Haas Super Mini Mill

8/26/22 Haley Holcomb haleyholcomb@ucsb.edu

Haas Super Mini Mill

Tool Type: “CNC”
Location: “mFL 3430”

Supervisor	**Tool Lead**
David Bothman	David Bothman
(805) 893-4125	(805) 893-4125
bothman@cnsi.ucsb.edu	bothman@cnsi.ucsb.edu

Description: CNC milling machine
Manufacturer: Haas

About

The Haas Super Mini Mill is a CNC milling machine. Parts can be modeled in SOLIDWORKS and then toolpaths are programmed through an HSM add-in that generates G-code for the machine to read. The Haas can read G-code and .EGC files.

Safety Concerns

- Safety Glasses must be worn when machine is running
- The glass doors must be in closed before starting a job
- The spindle must be completely stopped before adjusting or removing work piece or tool
- After milling is completed, the work piece will have developed sharp burrs which must be removed with a hand file or sandpaper
- Proof your program before running it. The first time you run it, slow the rapid speed and stay by the control panel. It is a good idea to use single block mode the first time you run your program.

Training Documentation

Haas Super Mini Mill Training SOP

Haas mill setup checklist

Detailed Specifications

406 x 305 x 254 mm, 40 taper, 15 hp vector drive, 10,000 rpm, 30.5 m/min rapids, high-speed 10-station automatic tool changer, coolant pump, power-failure detection module, 1 GB program memory, 15″ colour LCD monitor, USB port, memory lock keyswitch, and rigid tapping. Three-phase power only.

- X-axis: 16“
- Y-axis: 12”
- Z-axis: 10“
- Spindle nose to table: 14” max 4“ min
- Table length: 36”
- Work area length: 28.75“
- Width: 12”
- T-slot width: 12“
- T-slot center distance: 4.33”
- # T-slots: 3
- Max load on table: 500 lb
- Max spindle rating: 15 hp
- Max speed: 10,000 rpm
- Max torque: 17 ft-lb @ 4600 rpm
- Taper: 40
- Rapids: 1200 in/min
- Max cutting: 833 in/min
- Max thrust: 2000 lb
- Max tool diameter: 3.5“

Operating Procedures

1. Program tool paths using a SOLIDWORKS add-in
2. Put tools in tool holders
3. Set tool offsets using tool probe
4. Set fixture offset(s) with touch probe (usually G54)
5. Load program
6. Check that tool numbers in program match tools in machine
7. Check that appropriate fixture offset is loaded

8. Load stock in the machine
9. Proof the part (test the program with air cuts). One easy way to do this is to add a height to the G54 Z value sufficient to make sure that all cuts will be above the top of the part surface.
10. Use rapid and feed override at the beginning of the program to make sure that the initial Z axis moves are safe. You can also use single block mode
11. Keep your hand close to the emergency stop in case the program doesn't do what you expect.
12. Reset G54 to the correct value.
13. Run the program for the first time - follow the precautions in step 10 to make sure that nothing unexpected happens. You can adjust the feed rate override as needed.

Reference Documentation

Pre Install Manual

Programming Manual

Probe Training Manual

Operators Manual

Programming Workbook

RS232 file transfer documentation

Haas / HSMWorks Intro Text
