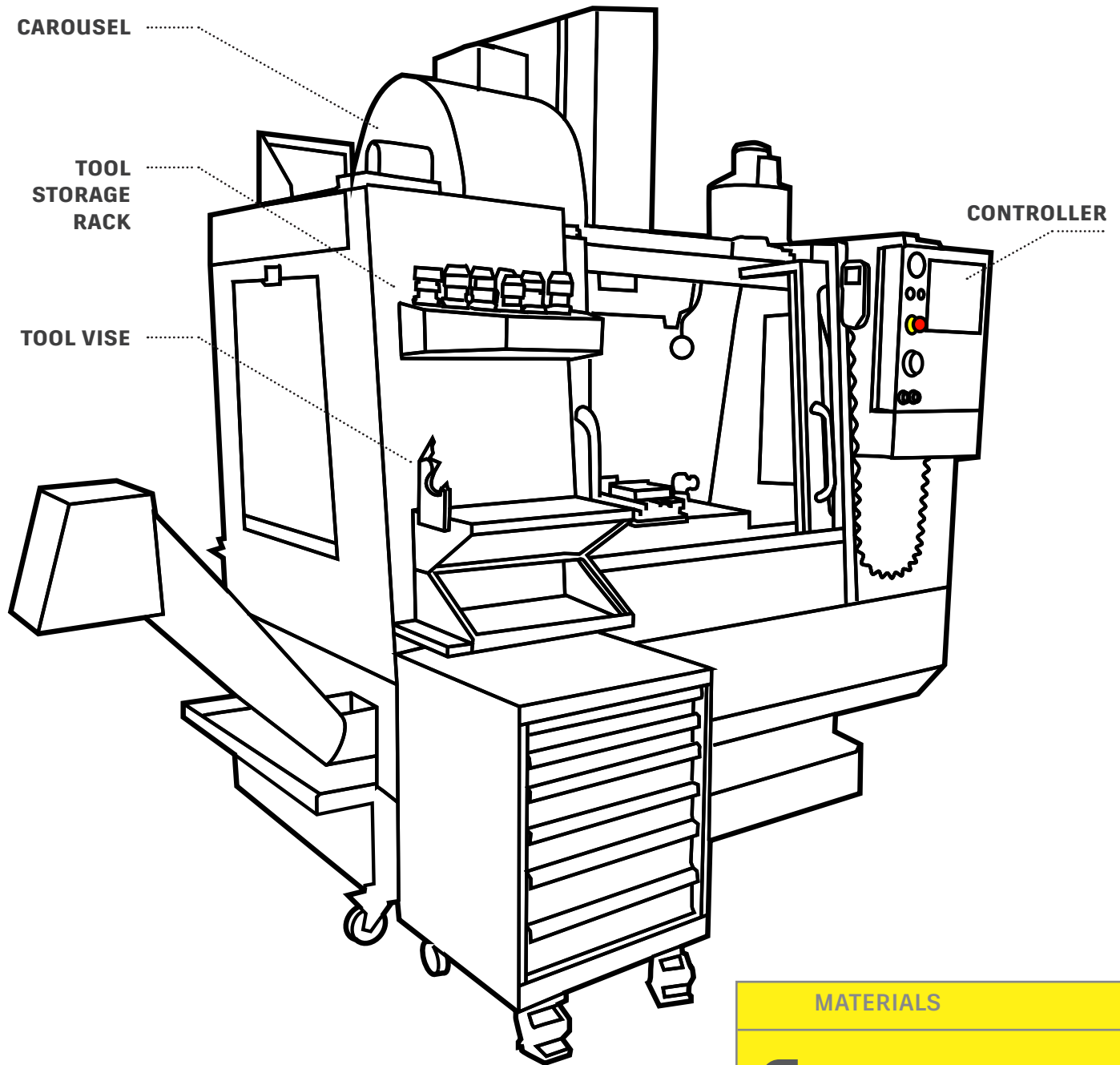
HAAS CNC MILL

MACHINE
CONTROLS

VERSION 3.1

THE HAAS MILL IS CAPABLE OF MAKING COMPLEX, ACCURATE PARTS FROM YOUR CAD MODEL.

P. 2



MATERIALS

i ALLOWED MATERIALS

- + Metal
- + Plastic

⊗ BANNED MATERIALS

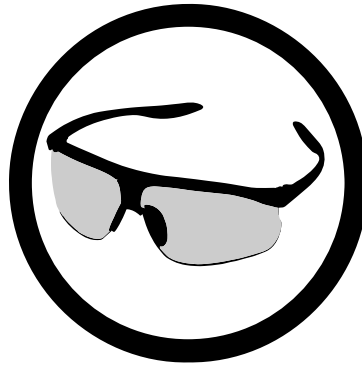
- + Wood
- + Carbon fiber and composites

? SEE SHOP STAFF FIRST

- + All other materials



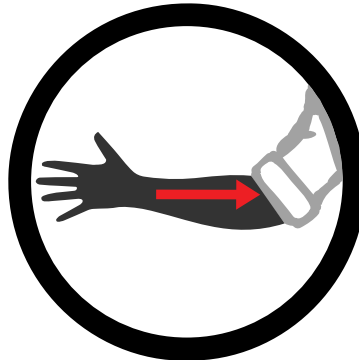
**PROTECT YOUR EYES
FROM FLYING CHIPS.**



Always wear safety glasses.



**NEVER OPEN THE DOOR
WHEN THE SPINDLE IS
ROTATING.**



Opening the door when the spindle is rotating is extremely dangerous, and will result in the loss of machining privileges.



**TAKE PRECAUTIONS
WHEN HANDLING SHARP
OR HEAVY ITEMS.**

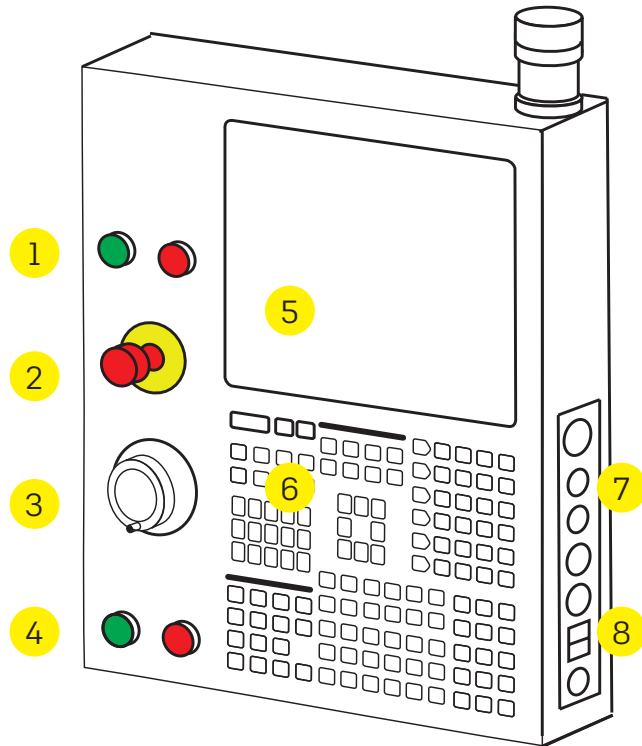


The tooling is sharp and can be heavy. Wear cut resistant gloves and ask for assistance if needed.

CONTROLLER

The controller is where you will run the mill.

1. Power On & Power Off buttons
2. E-stop
3. Handle jog
4. Cycle Start & Feed Hold buttons
5. Display
6. Control panel keys
7. USB port
8. Light switch

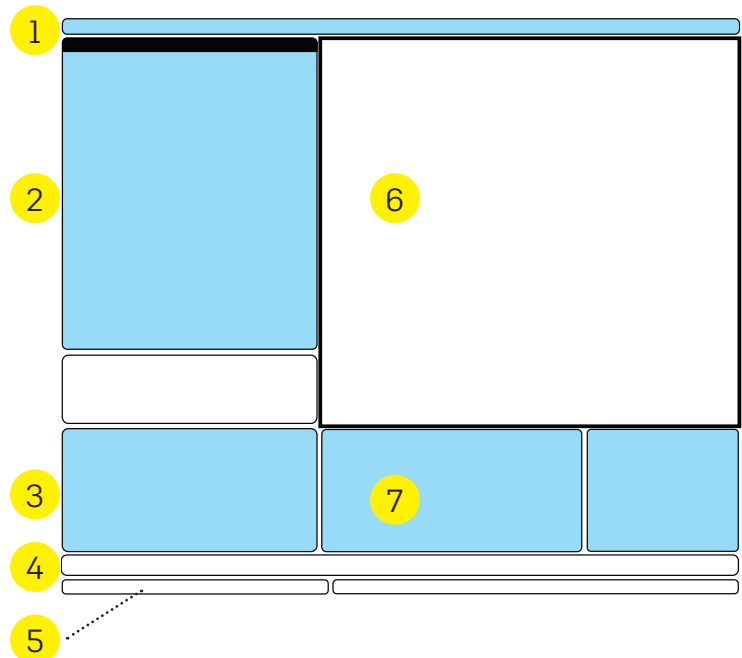


DISPLAY

The mill displays information in various areas of the screen. If a screen area is white, it is the *active window*. *Inactive* windows are light blue.

The contents of the windows can change, depending on which mode the mill is in. This diagram shows the mill is in *operation mode*.

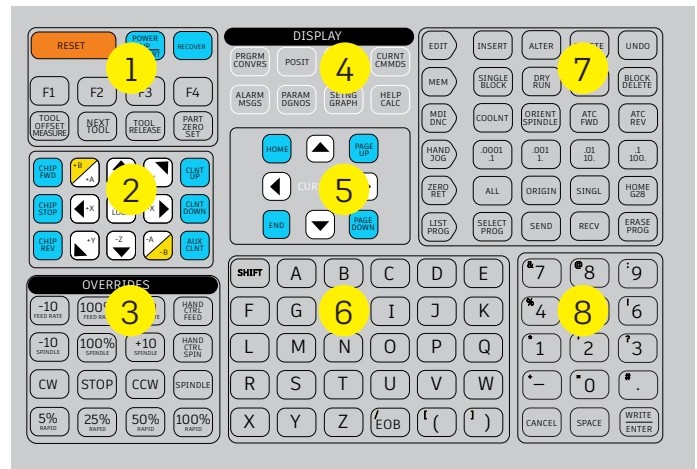
1. Current mode
2. Program display
3. Speeds and feeds information
4. User messages/Status bar
5. Command input
6. Main display
7. Position display



CONTROL PANEL

The control panel for the mill is laid out in groups of keys that have related functions.

1. Function keys
2. Jog keys
3. Override keys
4. Display keys
5. Cursor keys
6. Alpha keys
7. Mode keys
8. Numeric keys



MODE KEY DETAILS

The mode keys work differently than the other keys.

- + The first column of keys puts the machine into a specific mode.
- + The four keys to the right are commands related to that mode.

Example:

1. Press **LIST PROGRAM** to take you to the screen listing the .nc programs.
2. Press **SELECT PROGRAM** to select the program you have highlighted (using the cursor keys).
 - ▶ This only works if you are in LIST PROGRAM mode.

MODE	MODE FUNCTIONS			
EDIT	INSERT	ALTER	DELETE	UNDO
MEM	SINGLE BLOCK	DRY RUN	OPTION STOP	BLOCK DELETE
MDI DNC	COOLNT	ORIENT SPINDLE	ATC FWD	ATC REV
HAND JOG	.0001 .1	.001 1.	.01 10.	.1 100.
ZERO RET	ALL	ORIGIN	SINGL	HOME G28
LIST PROG	SELECT PROG	SEND	RCV	ERASE PROG

LIST PROGRAM SELECT PROGRAM

MACHINE SPECIFICATIONS

The Haas VF2SS CNC mill is a large, powerful mill with the following specifications:

- + 30 horsepower spindle
 - + 12,000 RPM maximum spindle speed
 - + Traverse (rapid relocation) speed up to 1,400 inches per minute
 - ▶ On Pier 9 you are required to operate the mill at 5% rapid.
 - ▶ All *non-cutting moves* and *tool changes* will happen at 5% of the maximum speed so that you have time to react if something goes wrong.
 - ▶ A reduced *rapid speed* will not change the *cutting speed*.
 - + Work envelope of 30" x 16" x 24" (X, Y & Z)
 - + Tool carousel can hold up to 24 tools
-

REQUIRED FOR USE

Before you start your work, make sure you have the following materials:

- + An .nc file from your completed CAM file
 - + A thumb drive to copy the .nc file to the mill
 - + Setup Sheets from your CAM software
 - + A copy of this Quick Start Guide
-

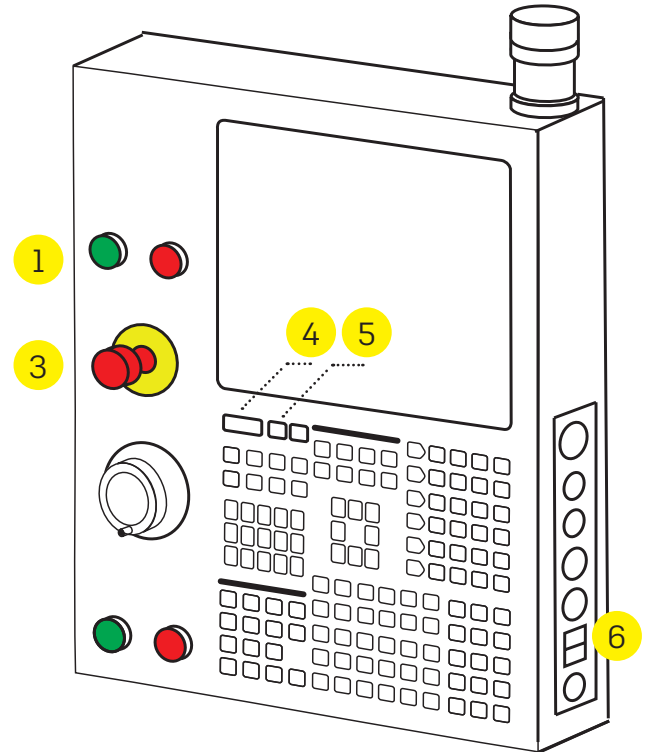
STEPS FOR SUCCESSFUL USE

To make a part, you need to complete each step in order.

1. Start up the mill.
2. Insert custom tools into the holders.
3. Inspect the Pocket Tool Table and Carousel.
4. Install tools into the machine.
5. Set the tool offsets.
6. Set the Work Coordinate System.
7. Load the program.
8. Run the program.
9. Clean up.

START UP THE MILL

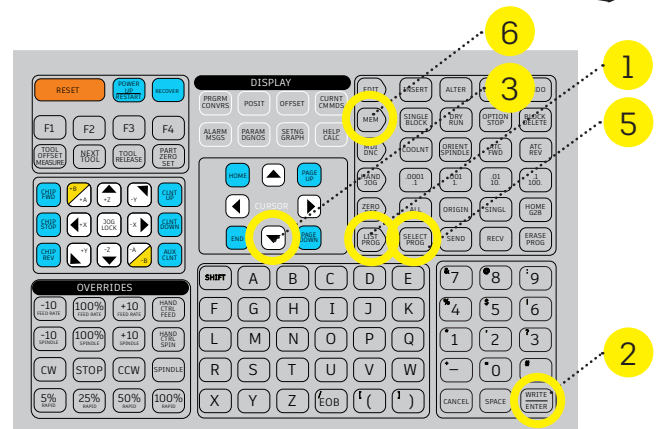
1. Press the green **POWER ON** switch.
 - ▶ Wait for the computer to boot up and give instructions.
2. Close the doors.
3. Reset the E-Stop by rotating clockwise.
4. Press **RESET**.
5. Press **POWER UP RESTART**.
 - ▶ This will cause the table and spindle to move quickly.
6. Turn on the interior light.



Warm up the spindle

If the mill has not been run today, you'll need to run the spindle warm-up program.

1. Press **LIST PROG**.
2. Press **WRITE/ENTER** to open the MEMORY folder.
3. Use the cursor keys to navigate.
4. Select 002020 (WARM-UP Pier 9).
5. Press **SELECT PROG** to select the program.
6. Press **MEM** to send the program to machine memory.

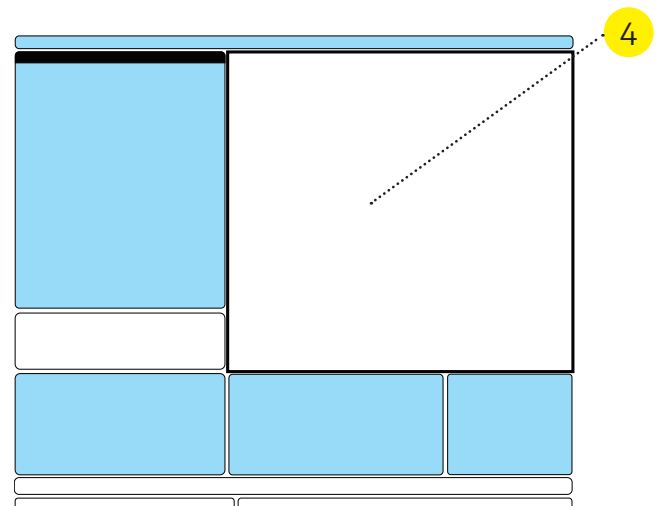


IN MEM MODE PRESSING CYCLE START WILL START THE PROGRAM AND RESULT IN MACHINE MOVEMENT.

7. Press **CYCLE START** to run the program.
 - ▶ This takes about 15 minutes.
 - ▶ Prepare your tools while waiting.

Note: You may get an error stating the mill hasn't been run recently.

- + Press **CANCEL** to acknowledge the error message.
- + Press **CYCLE START** to run the program.



LIBRARY TOOLS VS CUSTOM TOOLS

Library tools

- + These tools are stored next to the mill, and correspond to the Pier 9 Haas Mill Tool Library that you imported into your CAM software.
- + Each tool is engraved with a number that corresponds to its number in the CAM software, from 101-175.
- + These tools are loaded into tool holders by Shop Staff.
- + The mill knows how long these tools are, so you will not need to measure them for length.

Custom tools

- + These are the tools in your CAM program that are not in the library, such as drill bits.
- + Custom tools do not have a default number in the CAM software. You will assign custom tools numbers 1-99, in the chronological order that they are used.
- + You need to insert them into the tool holders.
- + The mill does not know the tool length, so you'll need to measure each custom tool in a later step.

INSERT TOOLS INTO THE HOLDERS

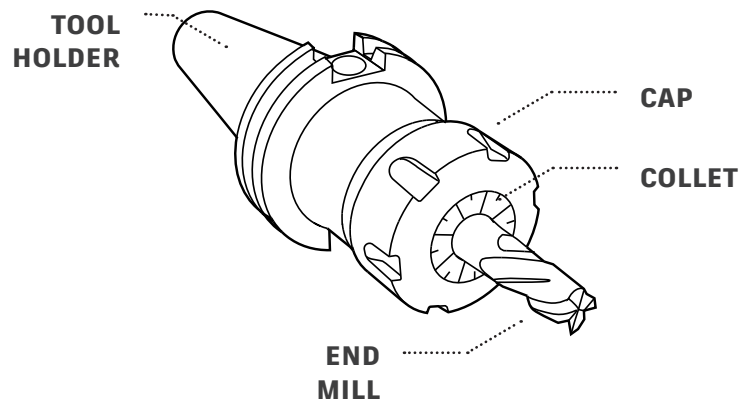
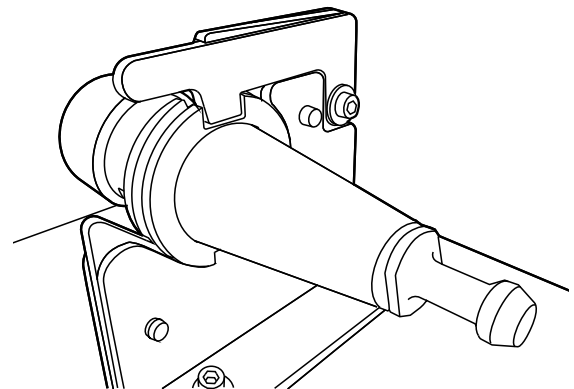
There are two kinds of tool holders: drill chucks and *collets*. Both kinds should be held in the *tool vise* on the front of the mill while preparing custom tools.

Drill chucks

- + Only use for holding drill bits.
- + Tighten by applying force clockwise with the correct wrench for the chuck.

Collets & tool holders

1. Use a collet that is the same size as the *shank* of the cutting tool.
2. Snap the collet into the *cap*.
3. Screw the cap onto the collet a few turns.
4. Insert the cutter as deep as possible.
 - ▶ Grip the shank, not the flutes.
5. Tighten the cap with the wrench.



STAGE THE HOLDERS

In the order listed in your Setup Sheet, set the tools in the storage rack on the front of the mill.

MEASURE TOOL LENGTH TO AVOID TOOL HOLDER COLLISIONS.

P. 9

Check the length of the custom tools

In the CAM software, the library tools and tool holders are modeled. The software will detect a collision involving the holder.

The holders for custom tools are not modeled, and collision checking does not happen in software. You need to check for collisions manually.

If you set your Work Coordinate System (WCS) to the top of the stock in your CAM setup, the *Minimum Z* entry in your Setup Sheet will show how deep the tool will go into the material.

1. Look at the *Minimum Z* entries in your Setup Sheet for each custom tool.
 - ▶ This is the deepest point that the tool will go into the stock.
2. Measure the amount of tool sticking out of the tool holder.
 - ▶ This is called the *approximate tool length*.
3. The approximate tool length must be larger than Minimum Z.

Operation 2/9

DESCRIPTION: 2D Adaptive1

STRATEGY: Adaptive 2D

WCS: #0

TOLERANCE: 0.004in

STOCK TO LEAVE: 0.02in

OPTIMAL LOAD: 0.126in

LOAD DEVIATION: 0.013in

MAXIMUM Z: 0.591in

MINIMUM Z: -0.768in

MAXIMUM SPINDLE SPEED: 3900rpm

MAXIMUM FEEDRATE: 49.449in/min

CUTTING DISTANCE: 138.441in

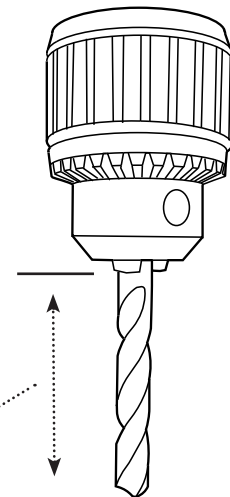
RAPID DISTANCE: 32.399in

ESTIMATED CYCLE TIME: 3m:45s (12.1%)

COOLANT: Flood

MINIMUM Z

TOOL LENGTH



INSPECT THE POCKET TOOL TABLE AND CAROUSEL

Pocket Tool Table and Carousel

- + The *Pocket Tool Table (PTT)* is a list of all tools inside the machine. It shows their location, in either a pocket or in the spindle.
- + A *pocket* is a location in the *carousel*, which holds the tool when not in use. There are 24 pockets in the carousel.
- + You must check that the PTT and the carousel are empty before starting.
 - ▶ You are checking the last person's work.
 - ▶ The *probe* (tool #100) should never be removed.

POCKET	TOOL	
SPNDL		0
1		0
2		0
3	*	0

Check the Pocket Tool Table

1. Close the doors.
2. Press **MDI DNC**. (*mode keys*)
3. Press **CURNT COMDS**. (*display keys*)
4. Press **PAGE UP** or **PAGE DOWN** (*cursor keys*) to navigate to the PTT.
5. Check that all the tool entries, except the probe tool, are set to 0 (empty).

Reading the Pocket Tool Table

- + A blank space under the Tool column means no tool is in a pocket.
- + The active pocket has an asterisk.
 - ▶ Active means it's the pocket at the bottom of the carousel.
- + Tool 100 is the probe.
- + The yellow highlight means that you have selected that pocket.
- + H or L indicates a heavy or large tool.

Heavy or Large tools

Heavy (or fragile, like the probe) tools are changed at a reduced speed.

- + Cursor to the Category column and enter **H WRITE/ENTER** to assign Heavy to the pocket.
 - ▶ Tools over 4 pounds are considered heavy.
 - ▶ All bottom drawer tools (#161-175) are heavy.

Large tools require an empty space on either side in the carousel.

- + Cursor to the Category column and enter **L WRITE/ENTER** to assign large to the pocket.
 - ▶ Tools over 3" in diameter are large.

To clear the label, enter **Space WRITE/ENTER**.

POCKET TOOL TABLE		
SPNDL	1 2 3	
1	0	
2	0	
3	*	0
4	H	100

Visually check the Carousel

You must check that the carousel matches the pocket tool table, because there is no way for the mill to know if they are in sync.

Visually verify that the carousel is empty before inserting tools. Putting a tool into an occupied space will break the tool, or worse.

Only half of the carousel can be seen at once. Check the visible half, then rotate the carousel to check the other half.

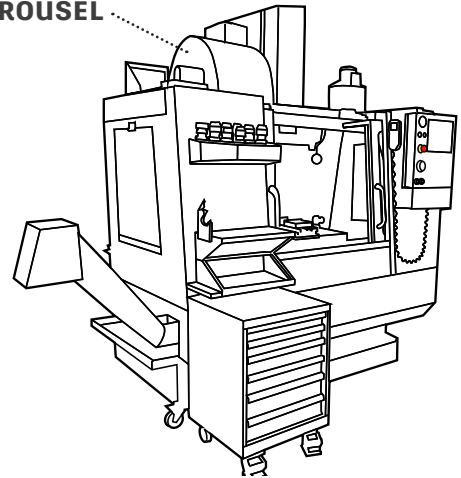
1. Visually check the carousel to see if any tools other than the probe are visible.
2. Look at the Pocket Tool Table and find the active pocket.
3. To rotate the carousel 12 positions, add 12 to the active pocket number.
 - ▶ If the answer is larger than 24, subtract 12 instead.
4. Enter **P[pocket number] ATC-FWD**.
5. Check the carousel again after it rotates.

Example: pocket 3 is active

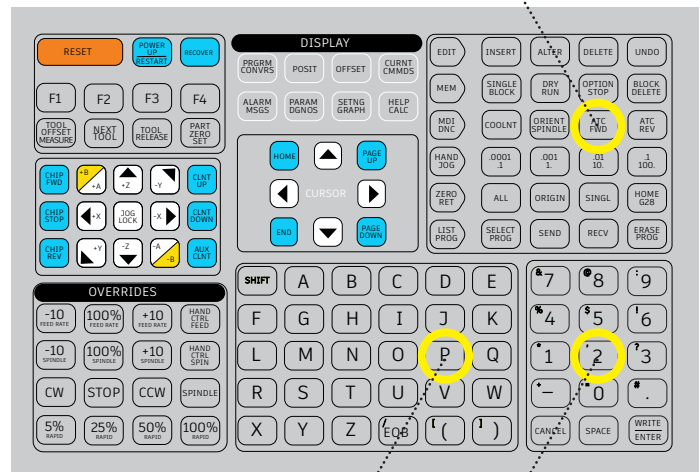
$3+12=15$

Enter: **P15 ATC-FWD**

CAROUSEL



ATC-FWD



P NUMBERS

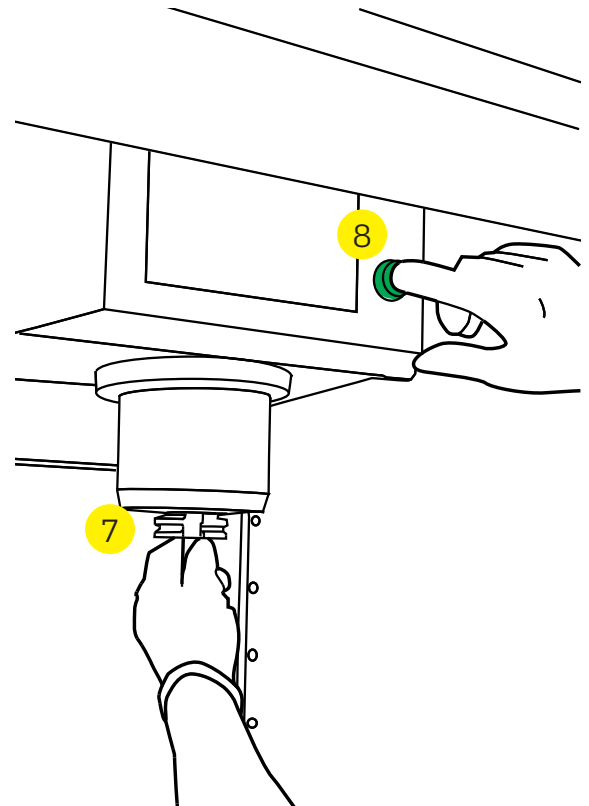
INSTALL TOOLS INTO THE MACHINE

Open the Pocket Tool Table

1. Press **MDI DNC** (mode keys)
2. Press **CURNT COMDS** (display keys)
3. Press **PAGE UP** or **PAGE DOWN** (cursor keys)

Use your Setup Sheets to help insert the tools.

1. Use the cursor keys to highlight the Tool column of an empty pocket.
2. Enter [**tool number**] **WRITE/ENTER**.
3. Press **RESET**.
4. Check that the doors are closed.
5. Press **T[tool number] ATC-FWD**.
 - ▶ The machine will perform a tool change.
6. Open the door.
7. Place the tool into the spindle.
 - ▶ Align the notches in the holder to the tabs on the spindle.
8. Press the *tool release* button.
9. Close the door.



Example - install tool #123:

1. Select the Tool column of an empty pocket.
 - ▶ You selected pocket 2, highlighted in yellow.
2. 123 WRITE/ENTER
 - ▶ You just told the machine that tool 123 is in pocket 2.
3. RESET
4. T123 ATC-FWD
 - ▶ The mill performed a tool change from pocket 2 (empty) into the spindle.
 - ▶ There's no tool in the spindle, even though it's listed in the PTT.
5. Open the door and put tool 123 into the spindle.
 - ▶ The PTT and the machine are now in sync.

POCKET TOOL

	SPNDL		
	1		0
1	2		0
	3	*	0

POCKET TOOL

	SPNDL		
	1		0
2	2		123
	3	*	0

POCKET TOOL

	SPNDL		123
	1		0
4	2	*	0
	3		0

SET THE TOOL OFFSETS

Tool Offsets are the length and diameter of the tools. Shop staff measures offsets for tool library tools with the Speroni optical system. The tool offsets are stored in the *Library Offset File*.

- + You need to import the Library Offset File each time you use the mill.
 - ▶ You do not need to measure library tools.
- + Measure the custom tools after installing them in the mill.

Import the Library Offset File

1. Press **LIST PROG**.
2. Press **CANCEL** to go up a level.
3. Use the cursor keys to navigate to the NET SHARE tab.
4. Press **WRITE/ENTER**.
5. Use the cursor keys to select the **Haas_VF2.nc** file.
6. Press **F2**.
7. Select **Copy to Memory**.
8. Press **WRITE/ENTER**.
9. Press **Y** when prompted to confirm.
 - ▶ *Disk Abort* means that the machine has been updated with the latest tool library offsets.

Custom Tool Offsets

Follow these steps to measure the tool offset for custom tools.

1. Press **MDI DNC**.
2. Press **T[tool number] ATC-FWD** to perform a tool change.
3. Press **OFFSET** (display section).
 - ▶ The OFFSET button will toggle between Work Offset & Tool Offset.
 - ▶ Be sure that Tool Offset is selected (the active window is white).
4. Use a tape measure to measure the length of the tool.
 - ▶ Measure from the tip of the tool to the base of the spindle.
 - ▶ Be within 1/4".
5. Use the cursor keys to navigate to the *Approximate Length* column.
 - ▶ Enter the tool length.
 - ▶ Press **F1** to set the value.

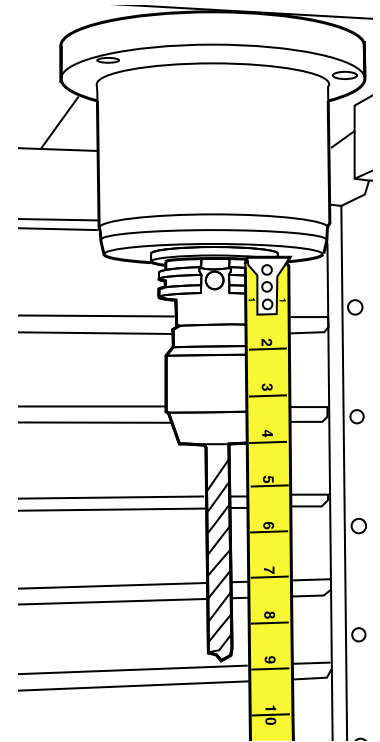
Note: Whole numbers need a decimal point, or they will be entered as ten-thousands of an inch. (8 = 0.0008 and 8. = 8)

6. Navigate to the *Flutes* field.
 - ▶ Enter the number of flutes.
 - ▶ Press **F1**.
7. Navigate to the *Tool Type* field.
 - ▶ Select the tool type.
 - ▶ Press **F1**.

If the tool is a custom end mill, follow the steps in the sidebar; go to step 8 when entering drill bits or taps.

8. Navigate to the *Probe Type* field.
 - ▶ For custom end mills, use probe type 1.
 - ▶ For drills or taps, use probe type 2.
9. Press [**probe type number**] and **WRITE/ENTER**.
10. Close the doors.
11. Press **TOOL OFFSET MEASUR** (near RESET).
12. Press **CYCLE START**
13. Navigate to the *Geometry (length)* field.
 - ▶ Compare this number to your approximate measurement.
 - ▶ If there is a large difference, stop and investigate.

Repeat these steps for each custom tool.



1. Navigate to the *Approximate Diameter* column.
 - ▶ Enter the approximate diameter of the tool.
 - ▶ Press **F1**.
2. Navigate to the *Edge Measure Height* column.
 - ▶ This is only used on end mills that don't have their full diameter at the bottom, like ball end mills.
 - ▶ Enter the distance from the bottom of the tool that the measurement should be taken.
 - ▶ Press **F1**.
3. Go to step 8.

SET THE WORK COORDINATE SYSTEM

In your CAM program, you selected a corner of your stock as the WCS. The G-code for the WCS is G54.

- + These steps will use the probe to find exactly where that point is.

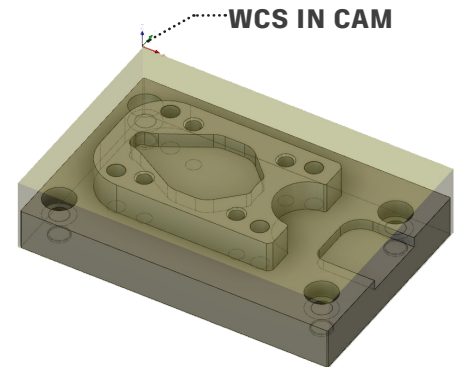
Prepare your stock

In your CAM software, you entered the approximate size of the stock. You'll need to cut the stock, then measure the actual size and update your CAM program.

1. Cut your stock to the dimensions listed on your setup sheet.
2. Measure the stock with calipers.
3. Update your setup(s) with your actual measurements.
4. Regenerate your toolpaths.
5. Generate your Setup Sheets.
6. Secure the stock in the mill.
 - ▶ Elevate your part with parallels to avoid machining the vise.
 - ▶ Use rubber mallet to seat the part firmly on the parallels.
 - ▶ Minimum clamp distance is 1/8".
7. Measure the *part stickout* with calipers.
 - ▶ Compare this to the minimum part stickout you calculated in your Setup Sheets.

Finding Z

1. Perform a tool change to put the probe (#100) in the spindle.
 - ▶ See the first two instructions on page 14.
2. Jog the tip of the probe to 1/4" above the top of your stock.
3. Press **OFFSET** until the Work Zero Offset is highlighted.
4. Use the arrow buttons to highlight **G54**.
 - ▶ Take a photo, so you can confirm that the values change.
5. Cursor to the right until the screen changes to show various work offset probing options.
 - ▶ You'll start by measuring Z only, on top of your stock.
6. Enter **11** and press **WRITE/ENTER**.
7. Navigate right to select the Work Probe Inputs.
8. Select the Z distance.
9. Enter **-0.4** and press **WRITE/ENTER**.
 - ▶ This value must be greater than the actual distance.
10. Press **CYCLE START** to run the probe.



JOGGING

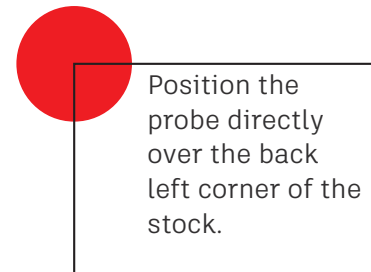
Take care while jogging the machine. It's easy to break the probe by moving too fast, too far or in the wrong direction.

- + When jogging, you'll need to set the axis (X, Y or Z) and the increment - how far the spindle or table moves for each click of the jog handwheel.
1. Press **HAND JOG** to enter jog mode.
 2. Press **.01** to set the increment (0.01" per click).
 3. In the Jog keys section, press the axis you want to move.
 - ▶ Look at the display and check that your desired axis is highlighted in yellow.
 - ▶ It does not matter if you choose + or - for this axis.
 4. Rotate the jog handwheel clockwise to move in positive direction and counterclockwise for negative.

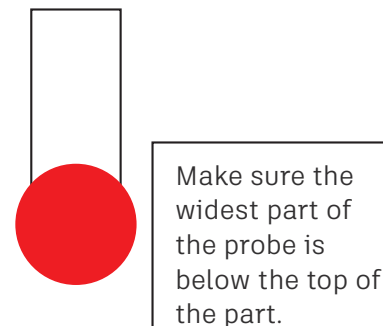
Finding X & Y

You have set your Z work coordinate; now set X & Y.

1. Jog the probe over the back left corner of the stock.
 - ▶ See image)
 2. Make the Work Zero Offset table the active window.
 3. Navigate to the Probing Options page.
 4. You want to probe the back side and left edge of the material.
 - ▶ Enter **9** and press **WRITE/ENTER** for Outer Corner.
 5. Navigate right to edit the Outer Corner value.
 6. Enter **4** and press **WRITE/ENTER**.
 - ▶ This will select the back left corner.
 7. Enter **-0.4** and press **WRITE/ENTER** for incremental Z, X & Y.
 8. PRESS **CYCLE START**.
 - ▶ Check that the full diameter of the probe contacts the material. (see image)
 9. Navigate left and check that the G54 coordinates have changed. Compare them to your photo.
 10. Look at the WCS on your setup sheet and make sure it matches the WCS in the mill.
- + A probing cycle is fairly safe. If the probe hits something unexpected, it will instantly stop, except in hand jog mode.
- + If there is a problem, the red light on top of the controller will flash and an error will display on screen.
- ▶ RESET to clear the error. Change your work probe inputs and try again.



TOP VIEW - SETUP



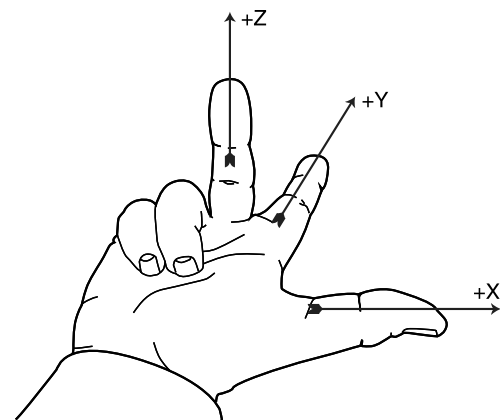
SIDE VIEW - IN USE

Right hand rule

To help remember the X, Y & Z axis, you can use the *right hand rule*.

Hold your right hand out, like in the illustration. Each finger points in the positive direction.

- + Make sure you know which way you're going to move before operating the probe.
- + Turn the handwheel very slowly to start.
- + Positive and negative directions are relative to tool motion, not table motion.



LOAD THE PROGRAM

Transfer the program from the computer to the mill.

- + The mill requires a file name of 4 or 5 numbers - no letters.
- + The file extension is .nc
- + Ensure your program numbers match your Setup Sheets.
 - ▶ Setup 1 = 1001.nc, Setup 2 = 1002.nc, etc.
- 1. Post process G-code to the thumb drive.
- 2. Insert the thumb drive into the USB port on the controller.
- 3. Press **LIST PROG** to open the Program Manager.
- 4. Press **CANCEL** to go up a level.
- 5. Select USB DEVICE and press **WRITE/ENTER**.
- 6. Navigate to your .nc program.
 - ▶ Double check the date on the file.
- 7. Press **F2**.
- 8. Select Memory and press **WRITE/ENTER** to write your program to memory.
 - ▶ If there's a name conflict, press **Y** to overwrite the old file.
- 9. Press **EDIT** (mode key) and **SETNG GRAPH** (display section).
- 10. Press **CYCLE START**.
 - ▶ This will run a graphic simulation of your program.
 - ▶ The machine will not move.
 - ▶ When the simulation is over, the status bar will briefly display M30 Found.
 - ▶ If you have an error, see Shop Staff.

PRE-RUN CHECKLIST

You're ready to run the program. Double check that these tasks have been completed.

- + Check the setup sheet to ensure all tools have been loaded.
- + Actual material size has been measured, entered into CAM and toolpaths regenerated.
- + Tool length offsets were set for custom tools.
- + Work offset (G54) has been set.
- + The clamp distance is at least 1/8", and part stickout is enough to avoid hitting the vise.
- + The program has been simulated to check for errors.

Using coolant

Because of the spindle's high speed and feed rate, flood coolant is required.

- + Behind the spindle, there are two ball valves.
 - ▶ Open the valve on the left.
 - ▶ The handle should be perpendicular to the doors.

Note: The system may be pressurized; prepare for a bit of spray.

RUN THE PROGRAM

IN MEM MODE PRESSING CYCLE START WILL START THE PROGRAM AND RESULT IN MACHINE MOVEMENT.

1. Press **MEM** to access machine memory.
2. Press **SINGLE BLOCK**.
3. Press **5% RAPID**.
4. Hold one finger over the red **FEED HOLD** button.
5. Press **CYCLE START** once.
6. Press **CURNT COMDS**.
7. Use **PAGE UP** to navigate to the Position screen.
 - ▶ The Position screen shows the current position of the tool tip and Distance To Go.
8. Press **CYCLE START** until the tool starts to move in Z towards your part.
 - ▶ Compare Work Offset G54 Z position with the actual distance in the mill.
 - ▶ If it looks like there will be a collision, press **FEED HOLD**.
 - ▶ If coolant is blocking your view, press **COOLNT** to turn it off.
 - ▶ If the tool tip stops 0.6" above your material, the Tool Offset is correct.
9. Turn the coolant back on.
10. Press **SINGLE BLOCK** to exit Single Block mode.
11. Press **CYCLE START** to start machining.
12. After the next tool change, press **SINGLE BLOCK** again.
 - ▶ Use **SINGLE BLOCK** until the tool is 0.6" above the stock.
 - ▶ Repeat for each new tool.

Single Block mode

Single Block allows only one line of the .nc file to be executed for each push of **CYCLE START**.

- + Use Single Block to step through the program and double check tool offsets.
 - ▶ Your .nc file should move the cutting tool to 0.6" above the stock and pause.
 - ▶ This will give you a chance to stop movement if something goes wrong.
- + Single Block must be turned off before actually cutting material.

FEED HOLD Button

FEED HOLD stops the table and head movement. It does not stop the spindle.

- + Use **FEED HOLD** to pause the machine for inspecting parts or cutters.

Once the machine is paused, you may want to stop coolant spray or stop the spindle.

SPINDLE STOP Button

Press **STOP** (in Overrides) to stop the spindle so you can open the door.

- + **The mill will not automatically restart the spindle when restarting the program.**
 - ▶ If you stopped the spindle you must press **CW** (clockwise) to restart the spindle before pressing **CYCLE START**.

Coolant

Press **COOLNT** (in MDI DNC) to toggle coolant on or off.

Don't remove the part yet.

When your program is finished, inspect the part before removing it from the vise. If there is a problem, you can do more machining.

- + Modify or add a toolpath.
- + Post process just that toolpath.
- + Load the new program into the machine.
- + There is no need to reset the WCS.

CLEAN UP

Remove the tools

1. Navigate to the Pocket Tool Table.
 2. Remove the tool in the spindle with the *tool release* button.
 3. Put the tool away.
 - ▶ Remove custom tools from holders.
 4. Put a 0 in the Pocket Tool table.
 5. Remove any notes for heavy or large tools.
- Repeat for all tools except the probe.

Double check your work

- + Check the Pocket Tool table for all 0s, except the probe.
- + Check the carousel for tools.

Hose out the mill

1. Close the coolant valve by the spindle.
2. Turn on coolant at the control panel.
 - ▶ This will pressurize the hose.
3. Turn on the auger.
4. Wash the chips off the bed, vise, walls and windows.
 - ▶ Jog the bed and spray under it.

Final Steps

1. Turn off the coolant.
 - ▶ Release pressure on the hose.
2. Turn off the auger.
3. Close the doors.
4. Turn off the mill - **E-stop** then **POWER OFF**.
5. Clean the area, including tables and the floor.
6. Make sure all the tools are put away and custom tools are removed from the holders.

CHIP AUGER

Pressing **CHP FWD** in the Jog panel will run an auger to move chips into the recycle bin.

- + The auger is just inside the mill, under the door.
- + Make sure you do not get caught in the auger when reaching inside the machine.

USING COMPRESSED AIR

Compressed air forces chips into the moving parts of the mill and can cause it to lose accuracy.

- + Use compressed air to dry parts and parallels, outside the mill.
- + Do not use it to clean the mill.

PREPARATION

1. Remember that most of the work will happen before you get to the mill.
2. Be alert, and minimize distractions.
3. If you are unsure of any steps, start that section over.
 - ▶ Ask Shop Staff for help if needed.

REGENERATE TOOL PATHS AFTER MEASURING THE ACTUAL STOCK.

USING THE MILL

1. Start up the mill. (*page 7*)
2. Insert custom tools into the holders. (*page 8*)
3. Inspect the Pocket Tool Table and Carousel. (*page 10*)
4. Install tools into the machine. (*page 12*)
5. Set the tool offsets. (*page 13*)
6. Set the Work Coordinate System. (*page 15*)
7. Load the program. (*page 17*)
8. Run the program. (*page 18*)

ALWAYS RUN IN 5% RAPID & USE SINGLE BLOCK TO CONFIRM TOOL OFFSET.

CLEANUP

1. Remove tool holders from the machine.
 - ▶ Put away library tools.
 - ▶ Remove and put away custom tools.
2. Clear the Pocket Tool table.
3. Turn on the auger & coolant.
4. Spray the inside of the mill until all the chips are gone and the mill is clean inside.
5. Turn off the coolant & auger.
6. Close the doors.
7. Turn off the mill.
8. Clean tables and the floor.
9. Put away all tools.

BE CAREFUL WITH THE CHIP AUGER.