Haas Mini Mill & Super Mini Mill 2

How to start, set up, run and look after the two Haas vertical mills in the shop.

Shop guide Draft, November 2026

1 About this guide
2 Safety rules
3 Know the machine
4 Student operating steps
5 Fusion to machine
6 Hand-written G-code
7 Quick cards
8 Instructor maintenance
9 To confirm in the shop
10 Sources

About this guide

This guide covers the two Haas vertical mills in the shop: the **Mini Mill** and the **Super Mini Mill 2**. Both use the same Haas control, so one set of steps works on either machine. Figures below are Haas catalog values for the base models; confirm them against each machine's serial plate.

	Mini Mill	Super Mini Mill 2
Travels X × Y × Z	16" × 12" × 10"	20" × 16" × 14"
Spindle	40 taper, 6,000 rpm	40 taper, 10,000 rpm
Tool changer	10-tool umbrella (standard)	Confirm on machine
Shop extras seen in photos	Tool cart with offset tags	Tool cart with offset tags, probe

Who reads what:

- **Students:** Safety rules, Know the machine, Student operating steps, and the Quick cards.
- **Students writing programs:** Fusion to machine, and Hand-written G-code.
- **Instructor:** Instructor maintenance, and the To confirm list at the end.

The steps assume the Haas Next Generation Control (NGC). The 2018 decals on the machines suggest NGC, but this needs a look at the screen to confirm.

Safety rules

No student runs either mill until the instructor has signed them off on this section. These are draft shop rules; edit them to match school policy.

Before you touch the machine

- Safety glasses on. Long hair tied back. No gloves, loose sleeves, lanyards, rings or dangling jewelry.
- Closed-toe shoes. No headphones.
- One operator at the control. Everyone else stays behind the operator.
- Know where the red EMERGENCY STOP button is before you press anything else.

While the machine runs

- Doors closed whenever the spindle turns or a program runs. Never defeat a door interlock.
- Never reach into the enclosure while anything is moving. Wait for the spindle to stop completely.
- Stay at the control with a hand near FEED HOLD for the whole program. Never leave a running machine.
- If something sounds, looks or smells wrong, press FEED HOLD. If there is danger to a person or the machine, press EMERGENCY STOP. Then get the instructor.

Tools, chips and coolant

- Cutting tools are sharp even when still. Hold a tool holder by its body, not by the cutter. Use a rag for large cutters.
- Never clear chips by hand or with compressed air aimed at yourself or others. Use a brush or the chip hook.
- Coolant on the floor is a slip hazard. Wipe it up straight away.
- Only machine materials the instructor has approved. The decal on the machine prohibits ceramics; ask before cutting anything other than aluminum, steel, brass or machinable plastic.

Which stop to use

Button	What it does	When to use it
FEED HOLD	Stops axis motion; the spindle keeps turning. CYCLE START resumes.	Anything you want to pause and check
RESET	Stops the program, spindle and coolant; clears alarms	The run is over or wrong and you want to start again
EMERGENCY STOP	Stops all motion and shuts off the servos	Danger to a person or an imminent crash

Know the machine

A vertical mill moves a part under a spinning cutter along three axes. Stand at the control, facing the doors:

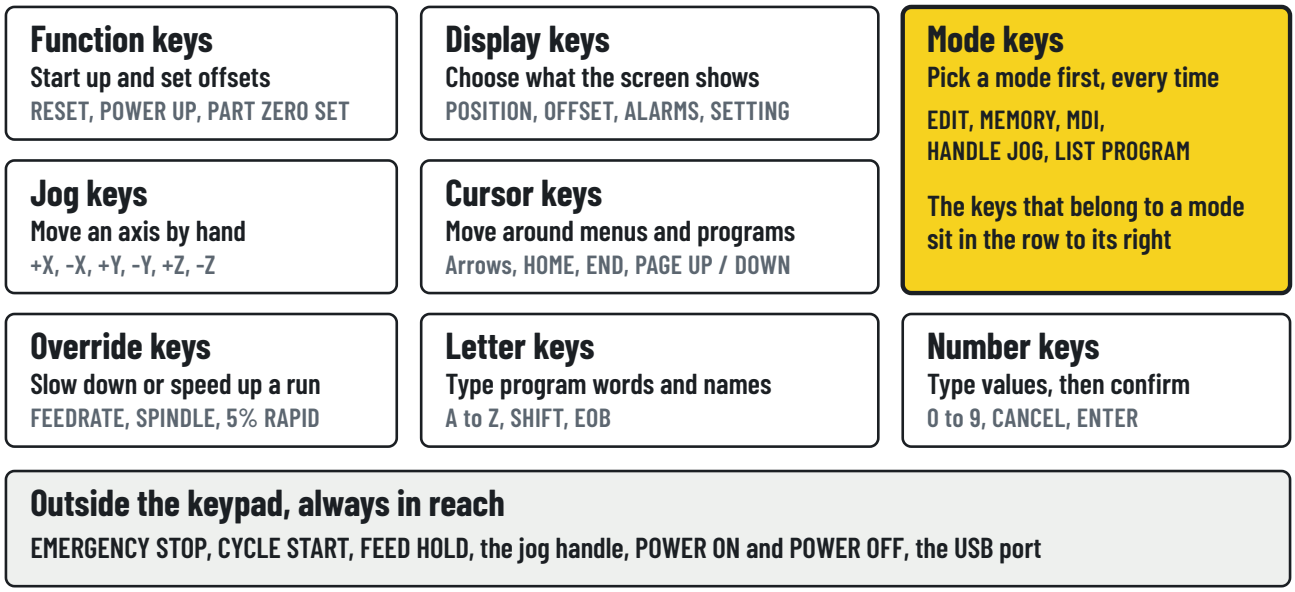
Axis	What moves	Positive direction, as the tool sees the part
X	Table, left and right	Tool moves right across the part
Y	Table, toward and away from you	Tool moves toward the back of the part
Z	Spindle head, up and down	Tool moves up, away from the part

Two zero points matter. Machine zero (home) is fixed by the machine and found at power-up. Part zero is a point you choose on your part; the machine stores it as work offset **G54**. Your Fusion setup and the machine must agree on where part zero is.

Two offsets make a program run in the right place. The work offset (G54) tells the machine where the part is in X and Y. The tool length offset (H number) tells it how long each tool is, so every tool reaches the same Z height.

The control panel at a glance

The panel has well over a hundred keys, but they sort into eight groups, and a first job uses about two dozen of them. Learn the groups before the keys.



A schematic of the Haas keypad. Positions are approximate; check them against the real control.

Read it in this order: choose a **mode** on the right, use the **display** keys to see what you need, then act with the function, jog or number keys. If a key seems to do nothing, the machine is usually in the wrong mode.

Three habits make the panel manageable:

- **Mode first.** HANDLE JOG to move by hand, MDI for a one-line command, MEMORY to run a program, EDIT to change one, LIST PROGRAM to choose one.
- **The screen tells you the mode.** The current mode is shown at the top left of the display. Check it before pressing CYCLE START.
- **Ignore the rest for now.** Keys not named in this guide are for features you will not use in your first projects.

Keys you will use

Group	Keys	What they are for
Power	POWER ON, POWER OFF, EMERGENCY STOP	Starting and stopping the machine
Start-up	RESET, POWER UP	Clearing alarms; homing all axes
Modes	HANDLE JOG, MDI, MEMORY, EDIT, LIST PROGRAM	Moving by hand; one-line commands; running, editing and choosing programs
Displays	POSITION, OFFSET, CURRENT COMMANDS, ALARMS	Where the machine is; offsets; tool table; alarm text
Setup	TOOL OFFSET MEASURE, NEXT TOOL, TOOL RELEASE, PART ZERO SET, ATC FWD, ATC REV	Measuring tools, loading tools, setting part zero
Running	CYCLE START, FEED HOLD, GRAPHICS, SINGLE BLOCK, 5% RAPID, 25% RAPID, FEEDRATE −10% / +10%	Starting, dry-running on screen, pausing and slowing a program

Tool carts and yellow tags

Each mill has a cart of tool holders. A yellow tag under each holder gives its tool number, a description and an **H** value, for example a 5/16" drill tagged T14 with H 6.1037. The Super Mini Mill 2 cart also holds a probe.

- A tool always goes back in its own tagged slot.
- The tool number on the tag must match the tool number in your program.
- Never swap a cutter in a holder without telling the instructor. The H value on the tag is then wrong.

Student operating steps

Every job follows the same eight stages, in this order. Key names are in capitals as printed on the control.

1. Power up

1. Check the area: doors closed, nothing loose on the table, no tools left inside.
2. Check that shop air is on. A low-air icon on the screen means the machine will not run until pressure is restored.
3. Press `POWER ON` and wait for the control to finish starting.
4. Turn the `EMERGENCY STOP` button clockwise to release it, then press `RESET` to clear alarms.
5. Open and close the door once if the screen shows a cycle-door icon.
6. Check the table and spindle have a clear path, then press `POWER UP`. All axes move to home straight away. Stand clear.
7. At the start of every school day, run the 20-minute warm-up program **O09220** before anyone cuts. School policy is a daily warm-up, whatever the machine did the day before.

2. Hold the part

1. Wipe the vise jaws, the parallels and the bottom of your stock. One chip under the part ruins the cut.
2. Seat the stock on parallels with enough material above the jaws for your deepest cut.
3. Tighten the vise firmly and tap the stock down with a soft mallet. Remove the vise handle.
4. Check that the stock size and orientation match your Fusion setup.

3. Load tools

1. Collect only the tools your program calls, from their tagged cart slots.
2. Press `MDI`. Type `T` and the tool number, for example T3, then press `ATC FWD`. The machine brings that pocket to the spindle.
3. Hold the tool by the holder body. Line up the two slots in the holder flange with the drive keys on the spindle.
4. Press and hold `TOOL RELEASE`, push the holder up into the spindle, then let go of the button. Tug down gently to confirm it is clamped before you let go of the holder.
5. Repeat for each tool.

The Super Mini Mill 2 may have a side-mount tool changer, which also needs each tool entered in the tool table under `CURRENT COMMANDS`. The instructor will show which changer each mill has.

4. Set tool length offsets

The H value for each tool must be in the control before you run. In this shop the value may come from the yellow tag or the probe; ask the instructor which. The manual touch-off method is:

1. Bring the tool into the spindle. Press `HANDLE JOG`, then `.1` for fast jogging.
2. Jog X and Y until the tool is over the middle of the part. Jog Z down to about 1" above it.
3. Press `.0001` for slow jogging. Hold a slip of paper flat on the part and lower the tool until the paper just drags.
4. Press `OFFSET` and choose the **TOOL** tab. Highlight the **H (length) Geometry** value on that tool's row.
5. Press `TOOL OFFSET MEASURE`. Check the number changed. Press `Y` if the control asks you to confirm a large change.
6. Press `NEXT TOOL`. The spindle moves up quickly and changes tools. Repeat for each tool.

5. Set part zero (G54)

1. Load the pointer or edge finder the instructor specifies.
2. Press `HANDLE JOG`, then `.1`. Jog Z to about 1" above the part, then switch to `.001` and come down to about 0.2".
3. Jog X and Y to your part zero. Use the same corner you picked in Fusion.
4. Press `OFFSET` and choose the **WORK** tab. Move the cursor to the **G54 X** value.
5. Press `PART ZERO SET` once to store X. Press it a second time to store Y.
6. **Stop there.** A third press writes a Z value into G54 and can crash the tool into the part.

6. Load the program

This is the sequence posted on the Super Mini Mill 2.

1. Plug your USB drive into the port on the side of the control.
2. Press `LIST PROGRAM`. Arrow across to the **USB** tab (or `MEMORY` if the program is already on the machine).
3. Arrow down to your file and press `SELECT PROGRAM`. The active program is marked with an asterisk.
4. Press `MEMORY`. Check that the program name at the top of the screen is yours.

To keep a copy on the machine instead: highlight the file on the USB tab, press `ENTER` to tick it, press `F2`, choose **Memory**, and press `ENTER`. Only .nc files copy to memory.

7. Prove it out

No program cuts metal until it has passed both checks.

On screen

1. In MEMORY mode press GRAPHICS, then CYCLE START. The program is drawn on screen and nothing moves.
2. Check the shape, the size and the depth against your drawing. Press F2 to zoom.
3. Fix any alarm before going further. Get the instructor's OK.

First real run

1. Press MEMORY to leave graphics. Close the doors.
2. Press 5% RAPID and turn on SINGLE BLOCK.
3. Press POSITION and choose the **DIST TO GO** tab. It shows how far the machine will travel on the current line.
4. Press CYCLE START once per line. Keep your other hand on FEED HOLD.
5. Watch the first approach to the part. If the tool is 1" above the part and Dist To Go for Z reads more than 1", press FEED HOLD and get help.
6. Once the first tool is cutting correctly, turn off SINGLE BLOCK. Raise rapids to 25% only when the instructor agrees.

8. Finish and shut down

1. Wait for the spindle to stop and the program to end before opening the doors.
2. Remove the part. Deburr sharp edges at the bench, not in the machine.
3. Brush chips off the vise and table into the chip pan. Wipe the vise jaws.
4. Return every tool holder to its tagged cart slot.
5. Remove your USB drive. Tell the instructor about any alarm, odd noise or broken tool.
6. Press POWER OFF only if the instructor asks. Otherwise leave the machine homed with doors closed.

Fusion to machine

A Fusion program runs correctly only when four things match the machine: part zero, tool numbers, units and the post processor.

In Fusion (Manufacture workspace)

1. **Setup.** Set the stock to the real size of your material. Put the work coordinate origin on the corner and face where you will set G54 at the machine. Top of stock, back-left corner is the usual shop choice.
2. **Tools.** Pick tools from the shop tool library so each tool number matches its yellow tag. Never invent a tool number.
3. **Toolpaths.** Keep clearance and retract heights well above the vise and clamps.
4. **Simulate.** Run the simulation with stock shown. Fix every red collision mark before posting.
5. **Post process.** Choose the post that matches the control:

Control on the machine	Fusion post to use	Output file
Next Generation Control	HAAS - Next Generation Control	.nc
Older (classic) control	HAAS (pre-NGC)	.nc

Give the program a number and a short name with your initials. Save the .nc file to your USB drive.

Before you walk to the machine

Open the .nc file in a text editor and check five things:

- ☐ The first line after % is an O-number, such as O01234. The control requires one.
- ☐ Units are inches (G20) unless the instructor says otherwise.
- ☐ The work offset is G54.
- ☐ Every T number and H number matches a tool you will load.
- ☐ The spindle speeds are within the machine's limit: 6,000 rpm on the Mini Mill, 10,000 rpm on the Super Mini Mill 2.

At the machine

Follow stages 6 and 7 of the operating steps: load from USB, run in graphics, then single-block the first run at 5% rapid.

Common mismatches

What you see	Likely cause
Graphics shows the part far off the table	Fusion origin is not where G54 was set
Tool stops well above the part, or dives toward it	Wrong H value, or the wrong tool in that pocket
Alarm as soon as the program starts	Wrong post, or a tool number the machine does not have
Part is 25 times too big or too small	Inch and metric mixed up
Rapids look slow and steady in a pre-NGC program	That post outputs positioning moves as fast feed moves by default

Hand-written G-code

A mill program is a list of lines the machine reads top to bottom: set up safely, pick a tool, move, cut, retract, stop. Reading G-code lets you spot a bad Fusion program before it cuts.

Starter program

This cuts one straight slot 0.050" deep along X with a 1/2" end mill. Part zero is the top, back-left corner of the stock. It is a teaching example with gentle numbers for aluminum; prove it out in graphics like any other program.

```
%
O01001 (SLOT - YOUR INITIALS)
(T1 = 1/2 END MILL)
G20 G17 G40 G49 G80 G90 (SAFE START: INCH, XY PLANE, CANCEL COMP AND CYCLES, ABSOLUTE)
T1 M06 (LOAD TOOL 1)
G00 G54 X-0.5 Y-1.0 (RAPID TO START, LEFT OF THE PART)
S3000 M03 (SPINDLE ON CLOCKWISE, 3000 RPM)
G43 H01 Z1.0 (APPLY TOOL 1 LENGTH, STOP 1 INCH ABOVE PART)
M08 (COOLANT ON)
G00 Z0.1 (RAPID CLOSE TO THE SURFACE)
G01 Z-0.05 F10. (FEED DOWN TO DEPTH)
G01 X3.5 F20. (CUT THE SLOT)
G00 Z1.0 (RETRACT)
M09 (COOLANT OFF)
M05 (SPINDLE STOP)
G28 G91 Z0. (Z TO MACHINE HOME)
G28 G91 Y0. (TABLE FORWARD TO UNLOAD)
G90
M30 (END AND REWIND)
%
```

Codes worth knowing

Code	Meaning
G00	Rapid move, not for cutting
G01	Straight cutting move at feed rate F
G02, G03	Arc clockwise, arc counter-clockwise
G20, G21	Inch, metric
G28	Return to machine home
G40, G41, G42	Cutter compensation off, left, right
G43 H__	Apply the length offset for that tool

Code	Meaning
G54	Use the first work offset (part zero)
G80, G81, G83	Cancel drill cycle, drill, peck drill
G90, G91	Absolute positions, incremental moves
M03, M05	Spindle on clockwise, spindle stop
M06	Tool change
M08, M09	Coolant on, off
M00, M01	Stop, optional stop
M30	End of program and rewind

Writing and editing at the control

- **New program:** press LIST PROGRAM, then INSERT. Enter an O-number and, if you like, a file name and comment.
- **Edit:** select the program, press EDIT, type a change, then use INSERT to add it, ALTER to replace the highlighted word, DELETE to remove it and UNDO to step back.
- **One-off commands:** press MDI, type a line such as S1000 M03, then press CYCLE START. Nothing typed in MDI is saved as a program.
- **VPS templates:** the control's Visual Programming System builds common features, such as engraving or bolt circles, from a fill-in form.

Decimal points matter. On a Haas, X2. means two inches; X2 without the point may be read as a much smaller move, depending on a machine setting. Always type the decimal point.

Quick cards

Four cards to print and post at each mill. Each is a short reminder for a student who has already been trained on the full steps above.

Card 1: Start-up

1. Glasses on, hair back, no gloves.
2. Doors closed, table clear, air on.
3. `POWER ON`. Release `EMERGENCY STOP`. Press `RESET`.
4. Cycle the door if asked.
5. Path clear? Press `POWER UP` and stand back.
6. First thing every day: run warm-up **O09220**.

Haas Mini Mill & Super Mini Mill 2

Card 2: Set up

1. Clean the vise. Seat the stock on parallels. Tighten. Handle off.
2. Load each tool: `MDI`, type T and the number, `ATC FWD`, then `TOOL RELEASE` to seat it.
3. Tool lengths: `OFFSET`, **TOOL** tab, `TOOL OFFSET MEASURE`, `NEXT TOOL`.
4. Part zero: `OFFSET`, **WORK** tab, cursor on G54 X, `PART ZERO SET` twice. Never three times.

Haas Mini Mill & Super Mini Mill 2

Card 3: Load and run

1. LIST PROGRAM. Arrow to **USB**. Arrow to your file.
2. SELECT PROGRAM, then MEMORY.
3. GRAPHICS, then CYCLE START. Check the picture. Get the instructor's OK.
4. MEMORY again. Doors closed.
5. 5% RAPID and SINGLE BLOCK on. Watch Dist To Go.
6. CYCLE START with a hand on FEED HOLD.

Haas Mini Mill & Super Mini Mill 2

Card 4: Stop and clean up

If	Then
You want to pause and look	FEED HOLD
The run is wrong	RESET, then get the instructor
Danger to a person or a crash is coming	EMERGENCY STOP
An alarm appears	Do not clear it. Read it, write it down, get the instructor

When finished: spindle stopped, part out, chips brushed into the pan, vise wiped, tools back in tagged slots, USB out.

Haas Mini Mill & Super Mini Mill 2

Instructor maintenance

Most of the upkeep is a two-minute daily look and a weekly check of coolant and air; the rest is monthly or slower. Intervals below follow Haas's VMC Maintenance Schedule. The Haas site would not load the full page while this was written, so the table was assembled from search excerpts of it. Check it against the live page before treating it as final.

Schedule

Interval	Task	System
Daily	Inspect the windows for damage	Enclosure
Daily	Check the door interlock works	Enclosure
Weekly	Clear chips from the tool changer	Tool changer
Weekly	Check coolant level	Coolant
Weekly	Check coolant concentration with a refractometer and log it	Coolant
Weekly	Check the spindle air regulator pressure	Pneumatics
Monthly	Check the axis lubrication reservoir and refill	Axis lubrication
Monthly	Check the spindle lubrication tank level	Spindle lubrication
Monthly	Clean the vector drive air vents and filter	Electrical cabinet
Monthly	Inspect and lubricate the way covers	Enclosure
Monthly	Umbrella changer: inspect motors and sensors; grease guide rails and extractor fingers	Tool changer
Quarterly	Side-mount changer: check cambox oil level	Tool changer
Six months	Drain and clean the coolant tank; replace coolant; clean the coolant filter	Coolant
Six months	Run the axis and spindle lubrication tests	Lubrication
Six months	Inspect air and lubrication hoses for cracks	Pneumatics, lubrication
Six months	Probe: check batteries and calibration	Probe
Six months	Side-mount changer: inspect arm plunger; clean and lubricate pocket plungers	Tool changer

Interval	Task	System
Annually	Measure spindle drawbar force	Spindle
Annually	Clean the air purge solenoid	Pneumatics
Annually	Side-mount changer: grease cams; replace worn pocket parts	Tool changer
As required	Clean the coolant filter screen; clean and lightly oil the spindle taper; grease pull studs	Coolant, spindle, tooling

In a student shop it is worth doing the weekly coolant and chip items daily, since many hands use the machine.

Daily walk-round for a student shop

These are shop practice, not Haas requirements.

- ☐ Air pressure normal; no low-air icon on the control.
- ☐ Coolant level adequate; no coolant on the floor.
- ☐ Chips cleared from the vise, table and tool changer.
- ☐ Every cart slot holds the holder its tag names; no chipped or loose cutters.
- ☐ No active alarms. Press `ALARMS` to read any that are showing.
- ☐ Windows and door interlocks intact.

When something stops the machine

Condition	First things to check
Low-air icon or alarm	Compressor running, shop air valve open, regulator pressure at the machine
Machine will not start a program	Doors closed, EMERGENCY STOP released, RESET pressed, axes homed with POWER UP
Tool change stopped part-way	Do not force the carousel. Read the alarm, then follow the tool changer recovery prompts on the control
Over-travel or travel-limit alarm	Part zero or the Fusion origin is wrong; the program is asking for a move outside the machine's travels
Tool cuts air or cuts too deep	Wrong H value for that tool, or the tool was changed in its holder
Poor finish, squealing	Dull or chipped cutter, part loose in the vise, speeds and feeds too high

Press ALARMS for the alarm number and text, and look the number up in Haas's online alarm reference. For anything electrical, inside the cabinet, or involving the spindle or drawbar, call the Haas Factory Outlet rather than opening the machine.

To confirm in the shop

Eight details could not be read from the photos. Each one changes a line in this guide.

- [ ] Control generation on each mill: Next Generation Control or classic. This decides the Fusion post and some key names.
- [ ] Tool changer type and pocket count on each mill: umbrella or side-mount.
- [ ] What the H values on the yellow tags are measured from. Values of 3" to 6" look like gauge lengths from a probe or tool setter. If so, part Z zero is set differently from the paper method in stage 4.
- [ ] Whether students set offsets themselves, or the instructor does.
- [ ] The pencilled note beside Graphics and Cycle Start on the posted program-loading sheet. It was not legible in the photo.
- [ ] Whether students run from USB or copy programs to machine memory first.
- [ ] Standard part zero for the shop, and whether a Fusion tool library matching the tags exists.
- [ ] Coolant brand and target concentration, and the spindle taper and pull-stud type for each mill.

Sources

Pages opened and read for the procedures:

- Haas Mill Operator's Manual, chapter 8, Part Setup: jogging, tool offsets, work offsets.
- Haas Mill Operator class manual, Productivity Inc. (Haas Factory Outlet), 2025: power-up, warm-up, device manager, graphics, overrides, tool loading.
- Haas Next Generation Control guide: copying and selecting programs, O-numbers and file names.

Pages found by search but not opened in full; details taken from search excerpts and due a re-check:

- Haas VMC Maintenance Schedule, RD0080: maintenance intervals.
- Haas Mini Mill / Super Mini Mill service manual: Mini Mill travels and spindle.
- Autodesk post library, Haas posts: post names for each control.

The safety rules, the starter program, the daily walk-round and the troubleshooting table are general machining practice written for this guide, not taken from a Haas document.