Welcome to Trident Machine Tools

Haas Mill Setup & Operation

Haas Mill Setup & Operation

- This half day course is designed to familiarize the user with the basic set-up and operation of a Haas CNC mill. Machine settings, tool holders, machine offsets and tool presetters and spindle probes will be covered during the course.

Schedule

- Introductions
- Classroom:
  - Machine components
  - Control settings
  - Tool holders
  - Height offsets
  - Work offsets
- Break
- Showroom
  - Set-up Demo
  - Tool setter/ spindle probe demo

Haas Mill Overview

- This tutorial is designed to train the user with skills necessary to operate a Haas CNC mill.
- This course will cover:
  - Machine Safety
  - Maintenance
  - Machine Components
  - Setup
  - How it works
  - Optional features
  - Trouble shooting

Machine Safety

- When working with heavy machinery safety comes first. Take the time to read through these precautions before operating a mill.

Machine Safety

- Safe Machine Operating Practices
  - Keep guard doors closed while the machine is in operation
  - Never leave the machine unattended while it is in operation
  - Do not press a button unless you are certain of what the machine will do when the button is pressed
  - Locate all of the emergency stop buttons on the machine
  - Do not open electrical panels or control doors
  - Never bypass any safety features on the machine

Machine Safety

- Safe Machine Operating Practices Cont.
  - Be sure the work piece is being held properly and that the set-up is correct
  - Always prove out new programs with a dry run
  - Dry run should run 2 inches above the part
  - Use the Single Block feature during the dry run
  - Use the distance to go feature as the tool approaches the work piece to avoid any collisions
  - Do not leave tools, work pieces or other items loose on the machine table.
  - Never remove chips with your hands
  - Keep skin contact with cutting fluids to a minimum

Machine Components

VF2

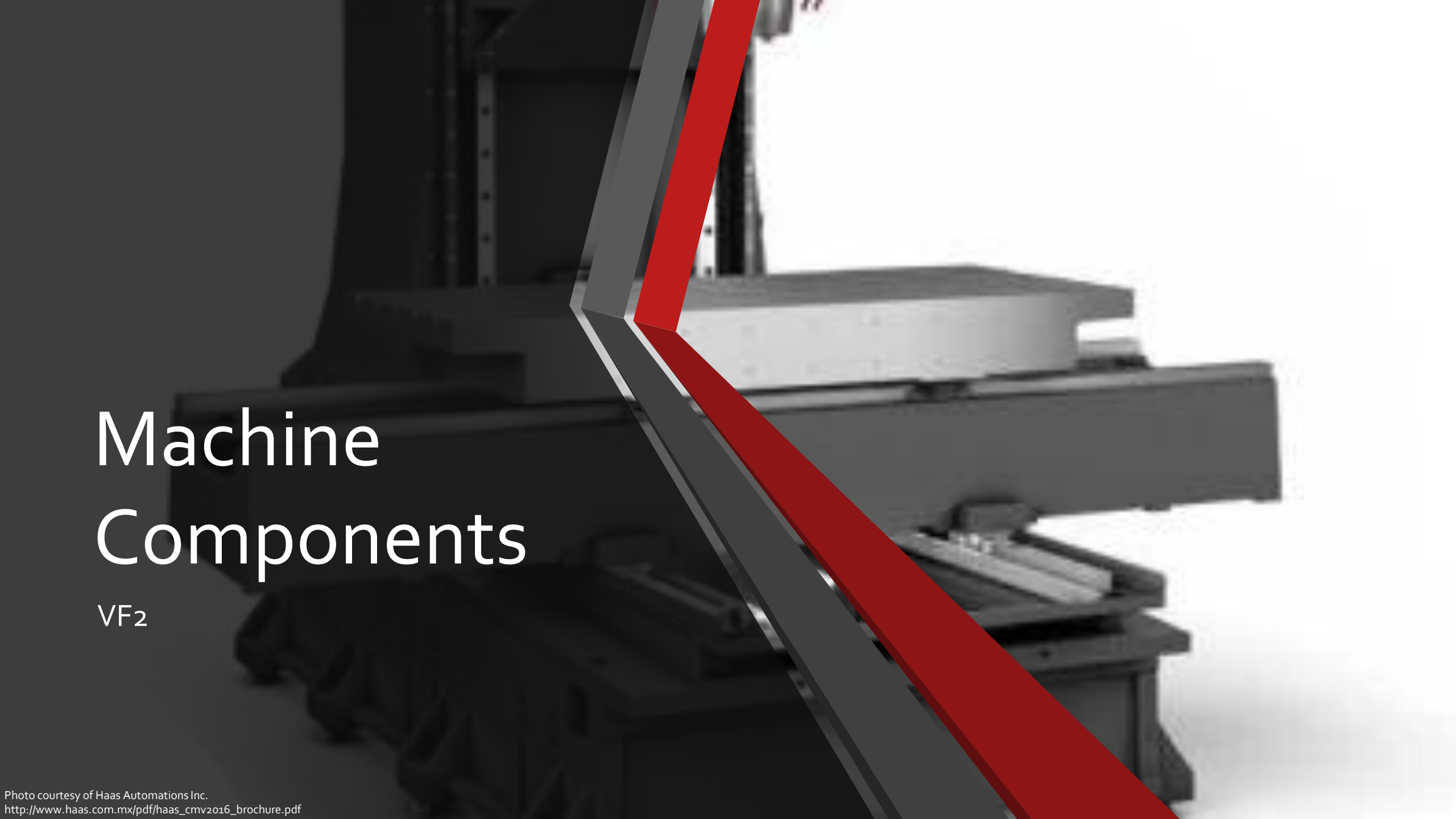


Photo courtesy of Haas Automations Inc.
http://www.haas.com.mx/pdf/haas_cmv2016_brochure.pdf

Machine Components

- Major components of a CNC milling machine
  - Frame
    - Bed
    - Column
    - These two parts are where all other components of the machine are bolted to. The machine is only as durable as the frame

Photo curtesy of Haas Automations Inc.
http://www.haas.com.mx/pdf/haas_cmv2016_brochure.pdf

Machine Components

- Major components of a CNC milling machine include:
  - Headstock
    - Spindle
    - Tool
    - Z axis ball screw
    - Z axis servo motor
  - The headstock locates the spindle components, which are used to rotate the cutting tool.

Photo curtesy of Haas Automations Inc.
http://www.haas.com.mx/pdf/haas_cmv2016_brochure.pdf

Machine Components

- Major components of a CNC milling machine
  - Table
  - Way covers

  The table is the work area for the machine. The way covers are used to protect the ball screws, servo motors, and electrical wiring from the elements of the machining process

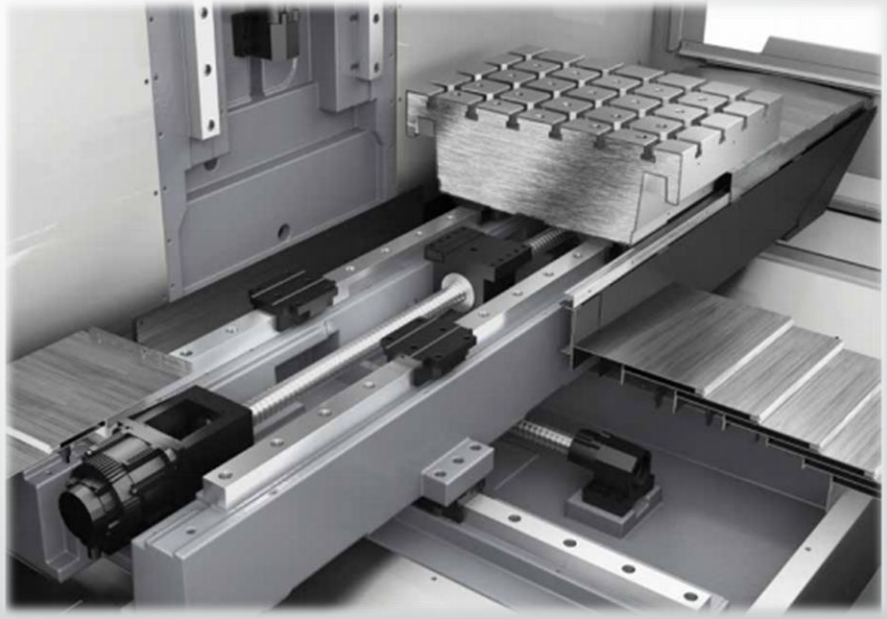


Photo curtesy of Haas Automations Inc.
http://www.haas.com.mx/pdf/haas_cmv2016_brochure.pdf

Machine Components

- Major components of a CNC milling machine
  - Automatic Tool changer (ATC)
  - Tool Storage Magazine

  The ATC and Magazine allow for the storage and use of multiple tools to complete a job. This also allows for uninterrupted operation of the machine in automatic mode

ATC

Photo curtesy of Haas Automations Inc.
http://www.haas.com.mx/pdf/haas_cmv2016_brochure.pdf

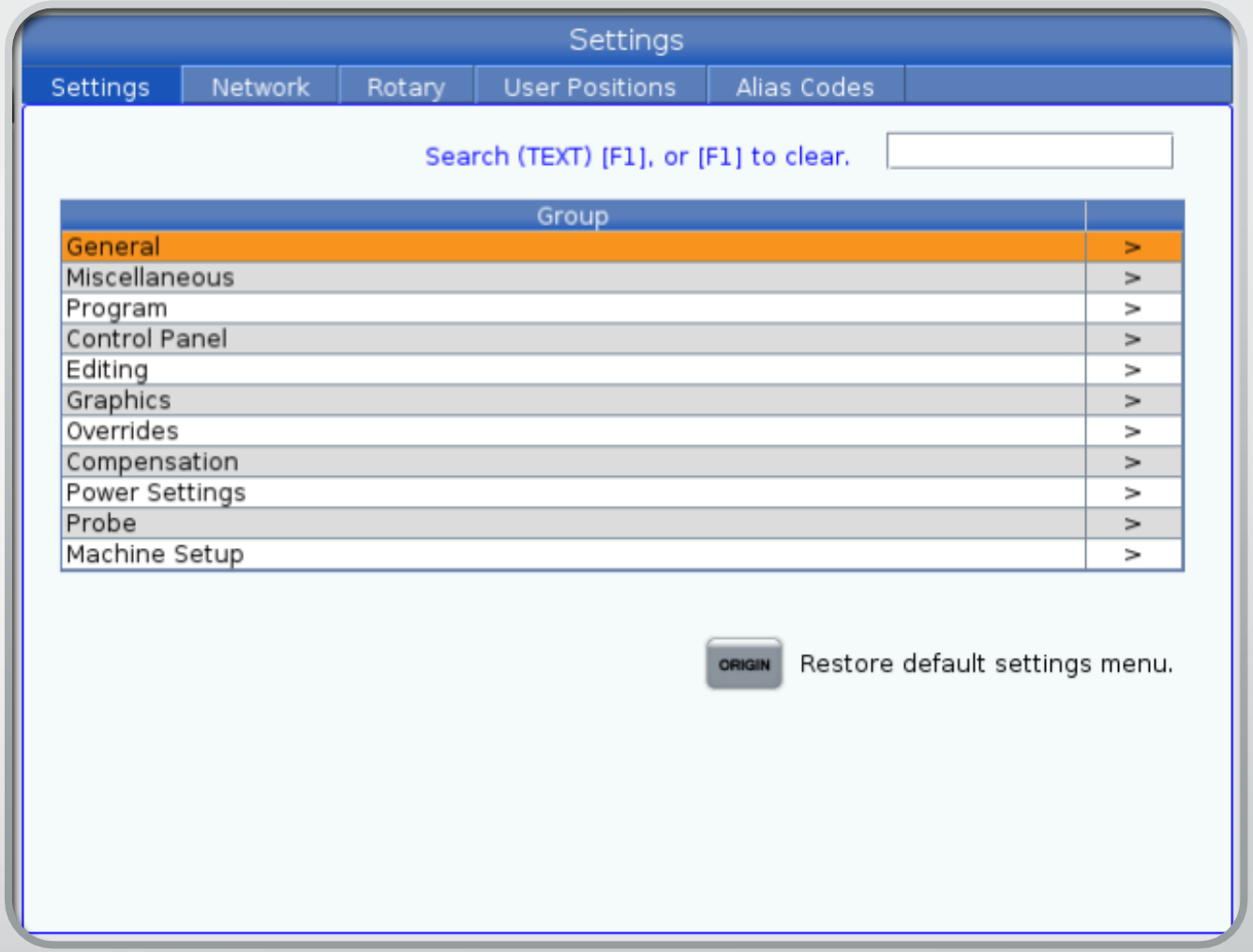
Machine Components

- Major components of a CNC milling machine
  - Control Pendant
    - The control pendant is the operator's interface with the machine

Photo curtesy of Haas Automations Inc.
http://www.haas.com.mx/pdf/haas_cmv2016_brochure.pdf

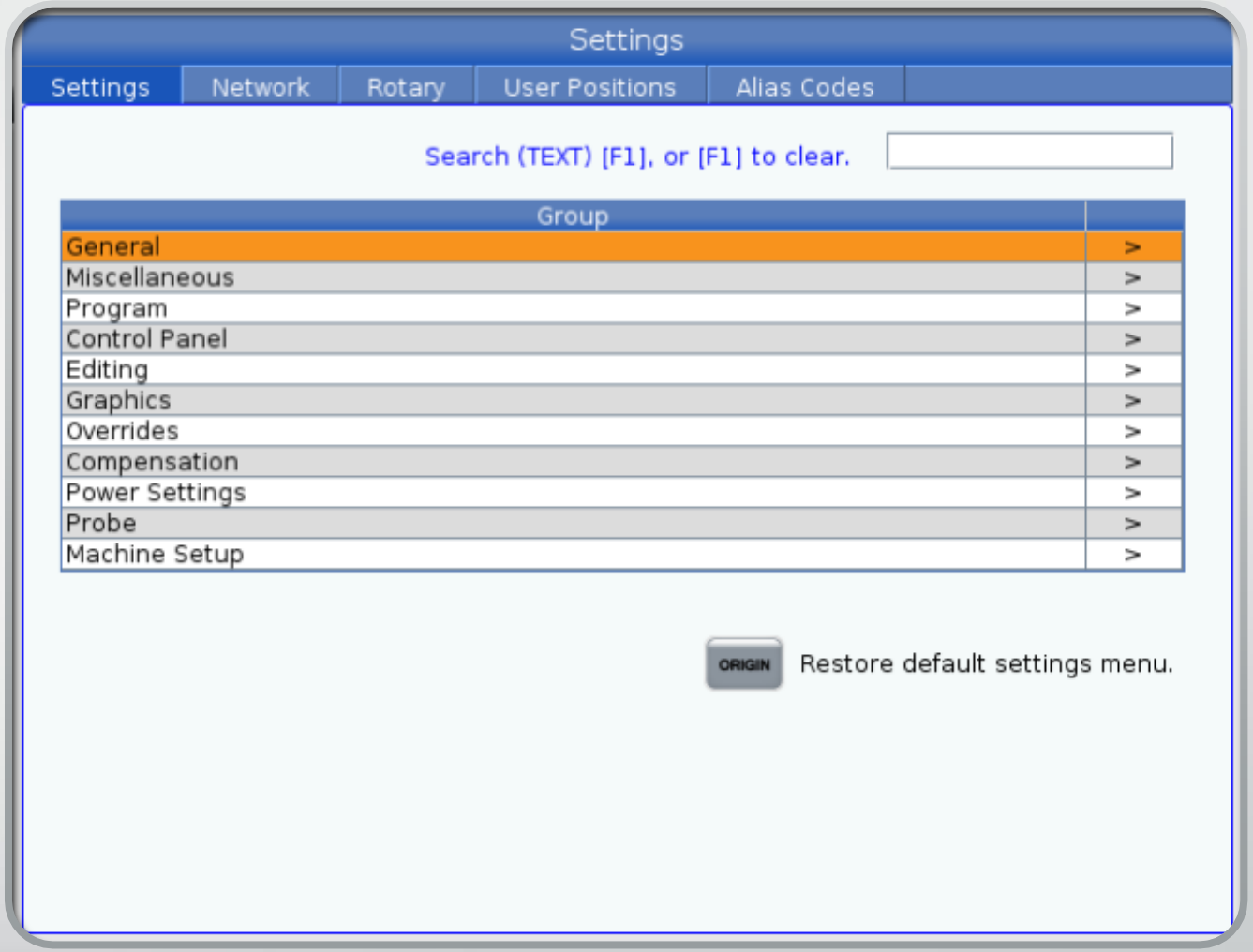
Settings on a Haas Control

- Settings are default values that can change the way the machine performs.
- Settings can be found in the display section of the control.
  - It is good practice and sometimes necessary to press the "Emergency Stop" button before changing settings.



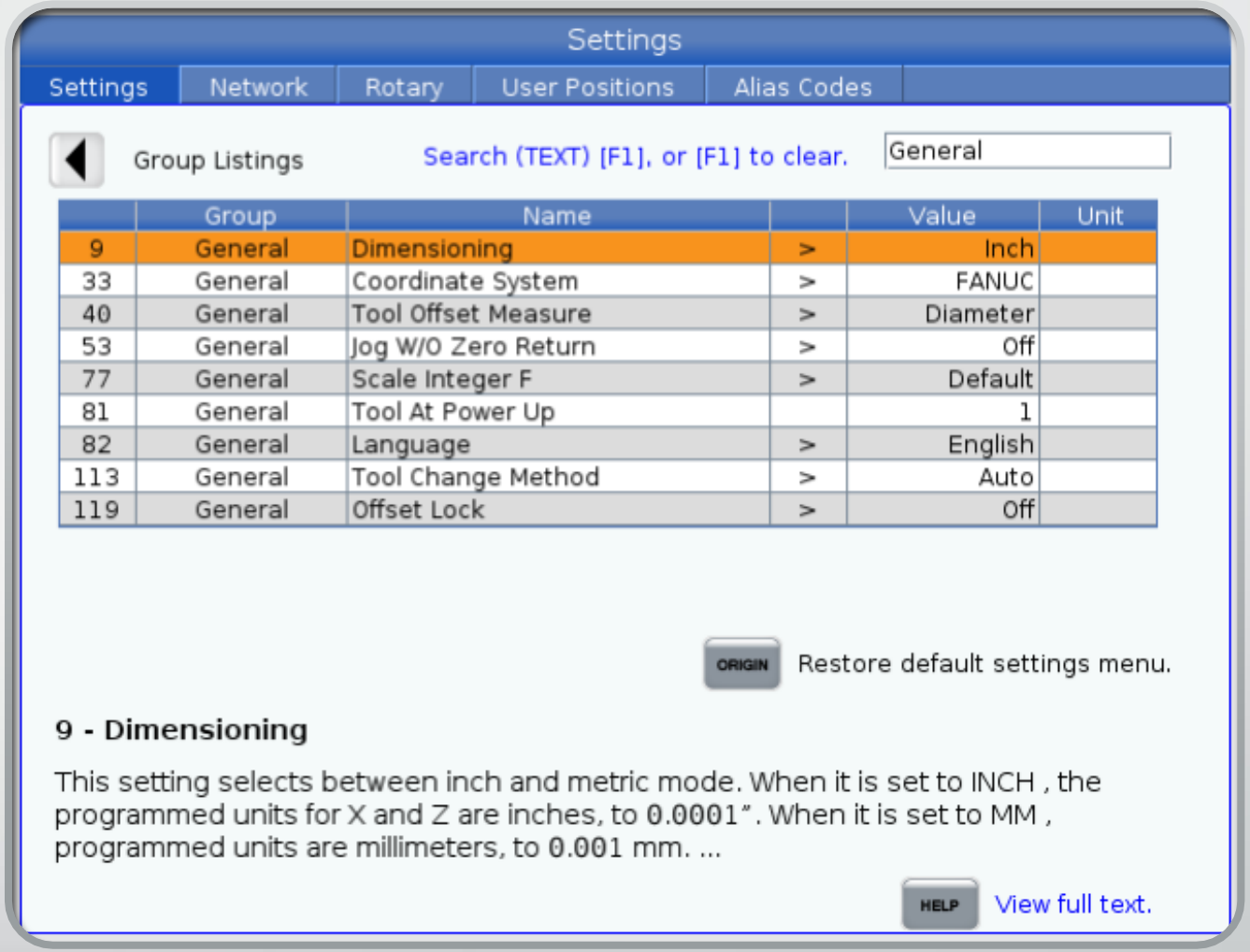
Settings

- Settings are broken into the following groups:
  - General
  - Miscellaneous
  - Program
  - Control Panel
  - Editing
  - Graphics
  - Overrides
  - Compensation
  - Power Settings
  - Probe
  - Machine Set-up



General Settings

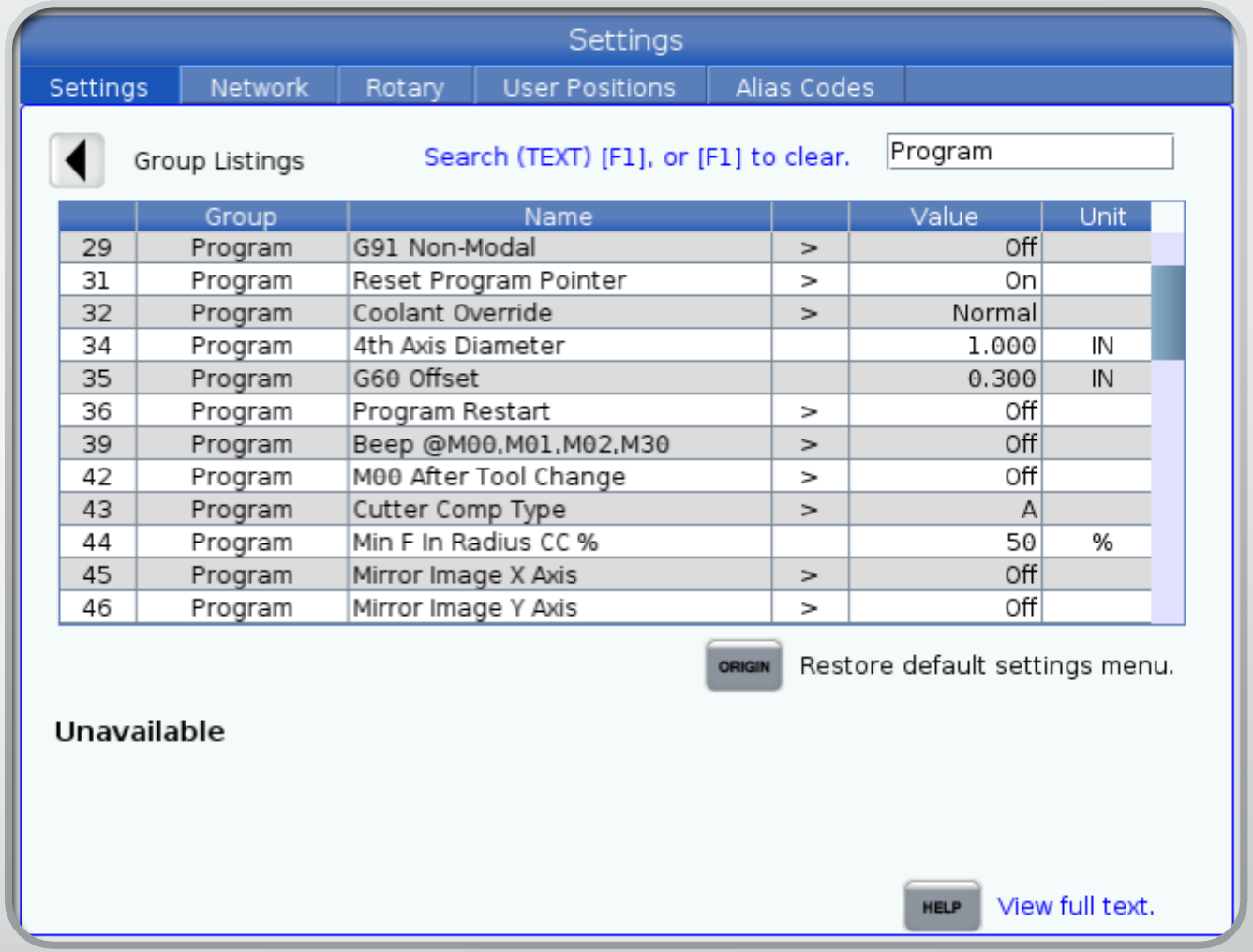
- Common general settings that are changed:
  - 9 – Dimensioning – This can change the display between inch and metric.
  - 53 – Jog W/O Zero Return – This can be activated so the machine can be moved before homing.
  - 119 – Offset lock – This can turn off the ability to make offset adjustments.



	Group	Name		Value	Unit
9	General	Dimensioning	>	Inch	
33	General	Coordinate System	>	FANUC	
40	General	Tool Offset Measure	>	Diameter	
53	General	Jog W/O Zero Return	>	Off	
77	General	Scale Integer F	>	Default	
81	General	Tool At Power Up		1	
82	General	Language	>	English	
113	General	Tool Change Method	>	Auto	
119	General	Offset Lock	>	Off	

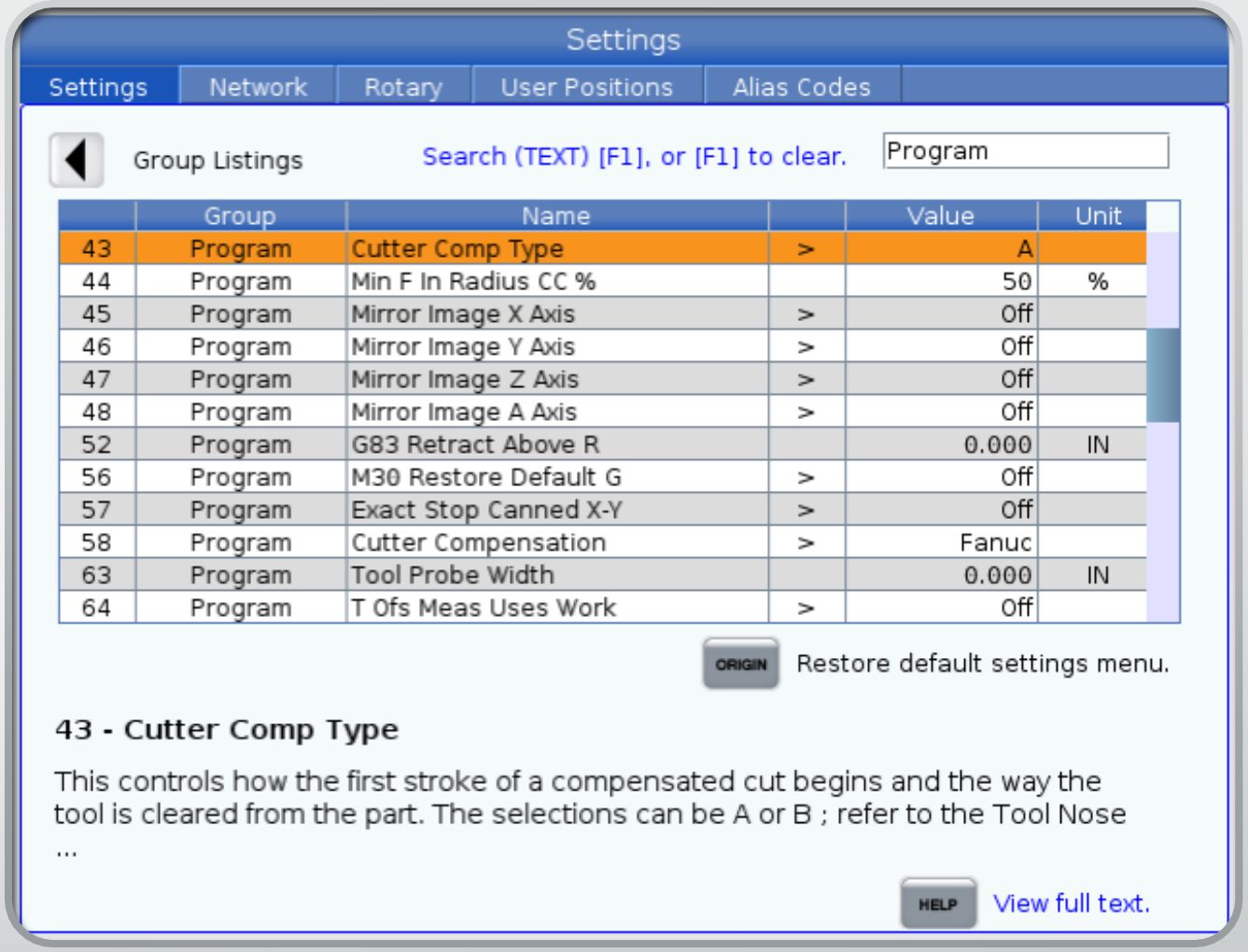
Program Settings

- Common Program settings that are changed:
  - 36 – Program restart – This is the ability to start a program at any point in the program.
  - 39 – Beep @M00, M01, M02, M30 – This controls the beeping sound at any of these codes.



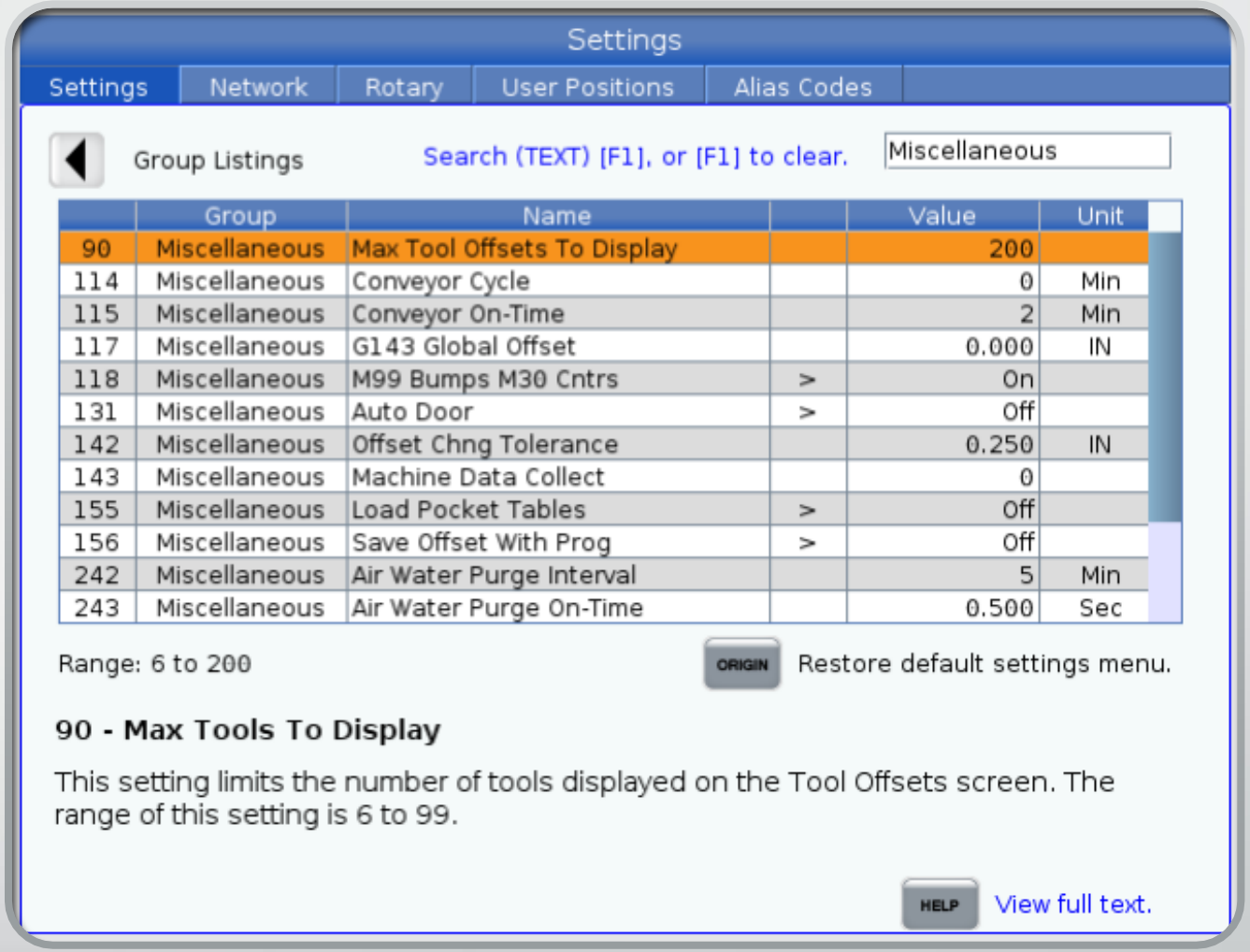
Program Settings

- Common Program settings that are changed:
  - 52 – G83 Retract above R – This value controls the retract setting in the G83 canned cycle.



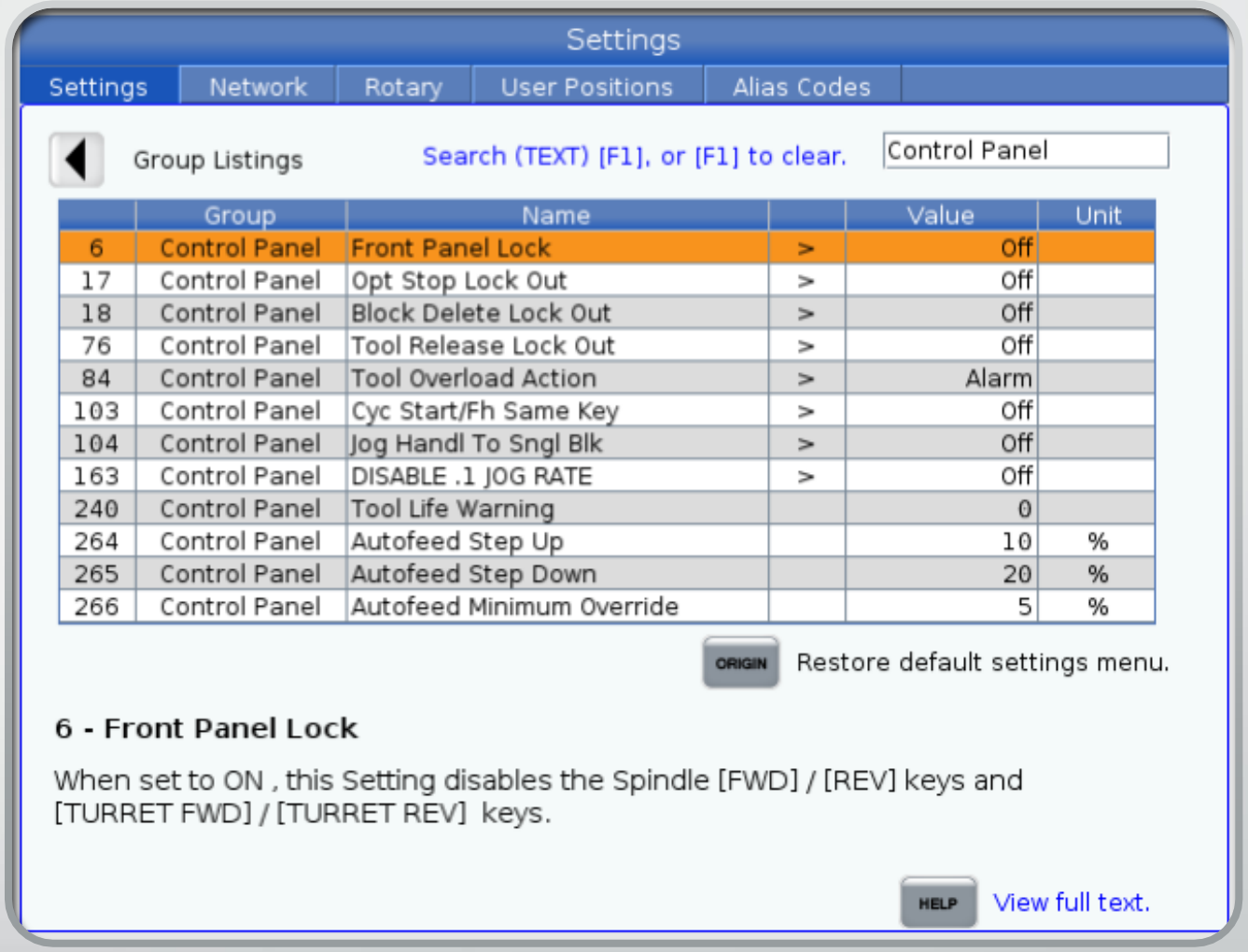
Control Settings

- 142 – Offset change tolerance – This prompts the user if an offset change is made greater then the 142 value.



Power Settings

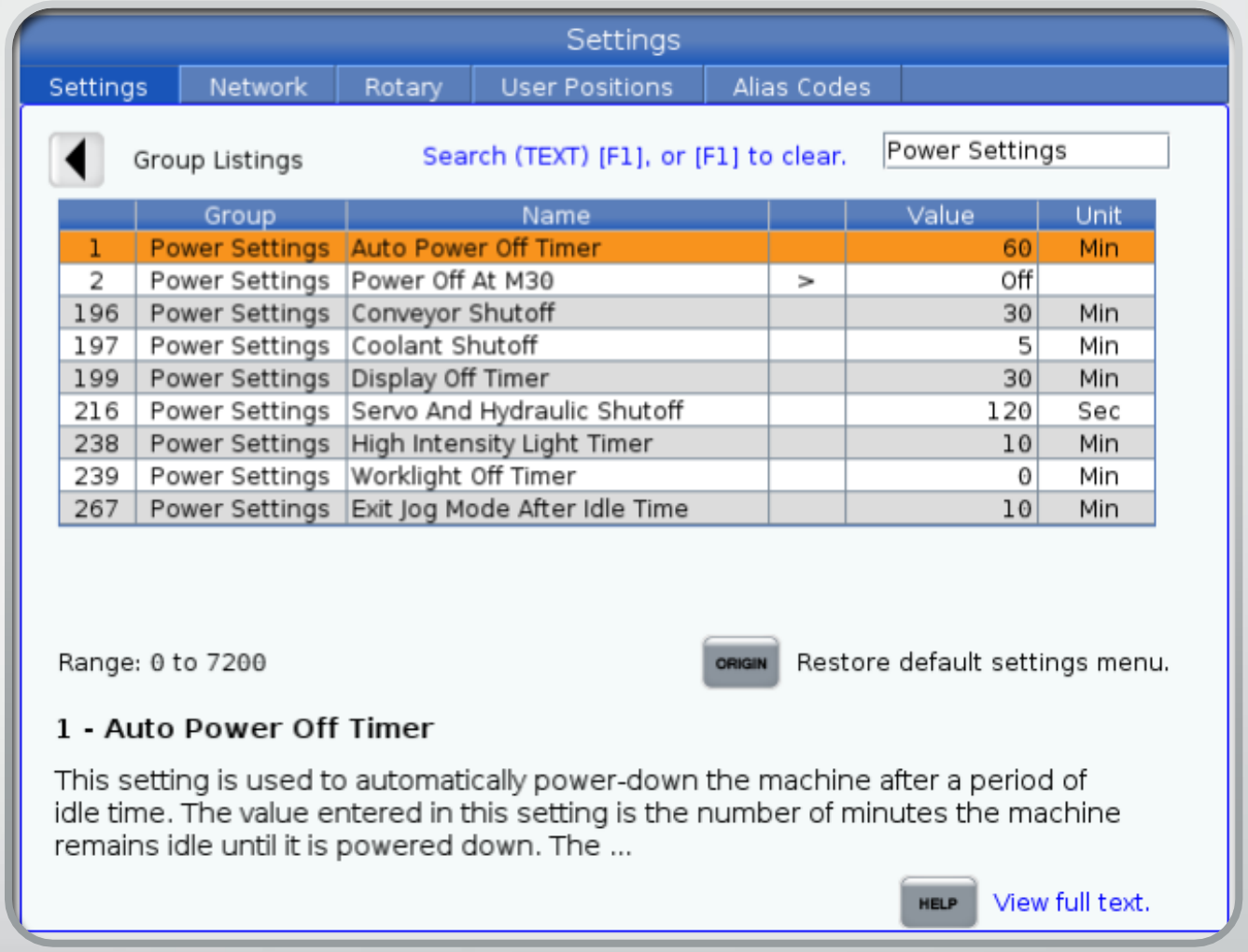
- 76 – Tool release lock out – this can lock out the tool release button on the control panel.
- 163 – Disable .1 jog rate – this can disable the .1 jog feed command.



	Group	Name		Value	Unit
6	Control Panel	Front Panel Lock	>	Off	
17	Control Panel	Opt Stop Lock Out	>	Off	
18	Control Panel	Block Delete Lock Out	>	Off	
76	Control Panel	Tool Release Lock Out	>	Off	
84	Control Panel	Tool Overload Action	>	Alarm	
103	Control Panel	Cyc Start/Fh Same Key	>	Off	
104	Control Panel	Jog Handl To Sngl Blk	>	Off	
163	Control Panel	DISABLE .1 JOG RATE	>	Off	
240	Control Panel	Tool Life Warning		0	
264	Control Panel	Autofeed Step Up		10	%
265	Control Panel	Autofeed Step Down		20	%
266	Control Panel	Autofeed Minimum Override		5	%

Power Settings

- Each of the power settings control when the feature shuts off. Each of these settings can be adjusted in "minutes" as a length of time.
- 1 – Auto Power Off Timer – This determines how long the machine can be inactive without turning off.



	Group	Name		Value	Unit
1	Power Settings	Auto Power Off Timer		60	Min
2	Power Settings	Power Off At M30	>	Off	
196	Power Settings	Conveyor Shutoff		30	Min
197	Power Settings	Coolant Shutoff		5	Min
199	Power Settings	Display Off Timer		30	Min
216	Power Settings	Servo And Hydraulic Shutoff		120	Sec
238	Power Settings	High Intensity Light Timer		10	Min
239	Power Settings	Worklight Off Timer		0	Min
267	Power Settings	Exit Jog Mode After Idle Time		10	Min

Types of Toolholders

- The most common types of tool holders are solid holder, collets, and Jacobs chucks.

Types of Toolholders

- Toolholders are used to hold the cutting tool in the spindle.
- Toolholders are removable, can be stored in a tool carousel, and changed on command in the program.

Parts of a Toolholder

The main parts of a toolholder are:

- Retention Knob
- Tool Taper Shank
- V-groove Slot Flange
- Key Slots
- Nose Projection
- Tool Engagement

Parts of a Toolholder - Retention knob

Retention Knob

- The retention knob, or pull stud, is used to hold the tool in the spindle.
- Each brand of CNC machine tool can use a specific retention knob. Refer to the machines manual for the correct retention knob.
- The retention knob should be properly torqued when installed in the toolholder.
  - Under-tightened retention knobs can loosen, while over-tightened retention knobs can stretch the end of the tool holder.

Parts of a Toolholder- Tool Taper

Tool Taper Shank

- The taper shank aligns the tool holder with the center of the spindle.
- Tapers are referred to in sizes.
  - Common tool taper sizes are 30, 40 and 50.
  - The larger the taper, the more rigidity the tool holder can have.
- Both the tool holder and spindle tapers should be cleaned often to minimize debris in the spindle.

Parts of a Toolholder - V Flange

V Groove Flange

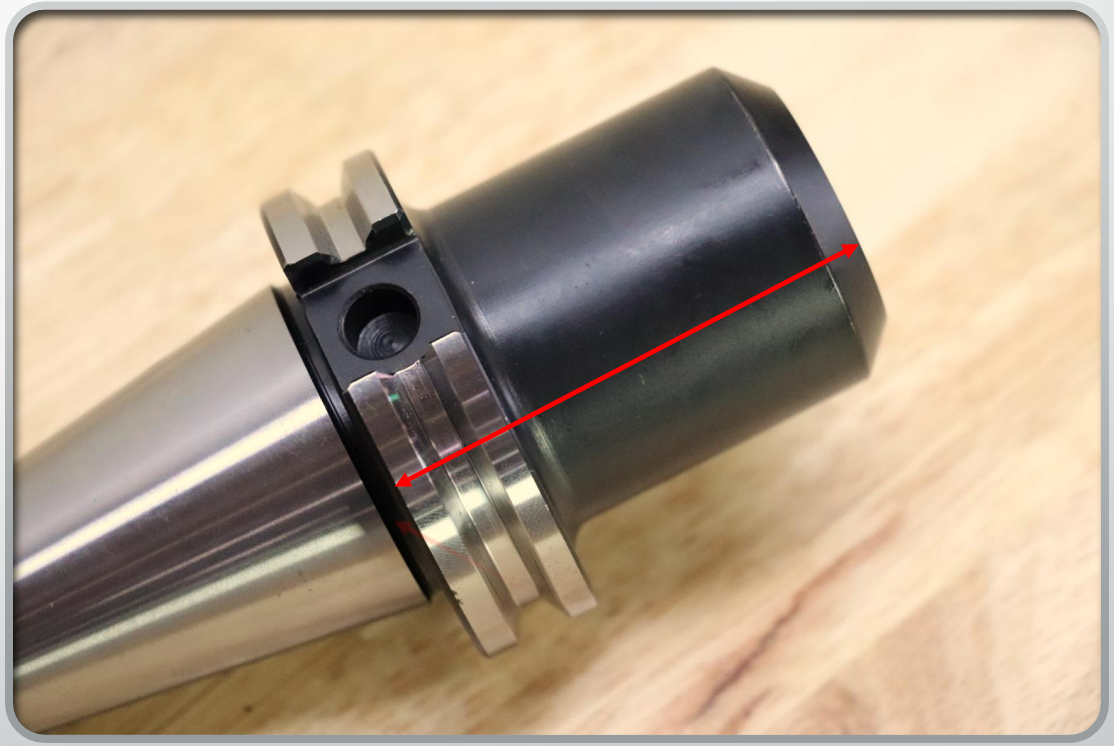
- The V-flange on the tool is used for tool change purposes.
- There are two types of flanges.
  - Cat – Uses a V flange centered in the flange of the tool.
  - BT – Uses a V flange that is offset to the bottom of the flange.
- Using the wrong flange in a machine with a tool changer can damage the tool change arm or carousel.

Parts of a Toolholder - Key Slots

- Key Slots
  - While the tool taper and retention knob hold the tool in the spindle and locate, they do not provide enough holding force to prevent the holder from spinning during tool cutting.
  - The two key slots in the V-flange locate with the bosses in the spindle and provide a physical stop to keep the toolholder from rotating during cutting.
  - The key slots also provide rotational orientation for the tool.

Parts of a Toolholder - Nose Projection

- Nose Projection
  - This references the distance from the back of the V flange to the end of the holder.
  - This distance should always be kept as short as possible for rigidity.
  - Some operations require an increased nose projection due to the work environment.



Tool Engagement - Tool Holder Styles

- Tool engagement refers to how the holder holds the tool/ type of tool during the cut.
- There are numerous specialty tool holders. This unit will focus on the following basic tool holders:
  - Shell Mill holder
  - End Mill holder
  - Collet Holder
  - Drill Chuck
  - Tap Holder

Tool Holder Styles - Shell Mill

- Shell mill holders have a boss and drive keys to mate to the shell mill.
- Drive keys width and boss diameters vary and must be purchased to fit a specific shell mill.
- Once the shell mill is installed on the holder, it is retained with a cap head screw.

Tool Holder Styles - End Mill

- Endmill holders use a precision bore and a set screw to hold an endmill in place.
- Endmills should have a flat on the shank to seat against the set screw.
- Tightening the set screw on the round diameter of the endmill will not create sufficient holding force.
- Because the endmill holder has a slip fit for the endmill, there is an inherent amount of runout when an endmill holder is used.
  - This causes the high side of the endmill to cut more then the low side and will cause uneven tool wear.

Tool Holder Styles - Collet

- Collet holders come is many styles:
  - ER, DA, TG are a few common collet styles
  - Collets also come in different size ranges:
    - For example: ER collets can be purchased in series 16, 20, 25, 30 and 40.

Tool Holder Styles - Collet

- Most collet styles are considered spring collets. This means the collets can expand/contract beyond their nominal size.
- Collets come in 1/32" increments and the closest size possible should be used.

Tool Holder Styles - Collet

- Collets need to be installed into the retaining nut and then screwed onto the collet holder.
- Once the assembly is complete, the tool can be installed and the retaining nut can be tightened.
- Collets provide minimal runout but also have minimal holding power and should not be used in applications with high cutting pressure.

Tool Holder Styles - Drill Chuck

- Drill chucks are primarily used to hold center drills and twist drills.
  - Drill chucks are useful during set-ups as they can hold the end of a test indicator.
- The chuck itself is often held onto a holder via a friction fit on a taper.
- Once a chuck is seated on the holder taper, a special wedge is needed to remove the chuck

Tool Holder Styles - Tap Holder

- Tap holders come in a variety of styles.
- The common theme with tap holders is the positive drive. A square drive at the end of the collet helps the tool from spinning during the thread cutting process.

Tool Holder Styles - Tap Holder

- A simple form of tap holder is an ER collet with the square drive in the end of the collet.
- This is a simple and effective way to hold smaller taps but the ER collet can still spin in the retaining nut when used with larger taps.

Tool Holder Styles - Tap Holder

- Tap holders similar to those pictured to the right provide maximum holding power.
- This holder slides into an endmill holder, but these can be purchased as one integrated unit.

Height Offsets

What are height offsets?

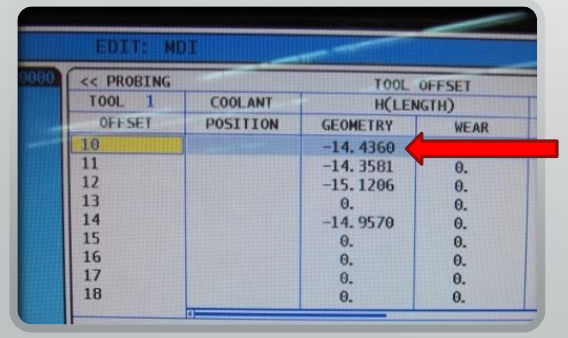
- Height Offsets are the measured distance from the tip of the tool to the Z PRZ of the work piece when the machine is at its Z home position

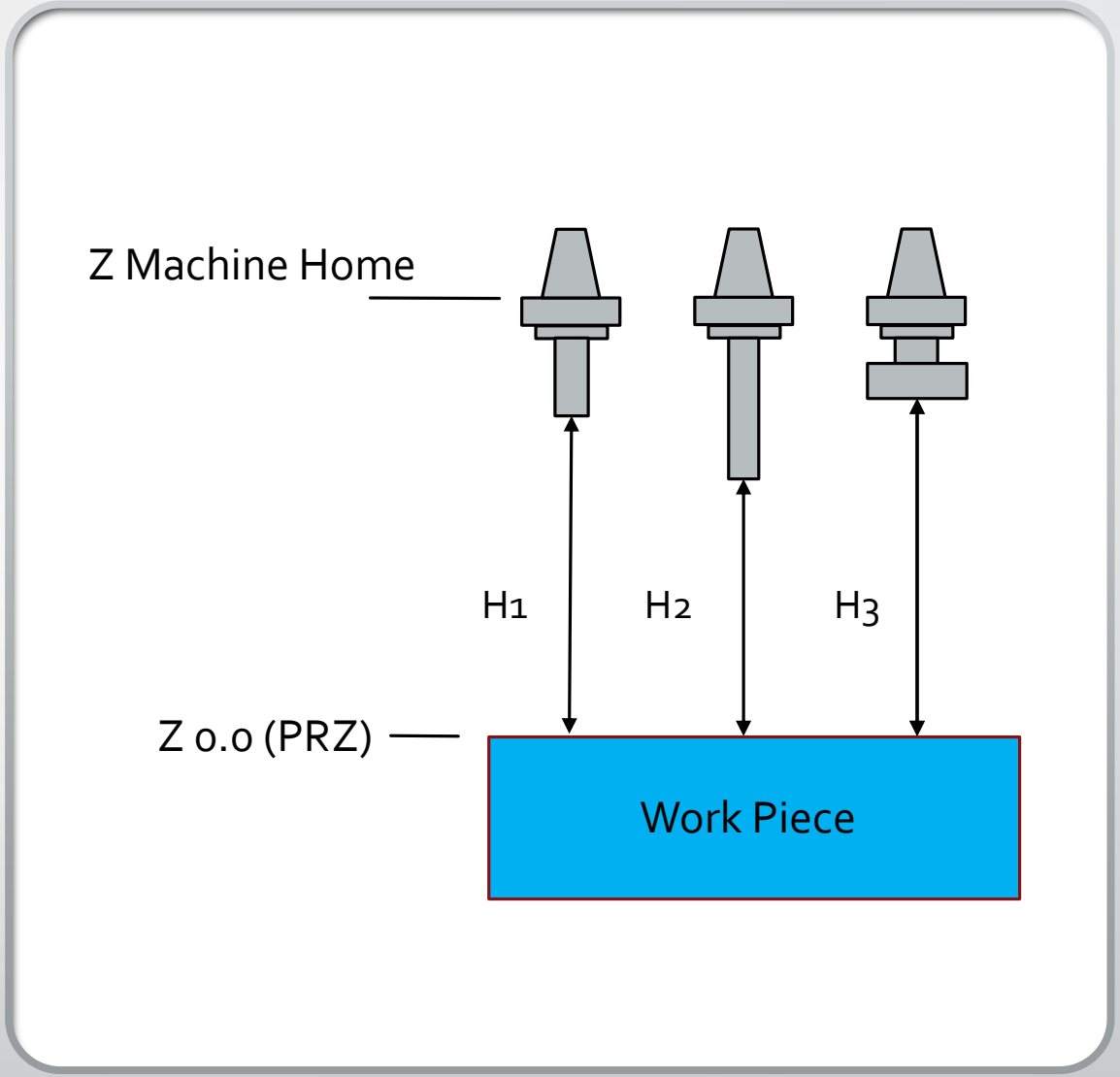
Two methods for calculating height offsets

- Air Gap Method:
  - Simple, easy to use
  - Slower when there are multiple tools
  - This is were we will start
- Gage Line Method
  - More math involved
  - A height gage or presetter is needed
  - Faster when there is a large tool list

Air Gap Height Offsets

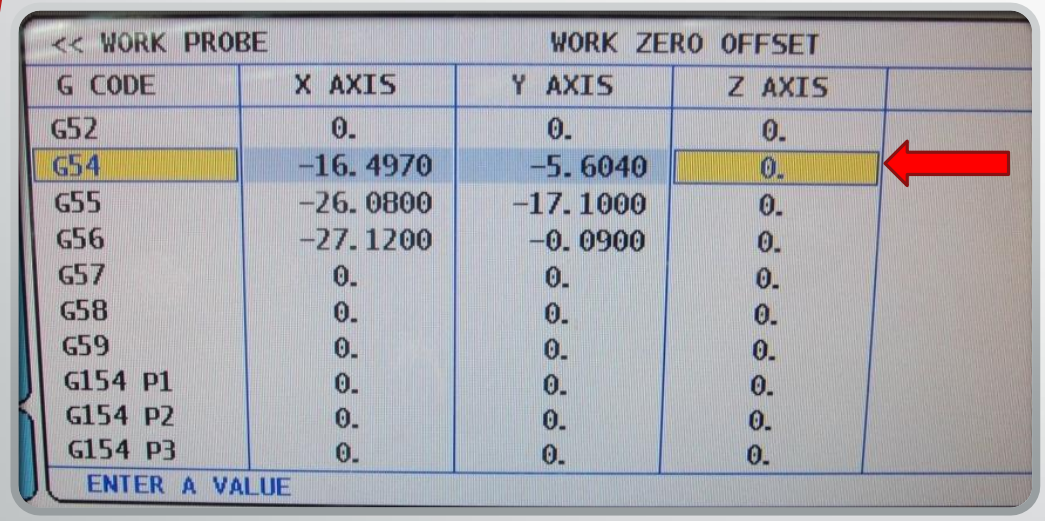
- Air Gap Method
  - All tools are measured in the machine by touching off the top of the part
  - Does require the operator to manually touch every tool off in the machine





Tool heights are stored in the tool offsets as a large negative value

Air Gap Method



- Leaves the Z Work Offset set as a 0.0 in the registry

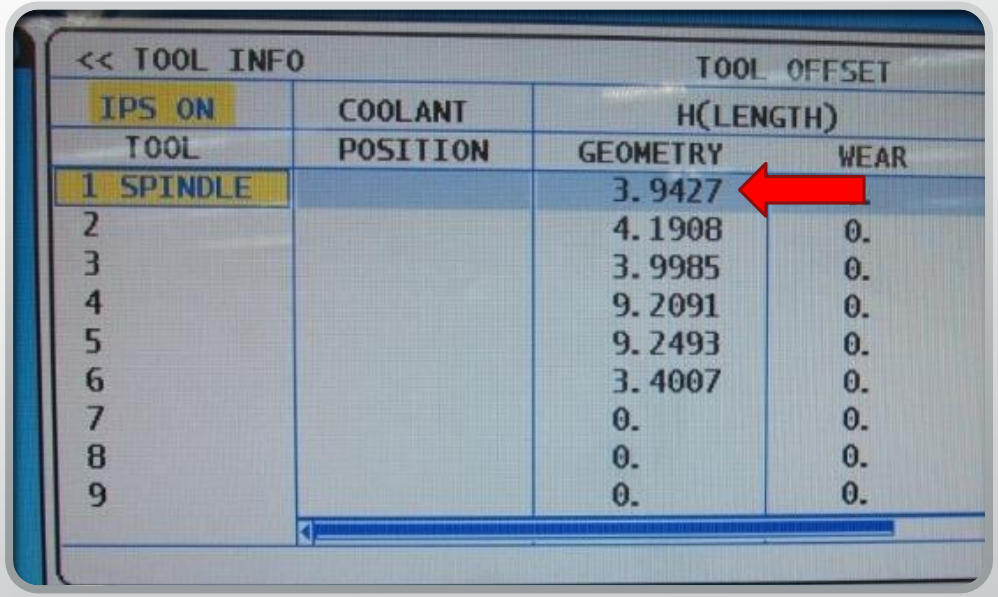
- **Pros:**
  - No pre-setter needed
  - Fast for parts with few tools
- **Cons:**
  - Each tool must be touched off in the machine
    - Cumbersome when parts have lots of tools
  - When a tool is dull and the operator changes it, it will need to be touched off again

Gage Line Method (How Tool Setters Work)

- **A multi step method:**
- **First**: find the distance from the gage line to the tip of the tool
  - The **gage line** is the theoretical line on the tool taper where both the tool and spindle meet - a constant for all tools

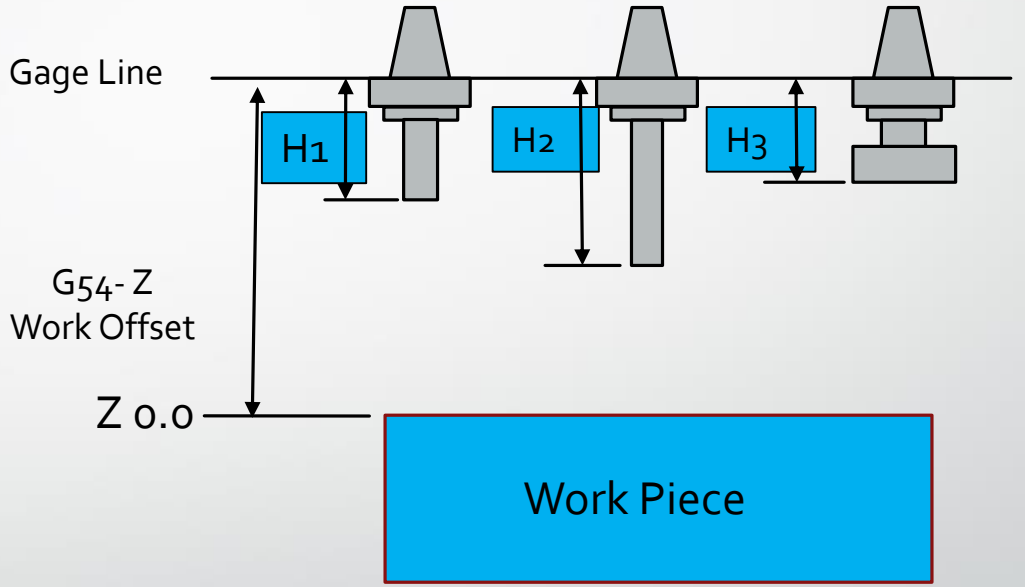
Gage Line Method

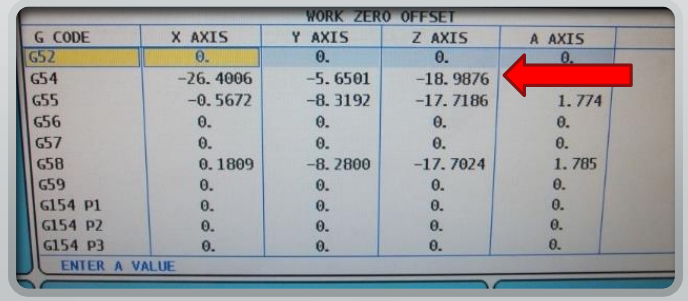
- **Second:** enter tool heights into the machine
- The offsets are entered into the Tool Offset page as a positive value



Gage Line Method

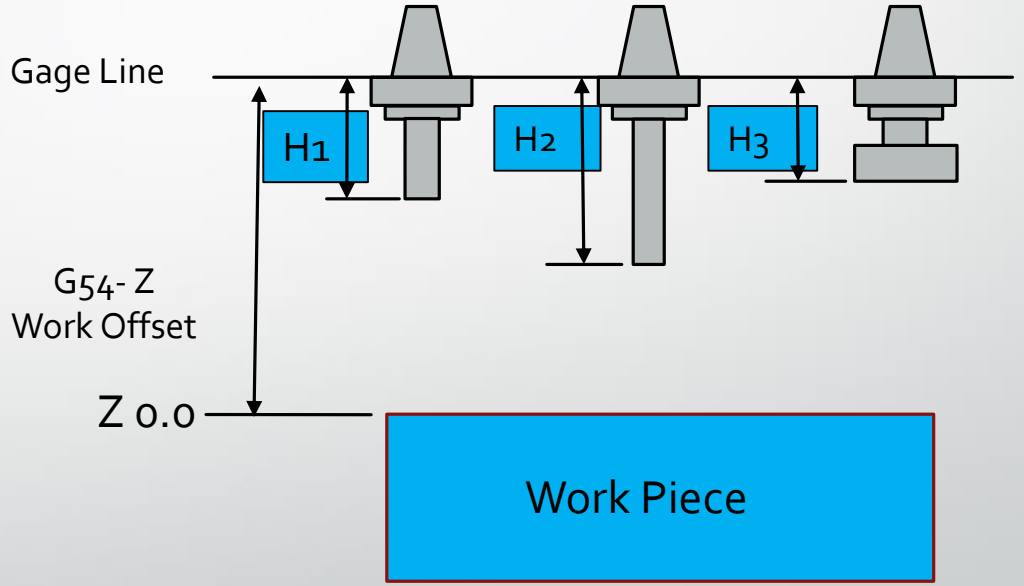
- **Third:** find the distance from the gage line of the spindle to the top of the part
- This measurement is the distance from the gage line of the spindle to the top of the part
  - Distance must be entered in the work offset for the Z axis as a negative value





How does this all work?

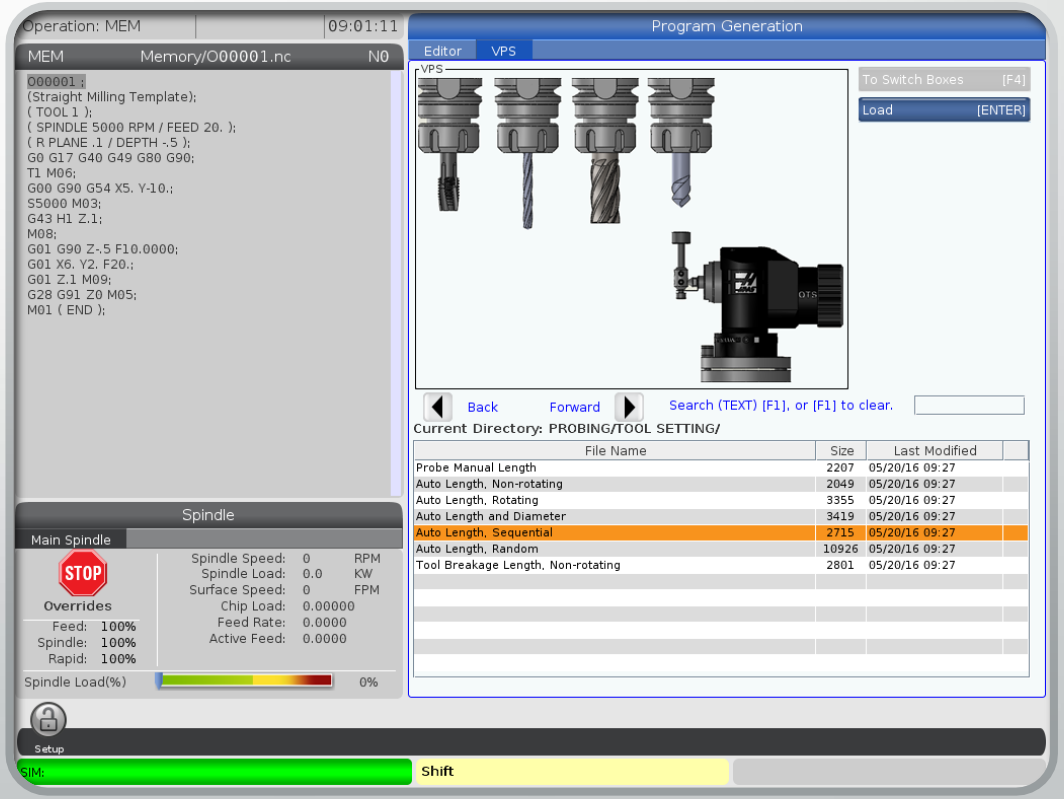
- The Z value in the G54 work offset is a negative value from the gage line to the top of the part
- When the height offset is called up in the program, it lifts the spindle up and puts the tip of the tool on the top of the part



Using the tool setter on a Haas CNC mill

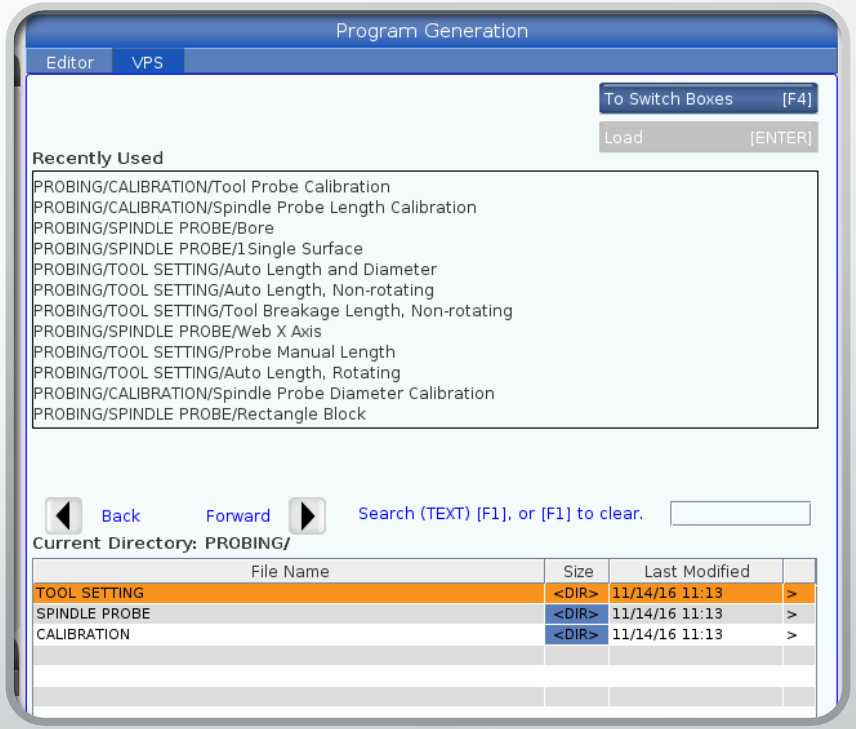
How is a Tool Setter Programmed

- To efficiently program tool setter cycles, Haas has incorporated tool setting cycle interfaces on the CNC control.
- This saves the programmer having to program the routines by hand.



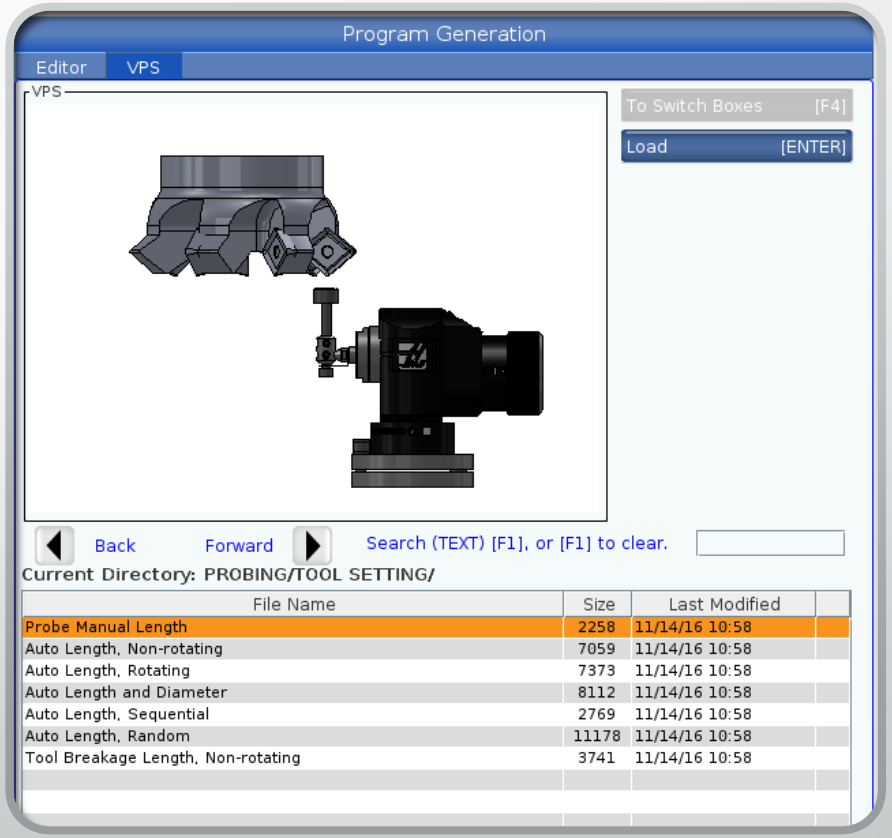
How is a Tool Setter Programmed Cont.

- The tool setting cycles can be found by entering the edit mode on the machine, then cursor over to the visual programming system (VPS) tab.
- Once in the VPS tab, tool setting can be highlighted and the right cursor key can be used to enter the tool setting directory.



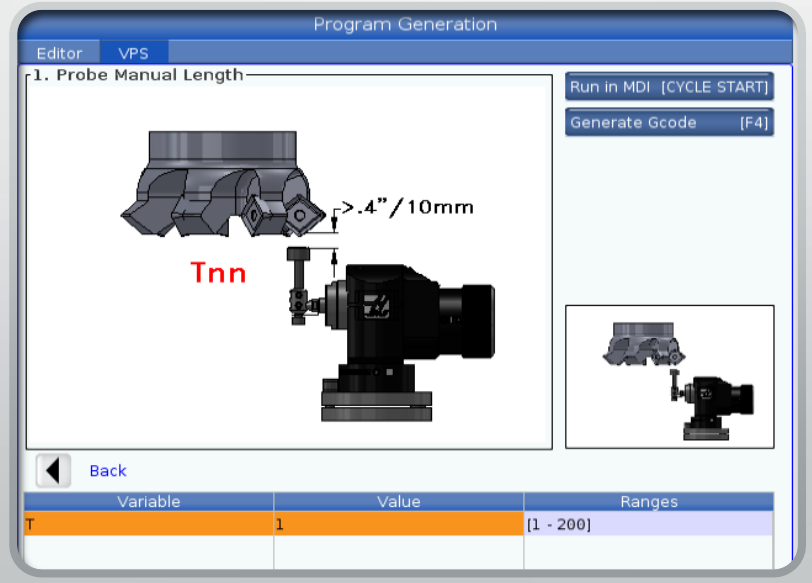
How is a Tool Setter Programmed Cont.

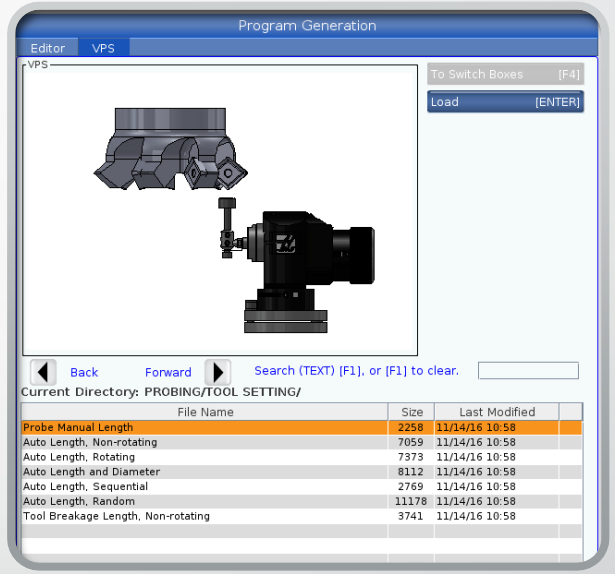
- Once in the tool setting directory there are seven tool setting cycles to choose from:
  - Probe Manual length
  - Auto Length, Non-rotating
  - Auto Length, Rotating
  - Auto Length and Diameter
  - Auto Length, Sequential
  - Auto Length, Random
  - Too Breakage Length, Non-rotating
- Each cycle has a different purpose and will be reviewed in detail



Probe Manual Length

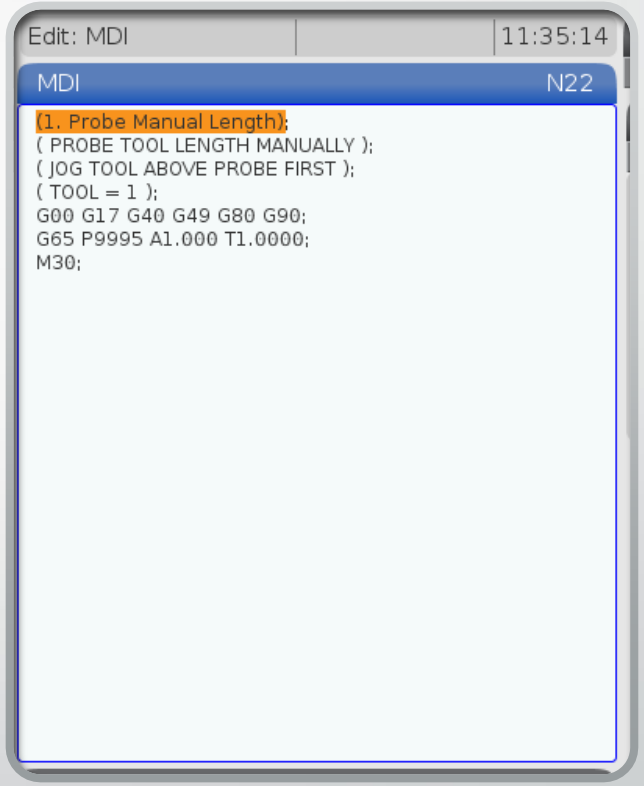
- Probe manual length is a basic tool setting cycle that requires the user to manually position the tool over the tool setter.
- The tool should be within .400/10mm of the stylus.
- The only variable needed to create the code is the tool number.

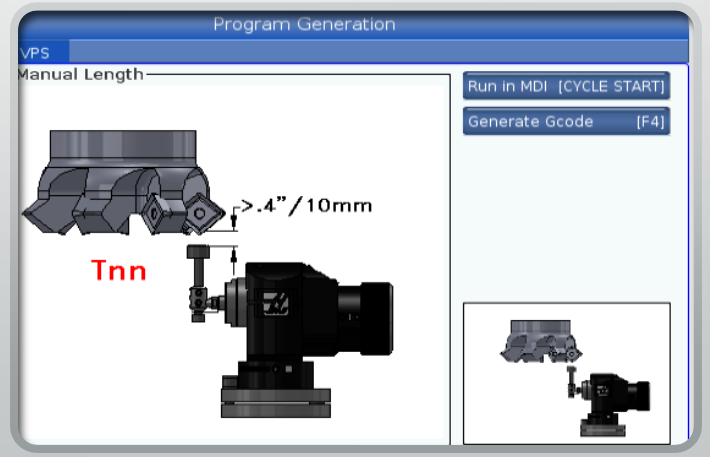




Probe Manual Length

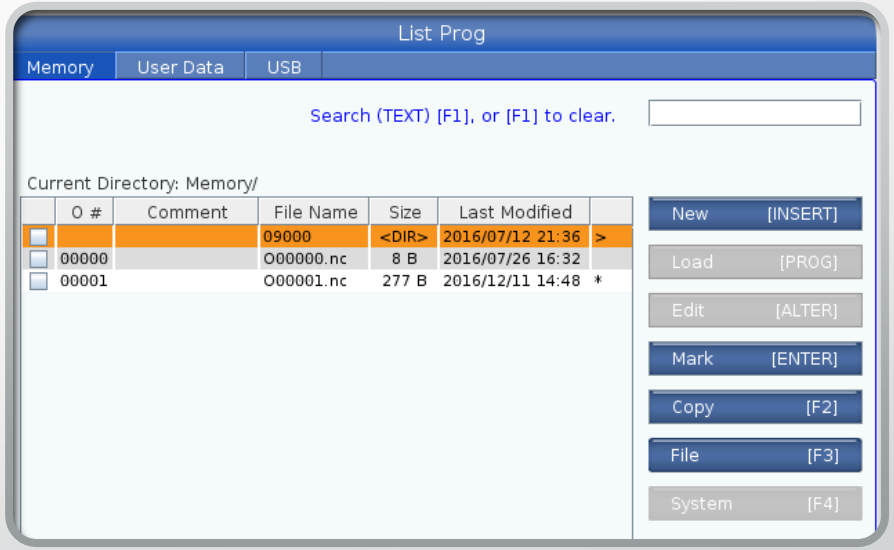
- After the tool number is entered, the code can be created by pressing cycle start. This will output the code to the MDI page.
- If F4 is used, the code can be inserted into a program or generated on the clipboard.
- Once the code is created and the tool is in position, the cycle can be executed.





What is Really Being Created?

- The tool setting interfaces in the VPS directory are simply macro variable inputs.
- This means that a pre-written set of programs is stored in the machines control, in this case they are O9000 programs.
- The user enters the inputs needed for their desired tool setting cycle. When the inputs are entered and the program is generated, the control uses the inputs to fill in blanks in the stored macro program.

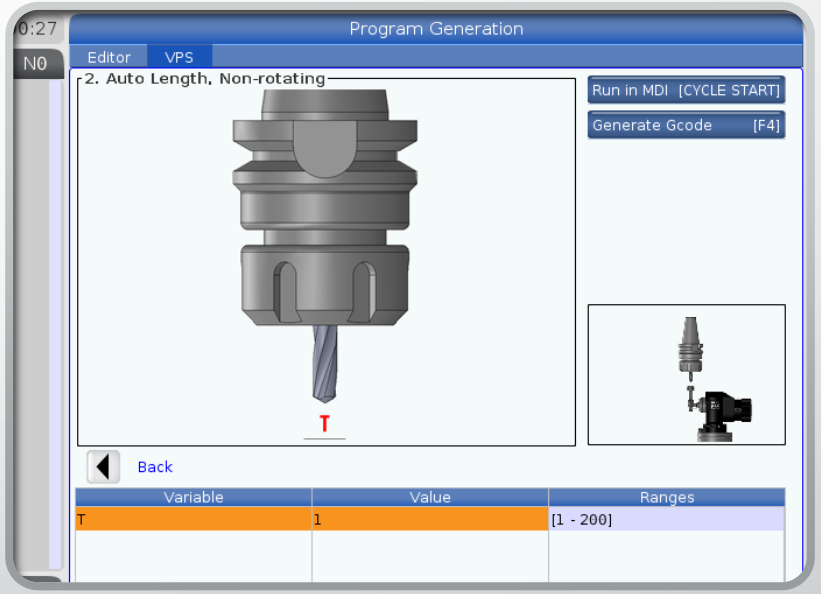


What is Really Being Created?

- The cycle that is output has the necessary variables to satisfy the stored macro routine.
- When the program is executed, it calls up the stored O9000 program to run.
- The O9000 program uses the variables from the main program to finish the tool setting cycle and store the data in the offset location.

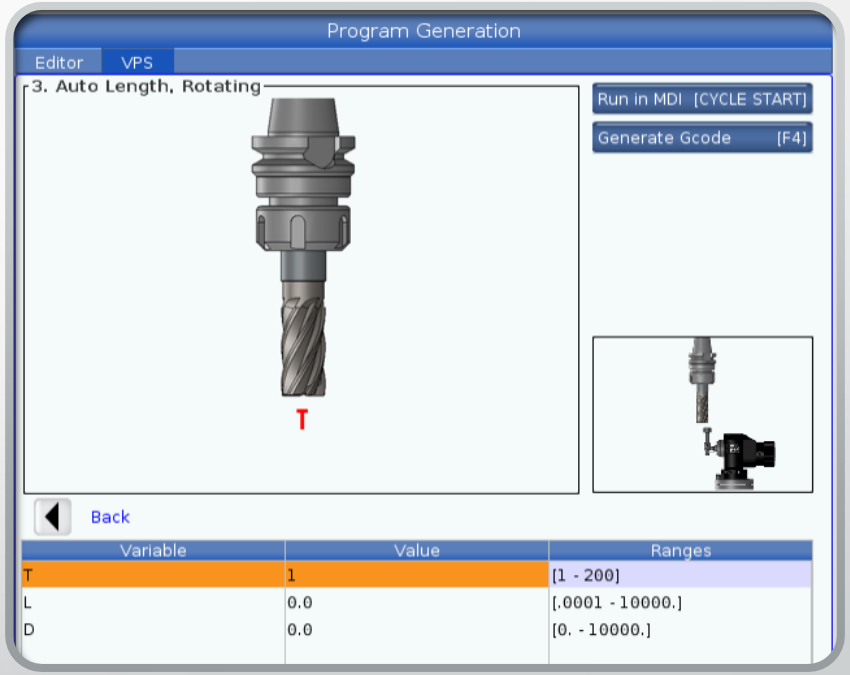
Auto length, Non-rotating

- Auto length, non-rotating is a tool setting function that will probe the tool automatically. No tool placement is needed from the user.
- The only variable needed is the tool number.
- When the cycle is executed, the tool will be changed to the corresponding number. The tool will position over the stylus and the tool will move down in Z until the it meets the stylus.



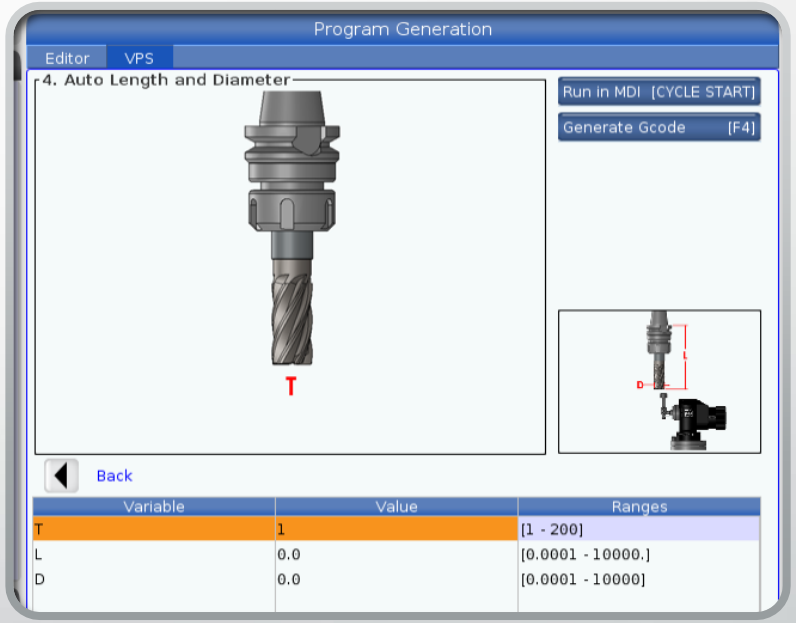
Auto length, Rotating

- Auto length, Rotating is a tool setting function that will probe the tool automatically. No tool placement is needed from the user.
- The following variables are needed to create the cycle:
  - Tool number
  - Approximate length
  - Tool diameter
- By rotating the tool, the offset is set to the true cutting height. This is effective when using multi flute tools that may have flute height variation.
  - An example would be a shell mill.
- Even though the diameter is not being checked, it is needed so the machine can offset the tool. This makes sure the cutting tip will contact the stylus surface.



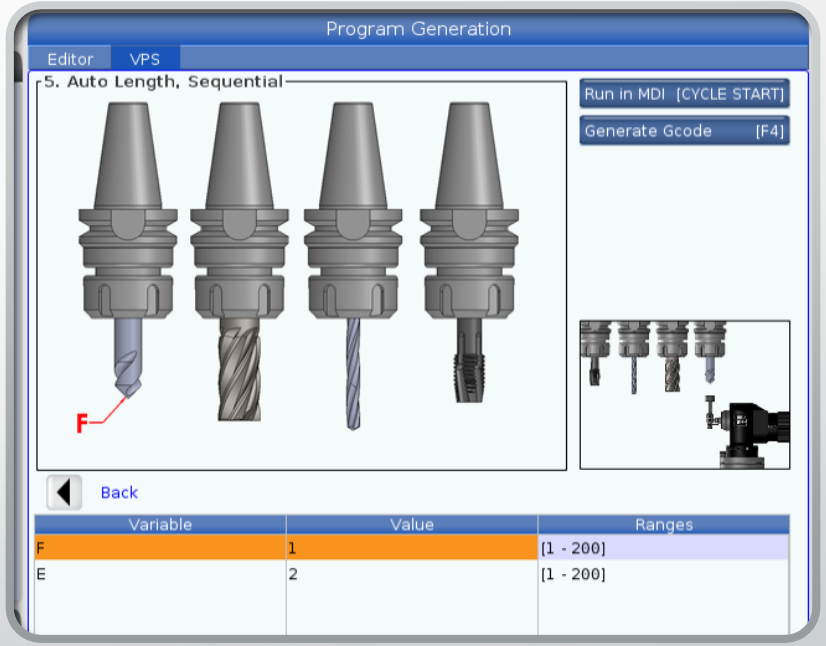
Auto length and diameter

- Auto length and diameter is a tool setting function that will probe the tool length and diameter automatically. No tool placement is needed from the user.
- The following variables are needed to create the cycle:
  - Tool number
  - Approximate length
  - Tool diameter
- By rotating the tool, the offset is set to the true cutting height. This is effective when using multi flute tools that may have flute height variation.
  - An example would be a shell mill.
- After the length is measured, the tool will set over and move down. The tool will then touch the side of the stylus to measure the actual cutting diameter.
  - The tool rotates counterclockwise while touching off. This prevents the tool from cutting into the stylus during the touch off.



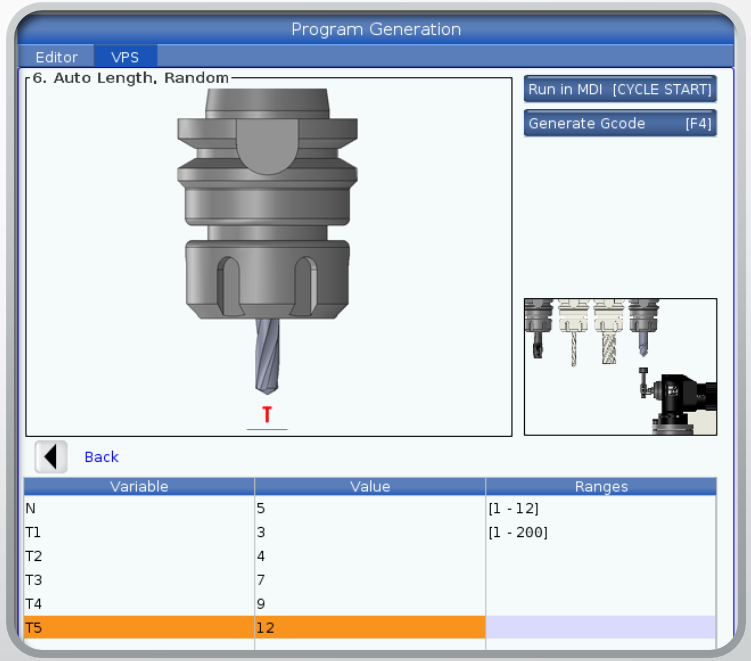
Auto length, sequential

- Auto length, sequential is a tool setting function that allows the user to set more than one tool automatically.
  - This cycle only works for tools that are sequentially numbered, such as tools 2 though 6.
- The following variables are needed to create the cycle:
  - The first tool number to probe
  - The last tool number in the sequence to probe
- The tools will be called up automatically and probed. When the probing is done, the tool will retract and the next tool will go through the probing cycle. This will repeat until the last tool sequence is done.



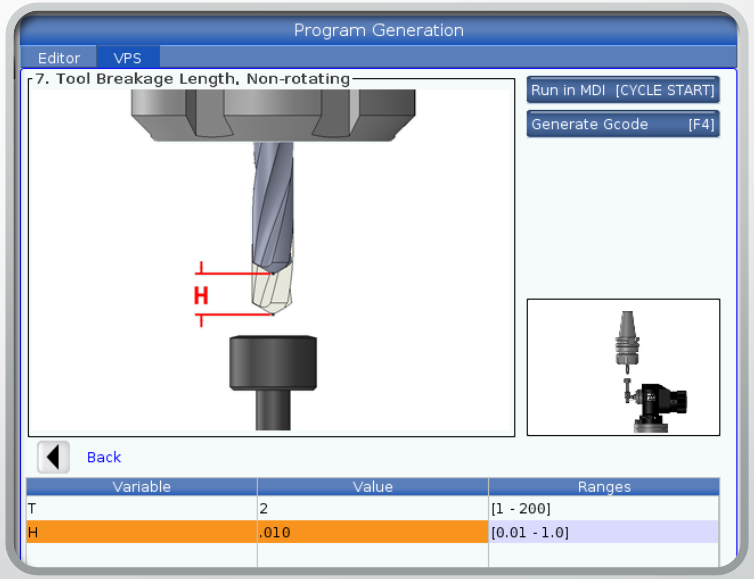
Auto length, random

- Auto length, random is a tool setting function that allows the user to set more than one tool automatically.
  - This cycle allows the used to automatically probe tools that are not sequentially numbered
- The following variables are needed to create the cycle:
  - The number of tools to be probed
  - Each tool number that needs to be probes
- The tools will be called up automatically and probed. When the probing is done the tool will retract and the next tool will go through the probing cycle. This will repeat until the last tool sequence is done.



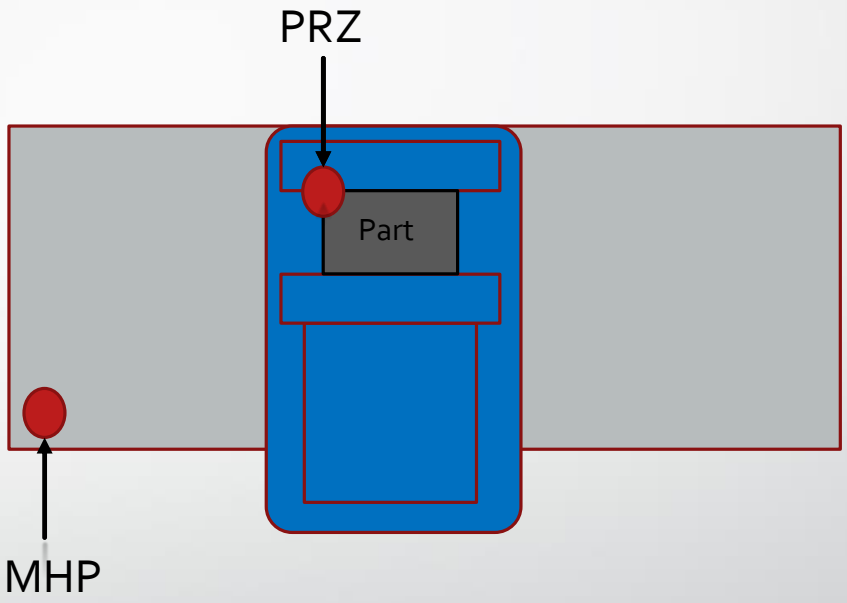
Tool breakage length, Non-rotating

- Tool breakage length, Non-rotating allows for the user to probe an existing tool to see if the height has varied from the original probed value.
- The following variables are needed to create the cycle:
  - The tool number to probe
  - The maximum height deviation allowed
- When the cycle is run, if the height is within tolerance, no error will be given. If the height is out of specified tolerance, an alarm will come up on the control.



Work Offsets

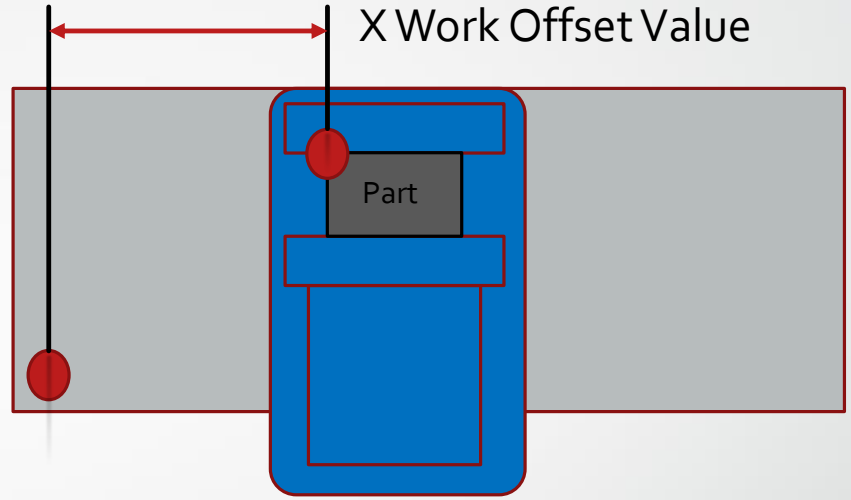
- Work Offsets are the distance from machine home to the XY PRZ of the part
- These offsets are stored in the work offset registry page of the machine
- Work Offsets allow us the flexibility to put a part anywhere on the machine table and record its position

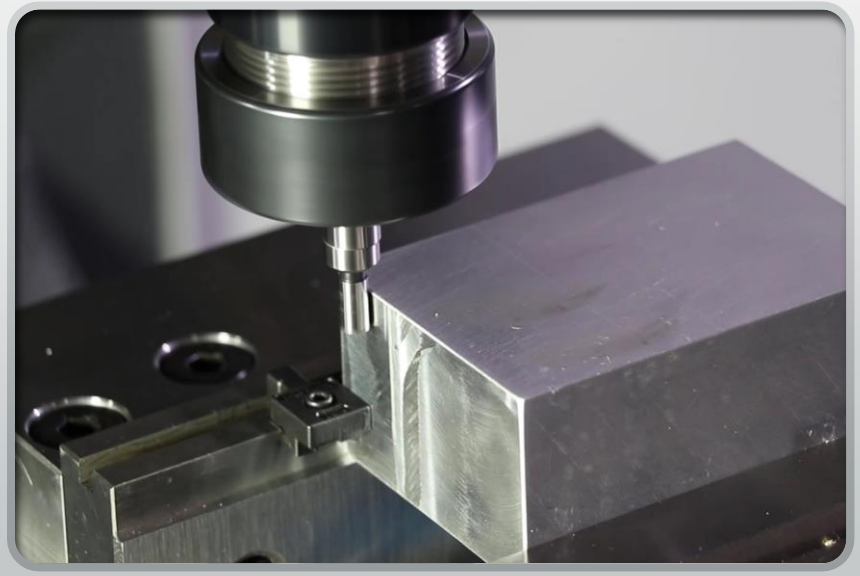


MHP - Machine Home Position

Work Offsets

- Any edge finder can be used to find the corner of the piece in both the X and Y location for the PRZ of the part

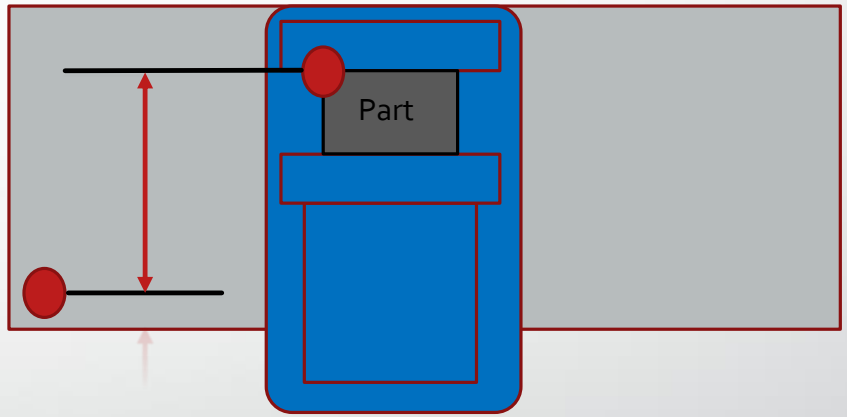




Work Offsets

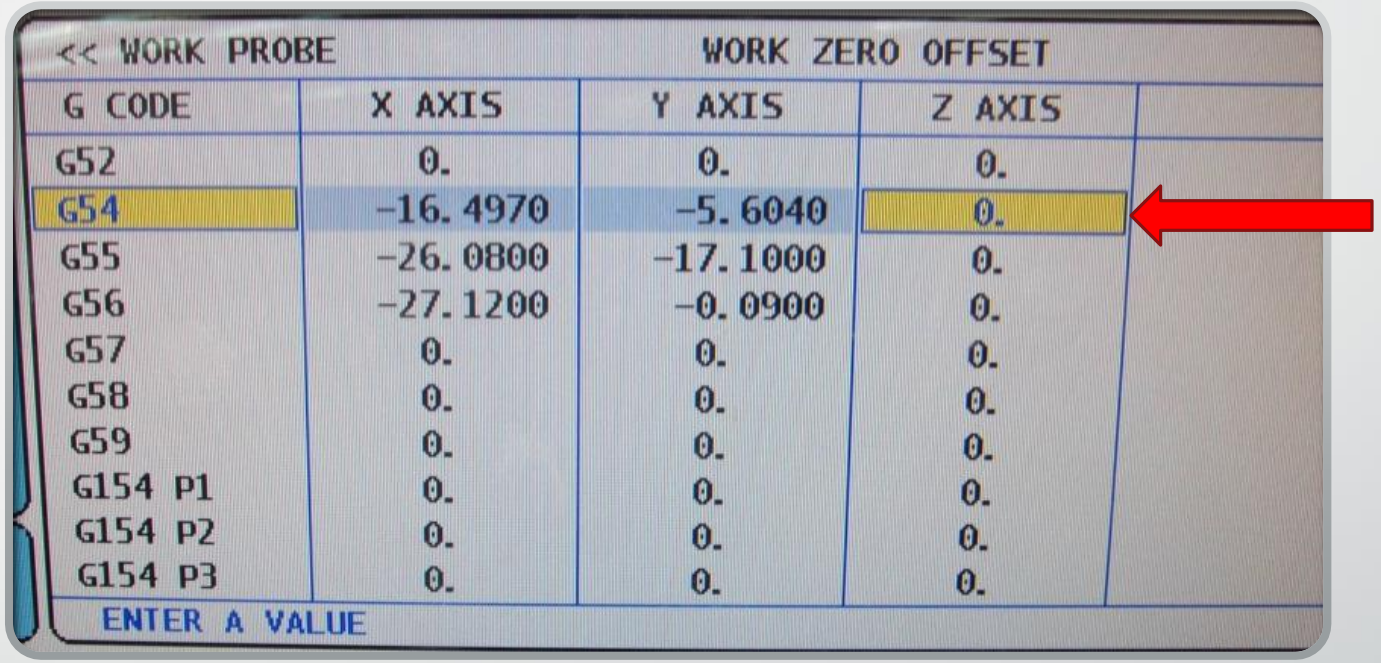
- Any edge finder can be used to find the corner of the piece in both the X and Y location for the PRZ of the part

Y Work Offset Value



Work Offsets

- Work Offset values are then stored in the work offset page
- G54 thru G59 are the standard work offset numbers that can be used
- Haas machines have 99 additional work offsets, making the total number of work offsets 105.
- The picture to the right shows how this can be done using G154 P1



What are Probes?

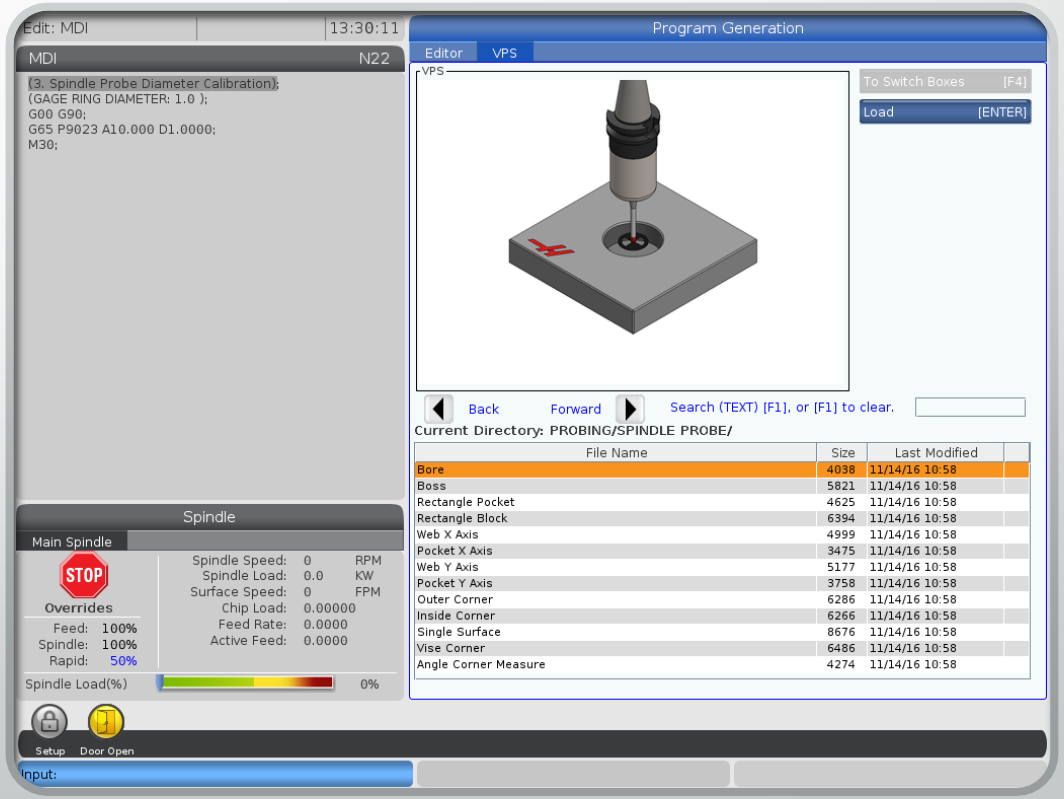
- **Probes** are used in the machine spindle
  - Utilize a stylus which contacts the part. After contact, the probe sends a signal and the control records the position.
    - The probes normal state is power off. Once the probe program is activated, the probe will receive a signal and power on. The probe LED lights will blink green when powered up.
- **What can probes used for?**
  - Work offsets
  - In process inspection
  - Final inspection before removing part from the machine

Probe Uses

- Probes are primarily used to set work offsets
  - This is a process that happens before cutting takes place.
  - Can be programmed to find corners, the center of a boss or a hole, and pockets.
  - The Visual Programming System (VPS) on the Haas Mills makes finding work offsets very simple.

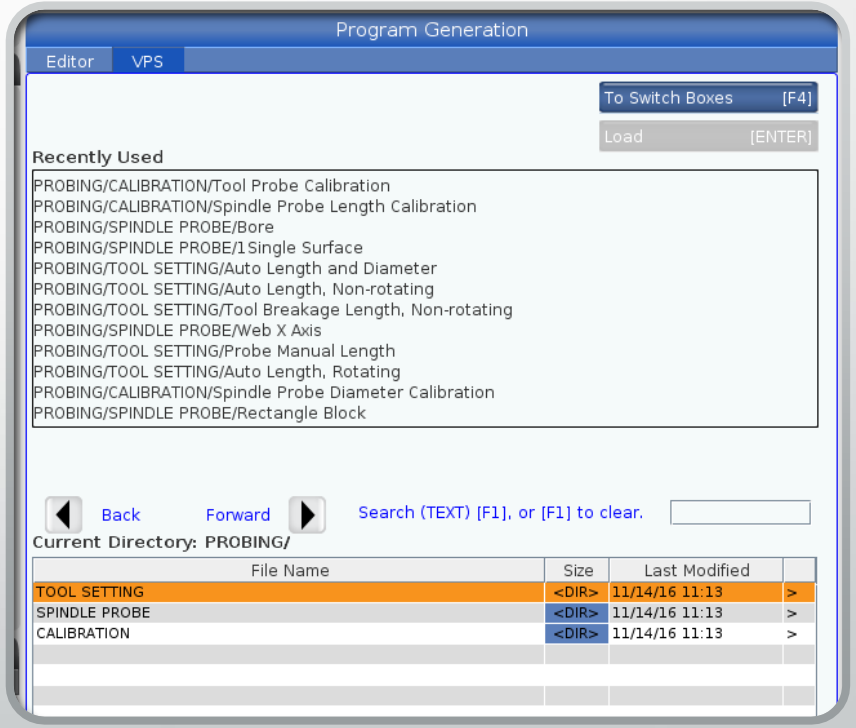
How is a Probe Programmed?

- To efficiently program probe cycles, Haas has incorporated probe cycle interfaces on the CNC control.
- This saves the programmer having to program the routines by hand.



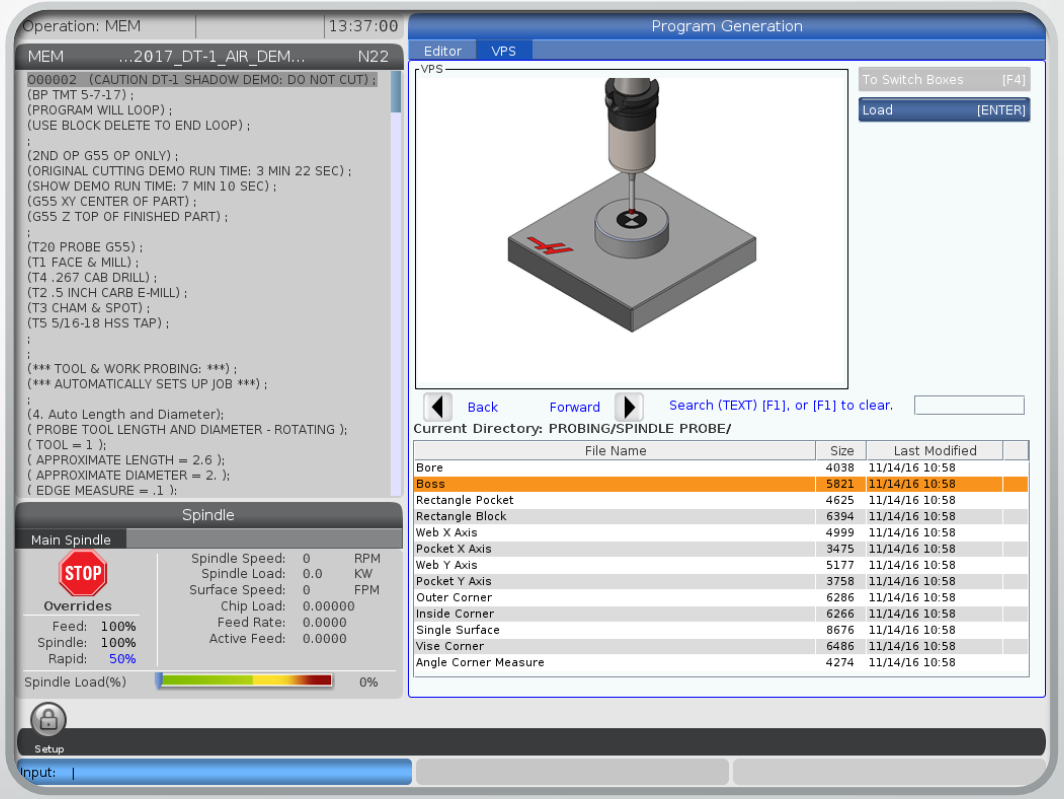
How is a Probe Programmed? Cont.

- The probe cycles can be found by entering the edit mode on the machine, then cursor over to the visual programming system(VPS) tab.
- Once in the VPS tab, spindle probe can be highlighted and the right cursor key is used to enter the tool setting directory.



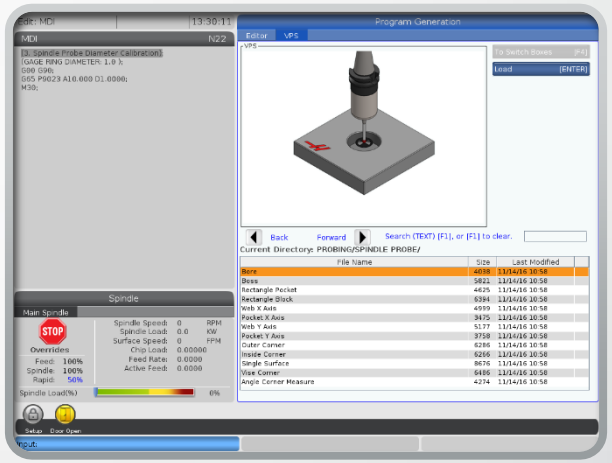
How is a probe programmed? Cont.

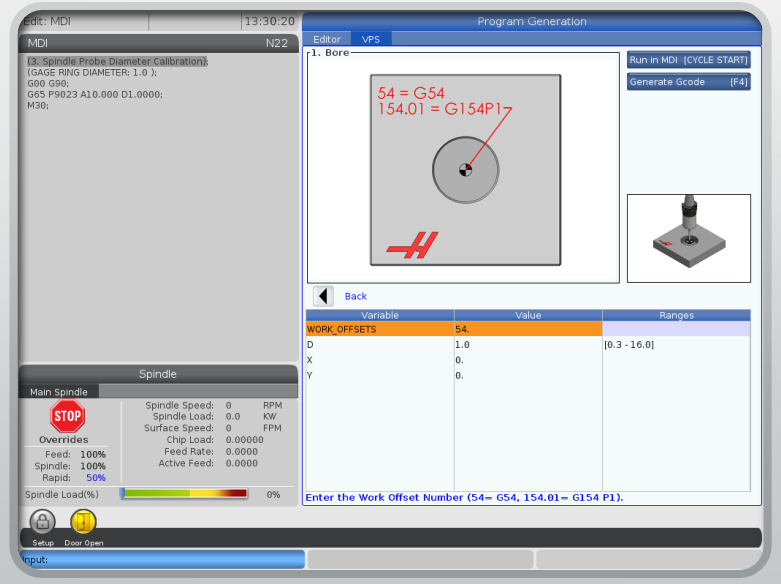
- Once in the probing directory there are twelve tool setting cycles to choose from:
  - Bore
  - Boss
  - Rectangle Pocket
  - Rectangle Block
  - Web X axis
  - Pocket X axis
  - Web Y axis
  - Pocket Y axis
  - Outer Corner
  - Inside Corner
  - Single Surface
  - Vise Corner
- Each cycle has a different purpose and will be reviewed in detail



Bore

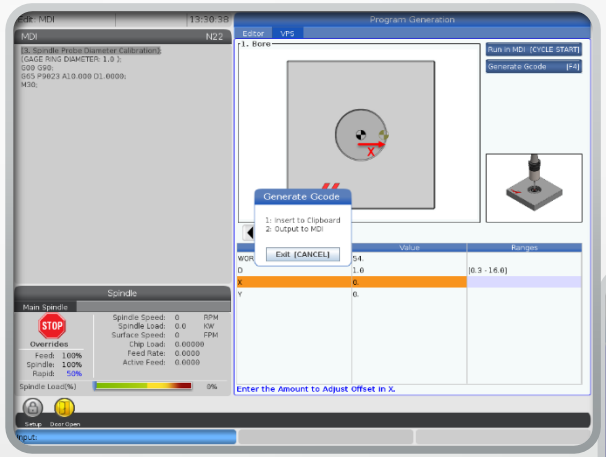
- Bore is a probe cycle used to find the center of a hole in a part.
- The work offset to set can be chosen and the diameter of the bore must be entered.
- The X and Y options are used to shift the work offset from the center of the bore once it is found.

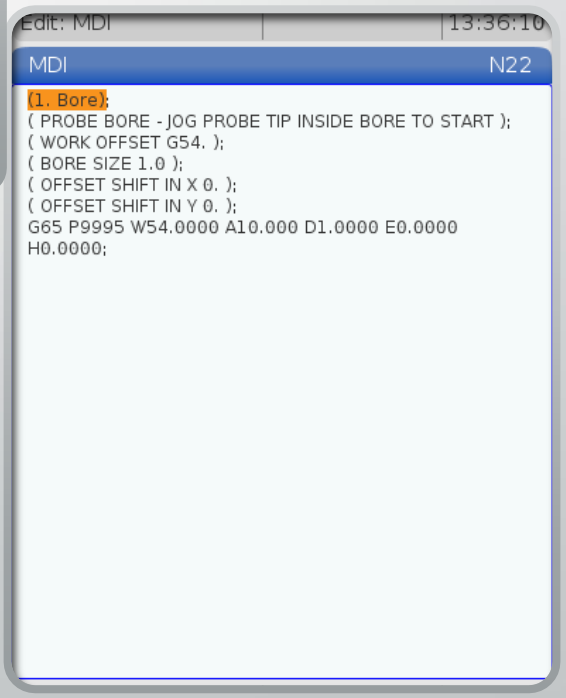




Bore

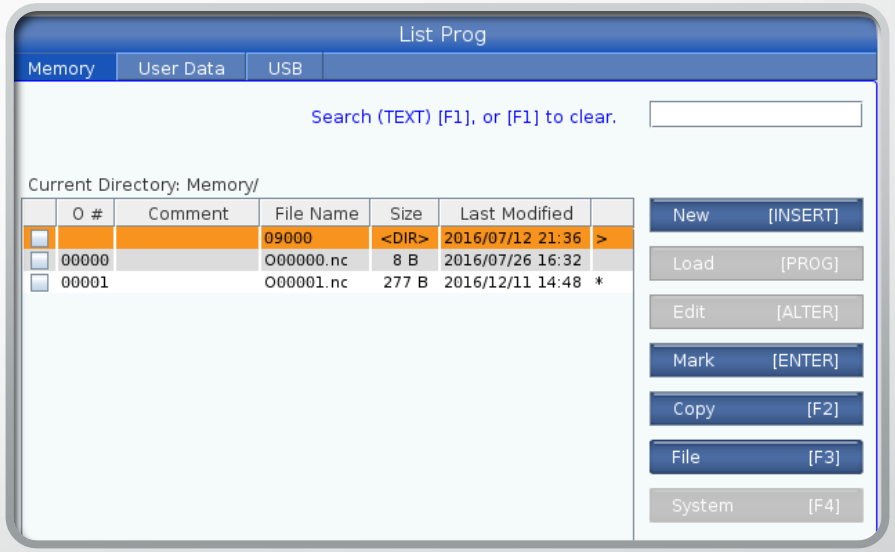
- After the information is entered, the code can be created by pressing cycle start. This will output the code to the MDI page.
- If F4 is used, the code can be inserted into a program or generated on the clipboard.
- Once the code is created and the tool is in position, the cycle can be executed.





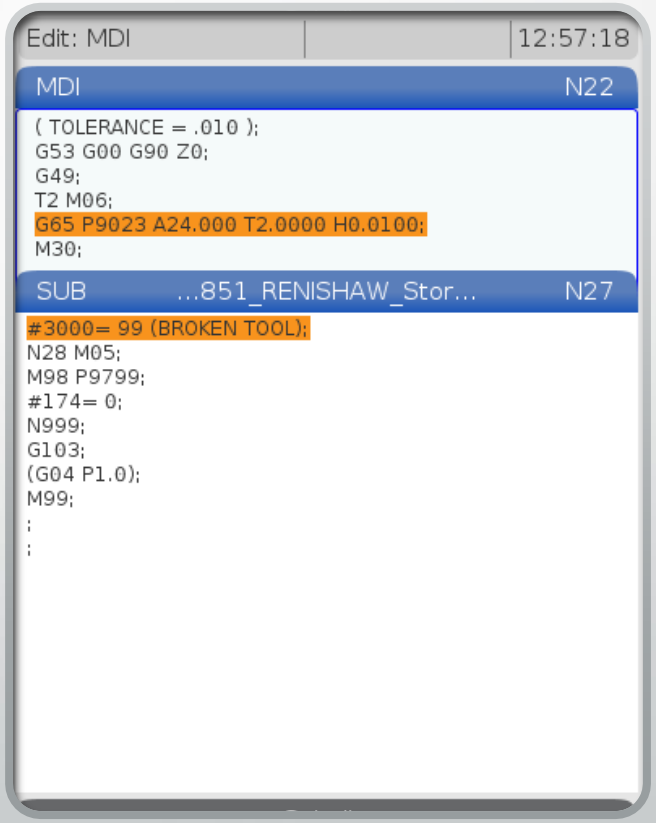
What is Really Being Created?

- The probe interfaces in the VPS directory are simply macro variable inputs.
- This means that a pre-written set of programs is stored in the machines control. In this case they are O9000 programs
- The user enters the inputs needed for their desired tool setting cycle. When the inputs are entered and the program is generated, the control is using the inputs to fill in blanks in the stored macro program



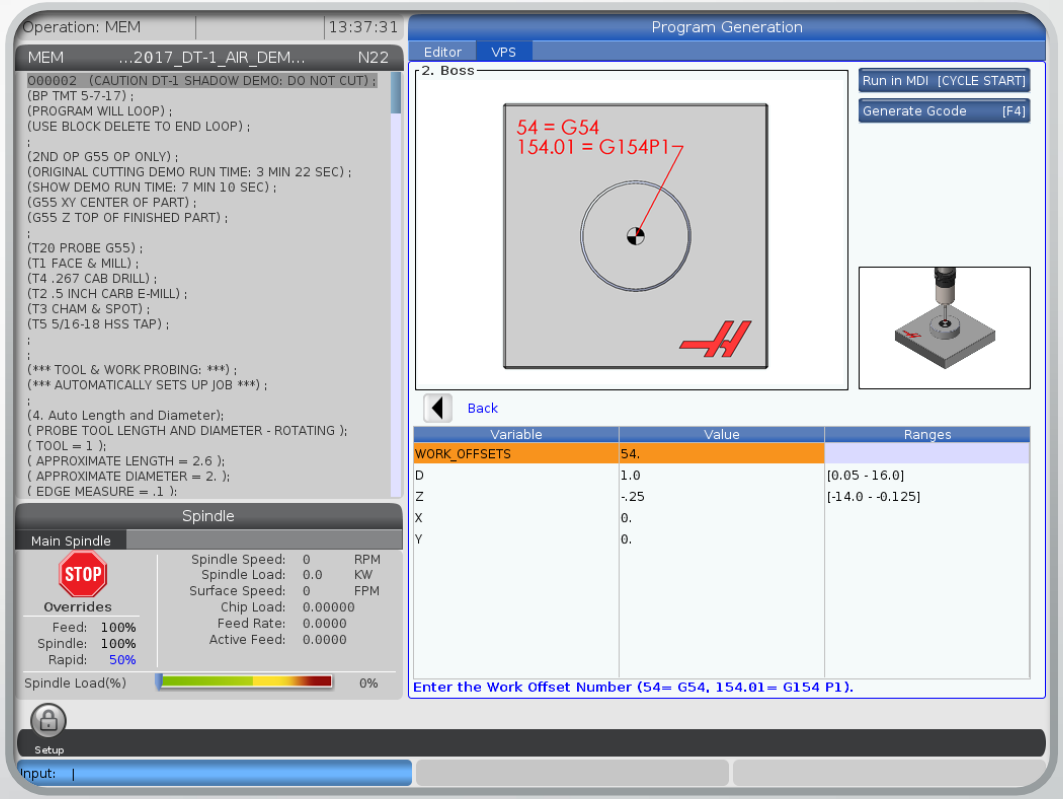
What is Really Being Created?

- The cycle that is output has the necessary variables to satisfy the stored macro routine.
- When the program is executed, it calls up the stored O9000 program to run.
- The O9000 program uses the variables from the main program to finish the tool setting cycle and store the data in the offset location.



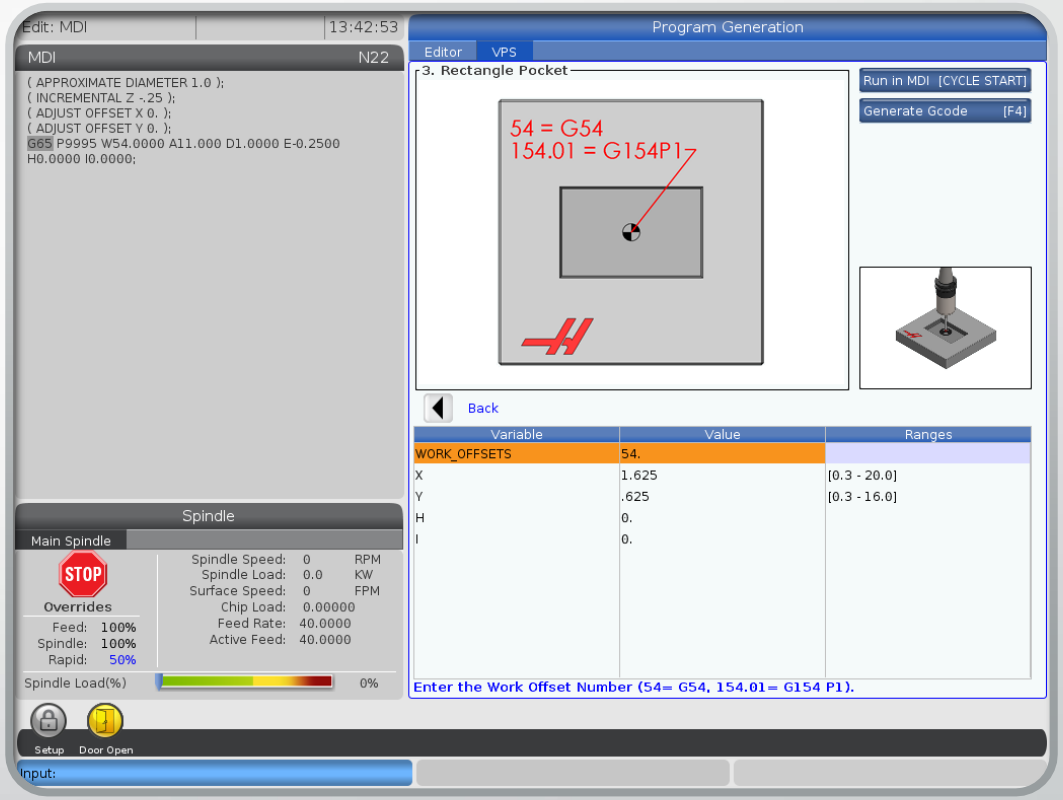
Boss

- Boss is a probe cycle used to find the center of a round feature that protrudes above the surface of a part.
- The following values must be entered for the cycle to work:
  - Work offset #
  - D – Diameter of the Boss.
  - Z – Depth of probe to travel when probing the boss.
  - X – Offset amount after finding the boss center.
  - Y – Offset amount after finding the boss center.



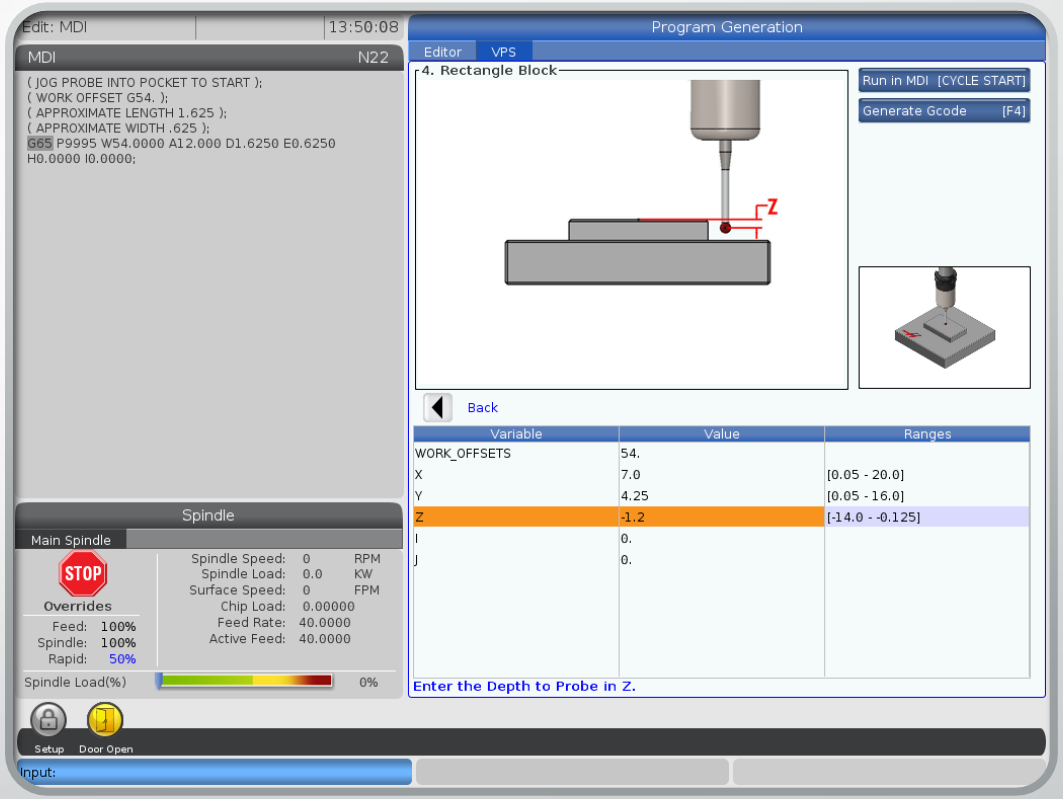
Rectangle Pocket

- Rectangle Pocket is a probe cycle used to find the center of a pocket in the X and Y axis
- The following values must be entered for the cycle to work:
  - Work offset #
  - X – Width of the pocket to be probed
  - Y – Width of the pocket to be probed
  - H – Offset amount in X after probing
  - I – Offset amount in Y after probing



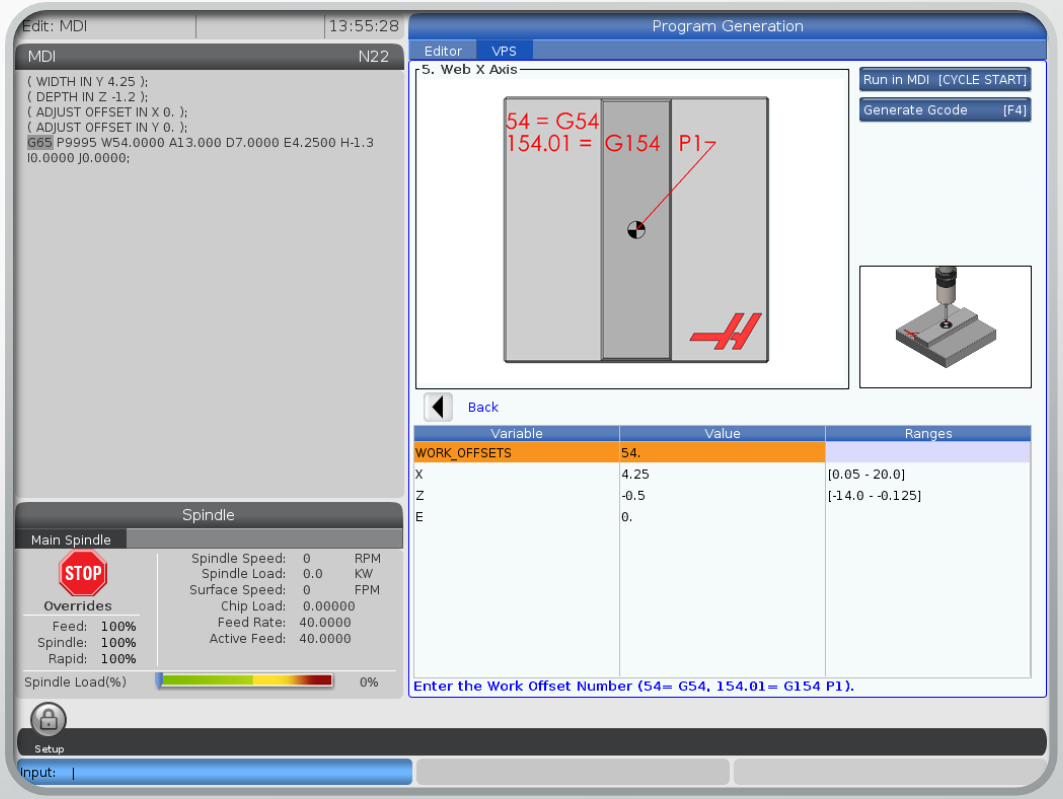
Rectangle Block

- Rectangle block is a probe cycle used to find the center of a block in the X and Y axis
- The following values must be entered for the cycle to work:
  - Work offset #
  - X – width of the block to be probed
  - Y – width of the block to be probed
  - Z – depth to travel to probe block
  - I – Offset amount in X after probing
  - J – Offset amount in Y after probing



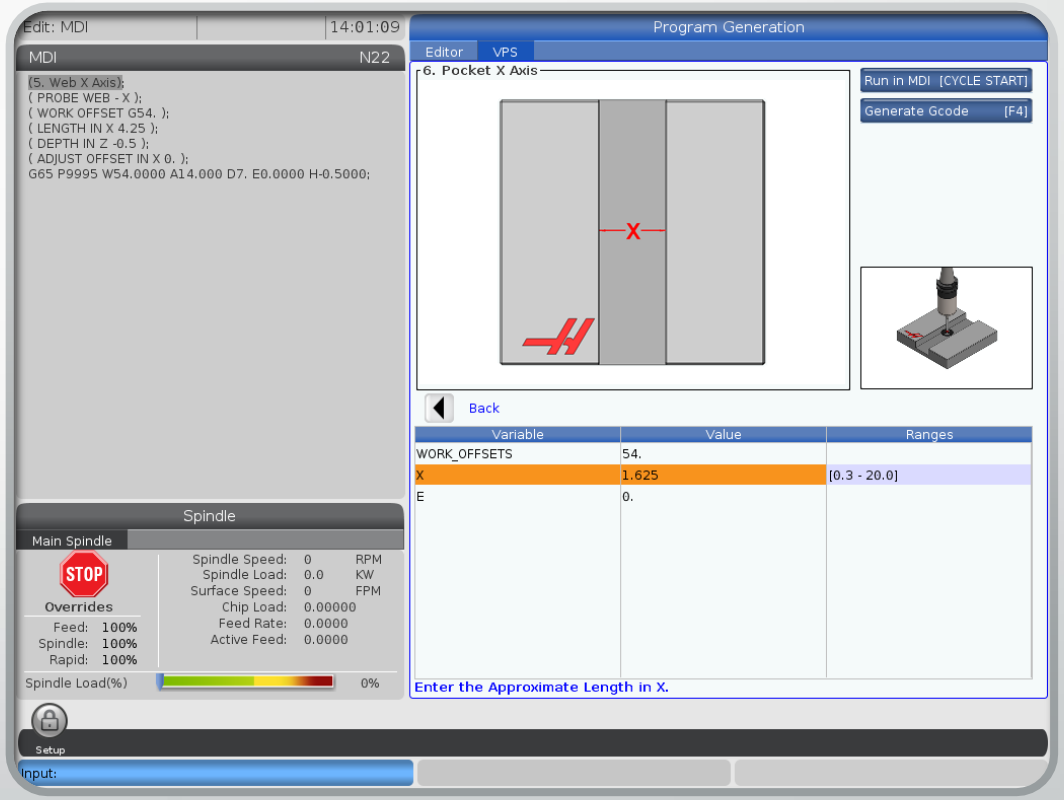
Web X Axis

- Web X axis is a probe cycle used to find the center feature above the part surface in the X axis.
- The following values must be entered for the cycle to work:
  - Work offset #
  - X – width of the web to be probed
  - Z – depth to travel to probe block
  - E – Offset amount in X after probing



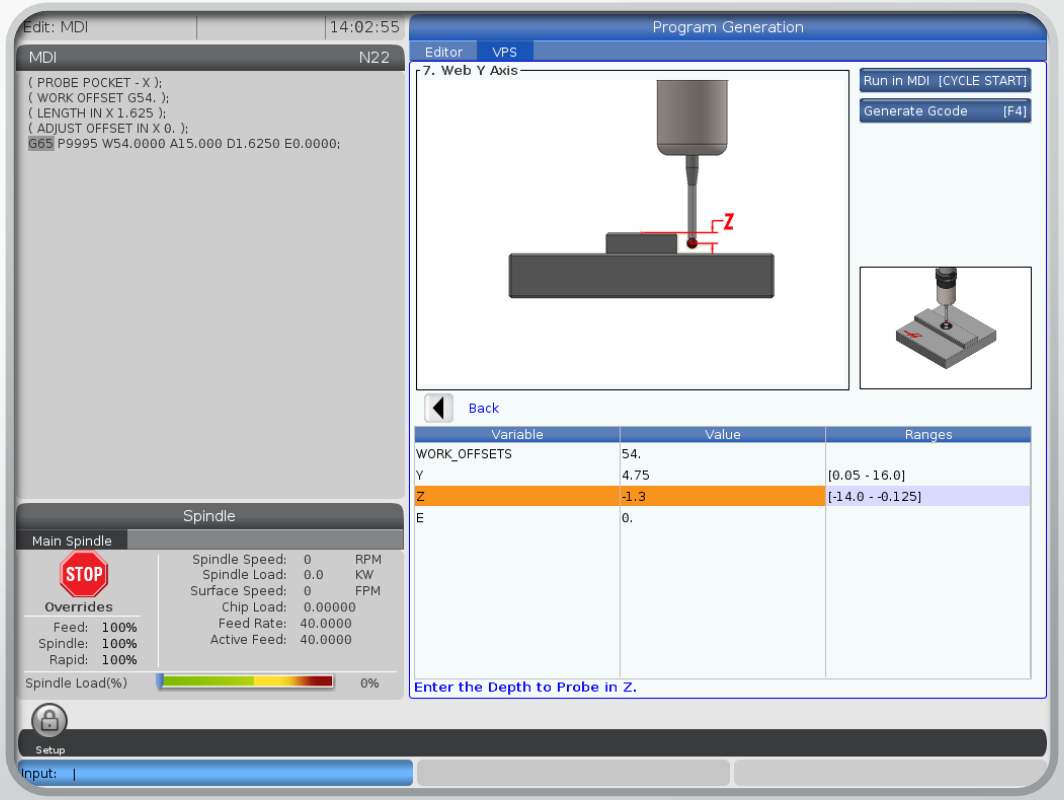
Pocket X Axis

- Pocket X axis is a probe cycle used to find the center slot or pocket in the X axis only
- The following values must be entered for the cycle to work:
  - Work offset #
  - X – width of the block to be probed
  - E – Offset amount in X after probing



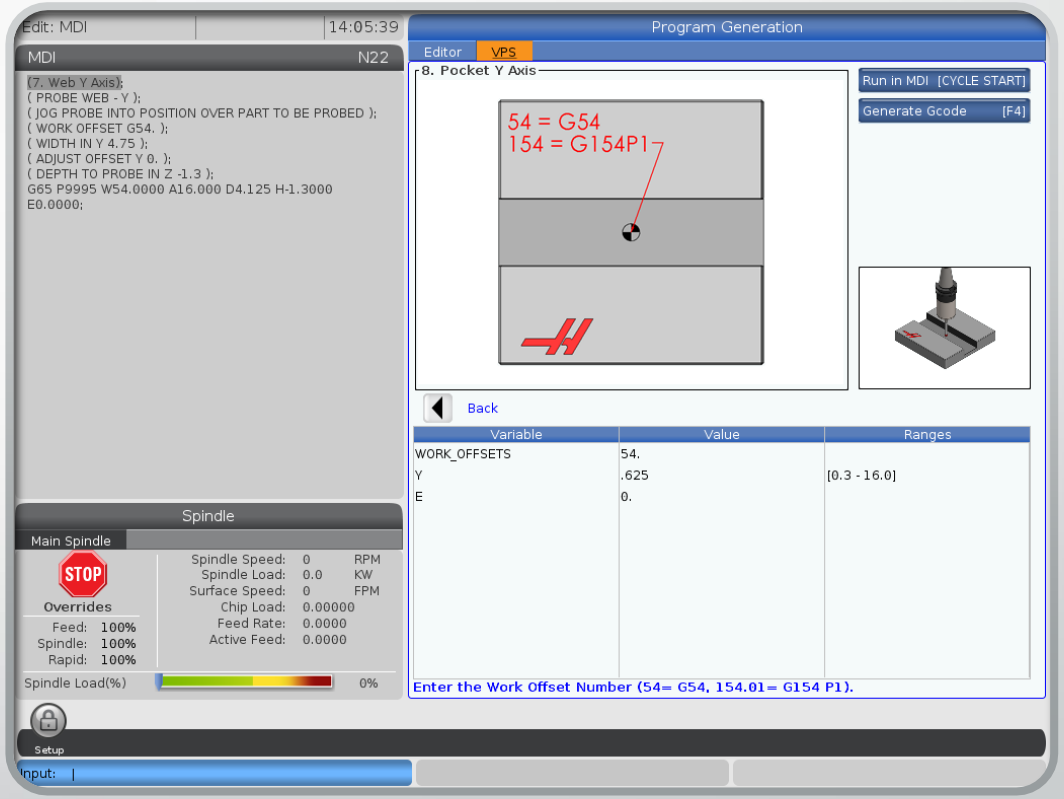
Web Y Axis

- Web Y axis is a probe cycle used to find the center feature above the part surface in the Y axis
- The following values must be entered for the cycle to work:
  - Work offset #
  - Y – width of the web to be probed
  - Z – depth to travel to probe block
  - E – Offset amount in X after probing



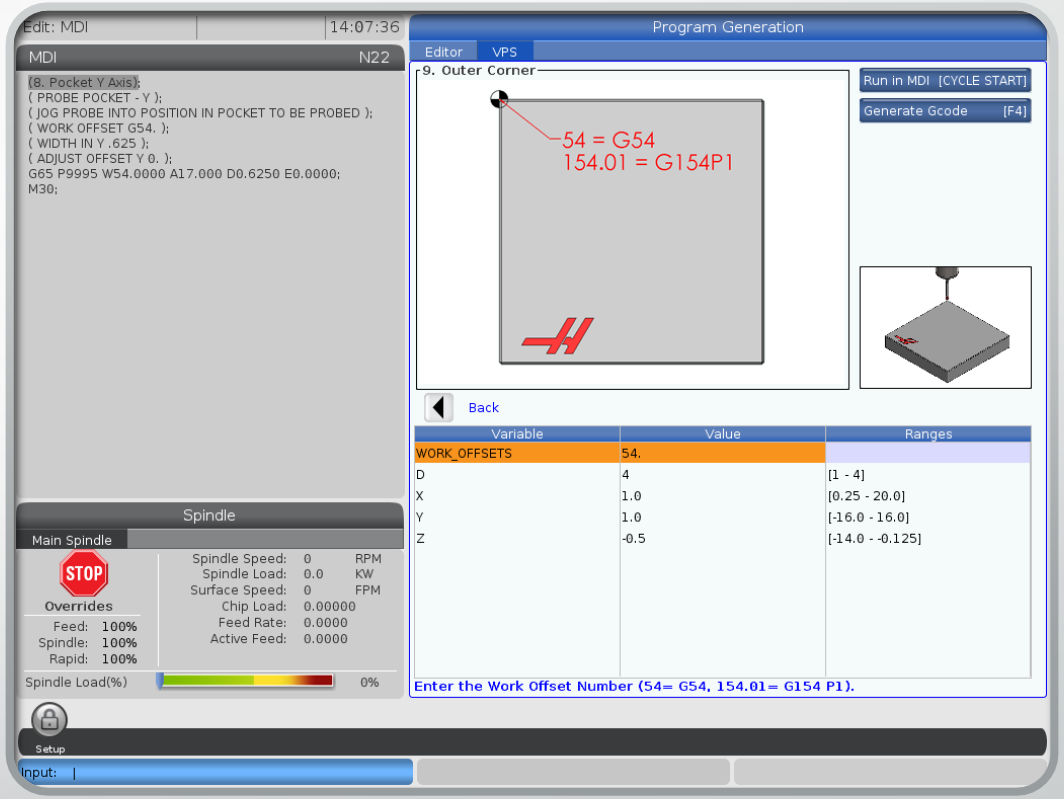
Pocket Y Axis

- Pocket Y axis is a probe cycle used to find the center slot or pocket in the Y axis only.
- The following values must be entered for the cycle to work:
  - Work offset #
  - X – width of the block to be probed
  - E – Offset amount in X after probing



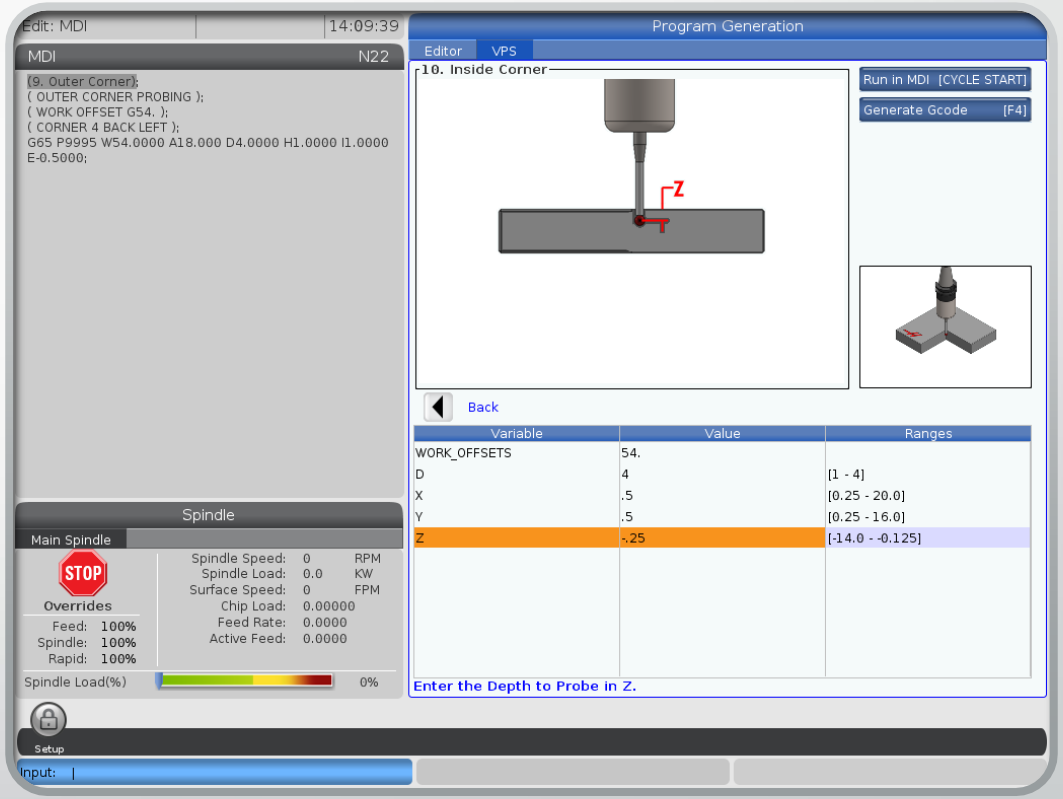
Outer Corner

- Outer Corner is a probe cycle used to find the corner of a part in the X and Y axis
- The following values must be entered for the cycle to work:
  - Work offset #
  - D – the corner being probed
  - X – travel amount in axis to probe surface
  - Y – travel amount in axis to probe surface
  - Z – depth to travel to probe X or Y surface



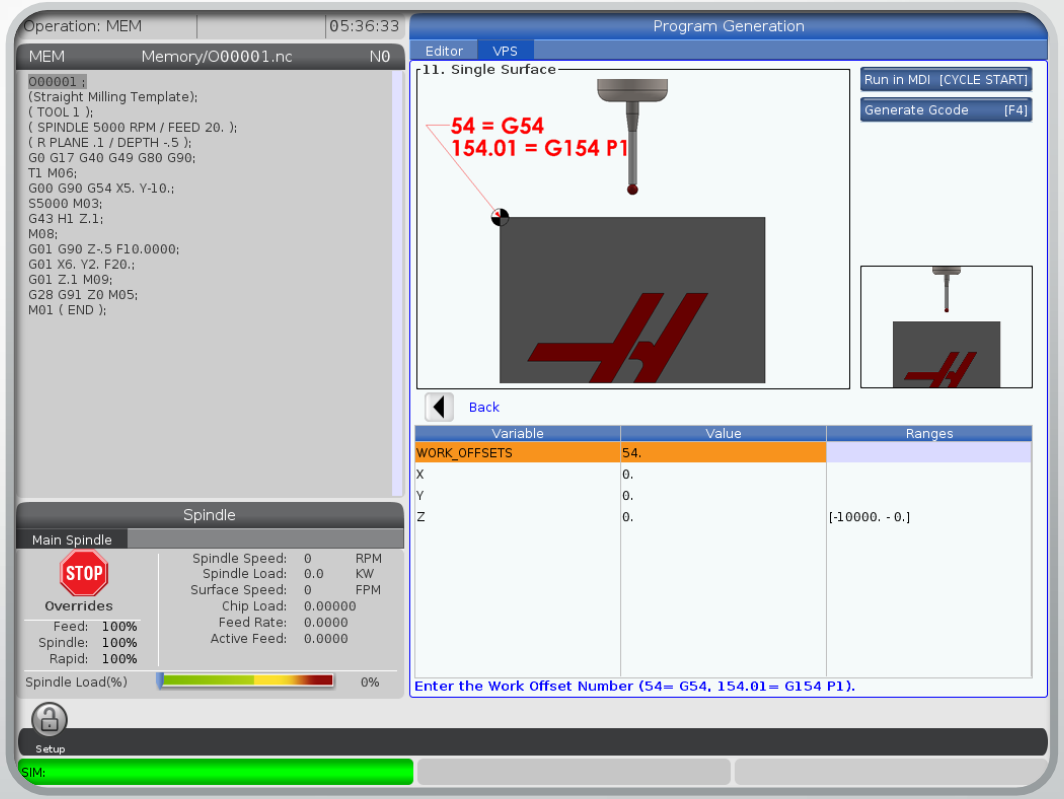
Inside Corner

- Inside Corner is a probe cycle used to find the X and Y axis intersection of an internal corner in a part.
- The following values must be entered for the cycle to work:
  - Work offset #
  - D – the corner being probed
  - X – travel amount in axis to probe surface
  - Y – travel amount in axis to probe surface
  - Z – depth to travel to probe X or Y surface



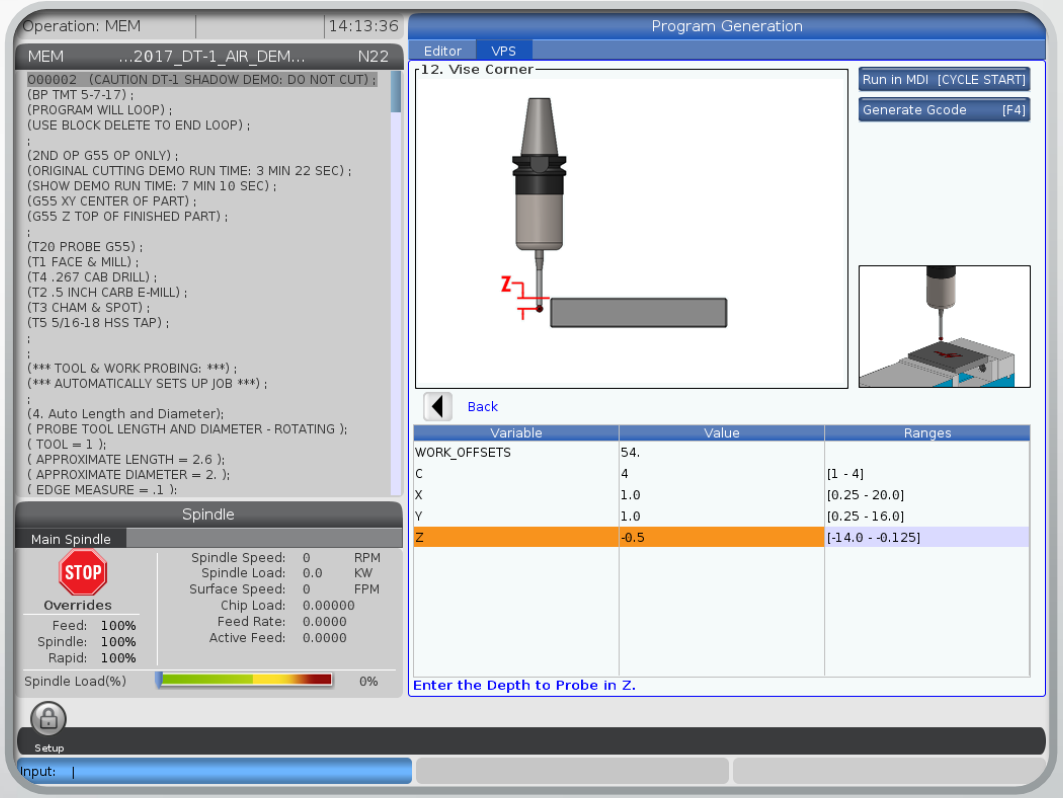
Single Surface

- Single surface is a probing cycle used to probe only one surface in one direction of travel.
- The following values must be entered for the cycle to work:
  - Work offset #
  - The axis being probed: X, Y, or Z



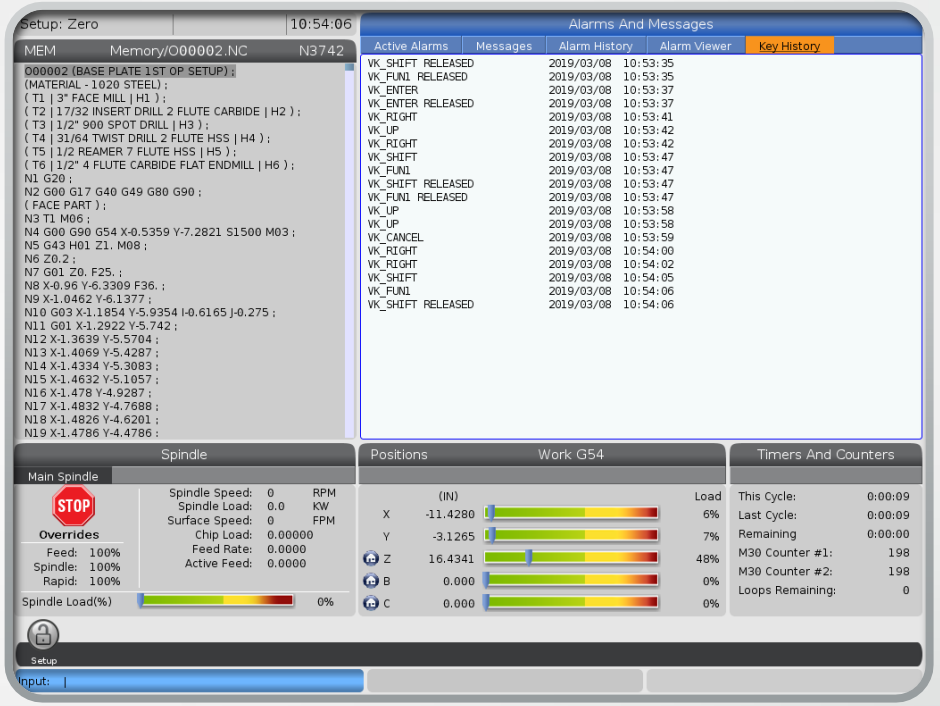
Vise Corner

- Vise Corner is a probing cycle used to pick up the Z surface of the part, then find both the X and Y axis of the part.
- The following values must be entered for the cycle to work:
  - Work offset #
  - C – the corner being probed
  - X – travel amount in axis to probe surface
  - Y – travel amount in axis to probe surface
  - Z – depth to travel to probe X or Y surface



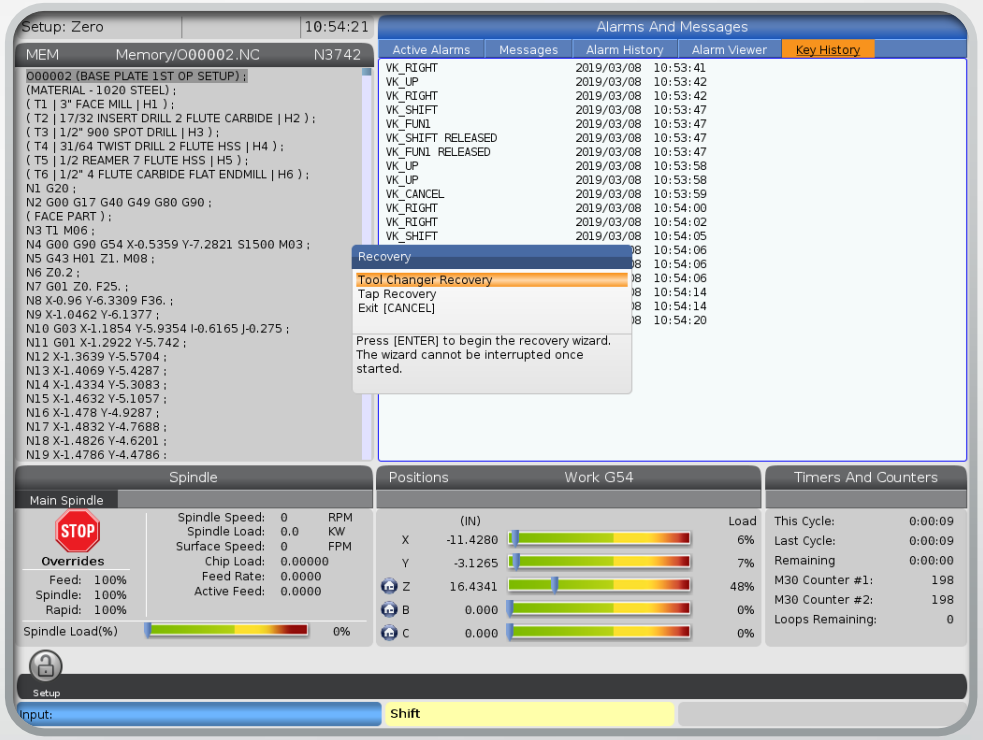
Tool Change Recovery

- Recovery is used to recover from an alarm or to prevent an inevitable error. A couple common uses are to recover from a bad tool change or from an incomplete tapping cycle.



Recovery

- After recovery is entered, the following are options:
  - Tool change recovery
    - Used when the tool changer is out of position or is in an alarm state.
  - Tap recovery
    - Used when a tap has stopped mid cycle in a hole.



Thru Spindle Coolant (TSC)

- All Haas mills have the capabilities of utilizing thru spindle coolant. This feature sends high pressure coolant thru the spindle and tool and directs the stream at the point of contact in the cut.
  - This is the most direct application of coolant, it is also the best for evacuating chips.
  - TSC requires tool holders and tools that are capable of TSC.
  - TSC is activated with M88/M89.

Photo courtesy of Haas Automations Inc.
http://www.haas.com.mx/pdf/haas_cmv2016_brochure.pdf

High Speed Machining (HSM)

- High speed machining(HSM) relies on fast movement with a small step over and a large depth of cut. HSM uses "look ahead" or the ability to read the program before the movements happen. This allows for the execution of a large amount of program at high rates of feed without interruption.
  - Without HSM, the machine accelerates and decelerates between each line of code. This means the machine may never reach the programmed feed rate.
  - With HSM, the control looks at the angle between two blocks of code and computes the optimum acceleration and deceleration to navigate from one block to the next.
    - This keeps the machine at the programmed feed rate longer.

Photo courtesy of Haas Automations Inc.
http://www.haas.com.mx/pdf/haas_cmv2016_brochure.pdf

Auto Air Blast

- Auto air blast sprays air directly on the tool. This feature allows for an alternative tool cooling and chip evacuation method. Air blast can cool the tool while avoiding the large thermal changes in tool temperature that are seen with coolant.

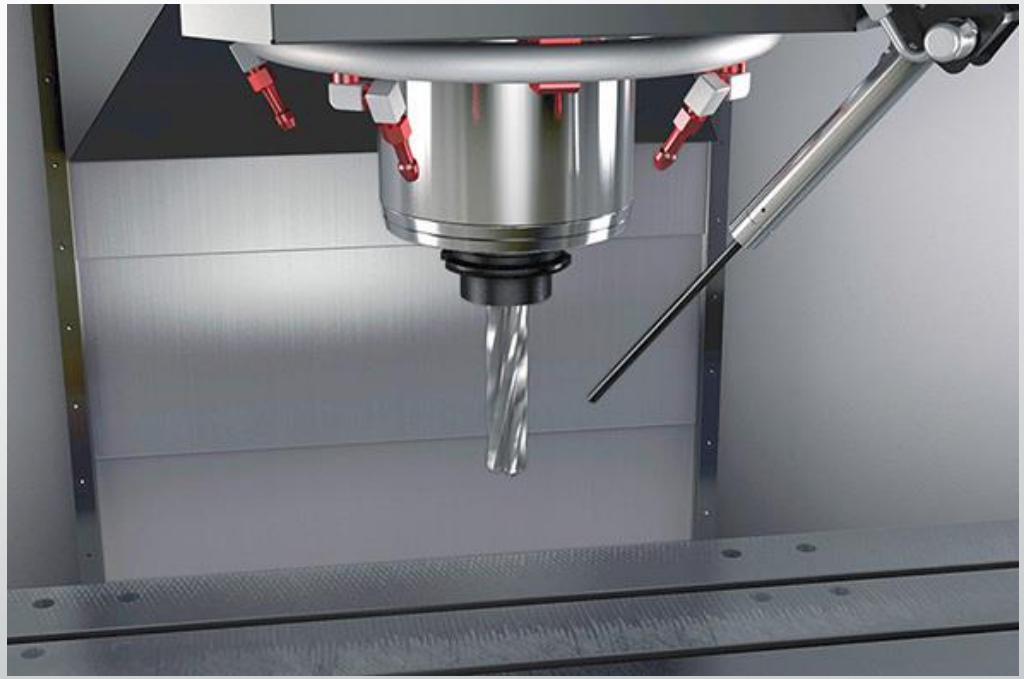
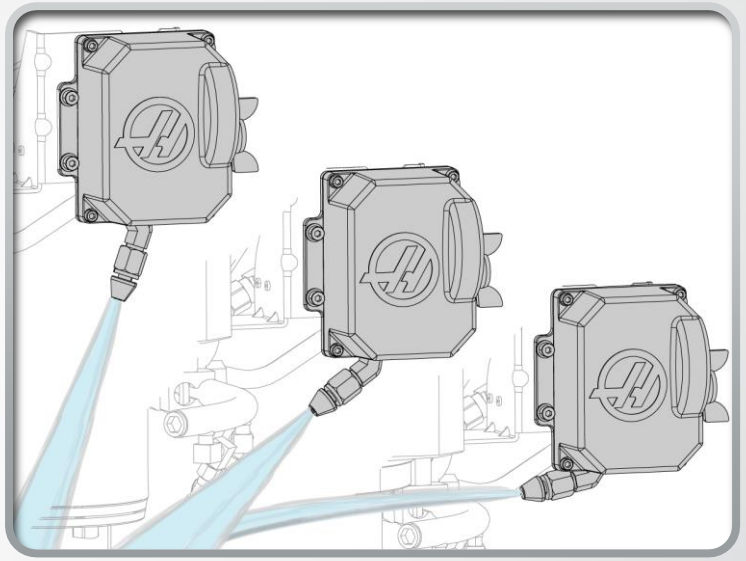


Photo courtesy of Haas Automations Inc.
https://www.haascnc.com/productivity/chip-and-coolant-management/aag.html

Programmable Coolant



- Programmable coolant is a coolant nozzle that can be programmed to a specific position.
- This allows for the coolant to be adjusted automatically from tool to tool without operator intervention.
- The "CLNT UP" and CLNT DOWN" buttons can be used to set the desired position. Then then position for each tool can be stored in the tool offsets page.

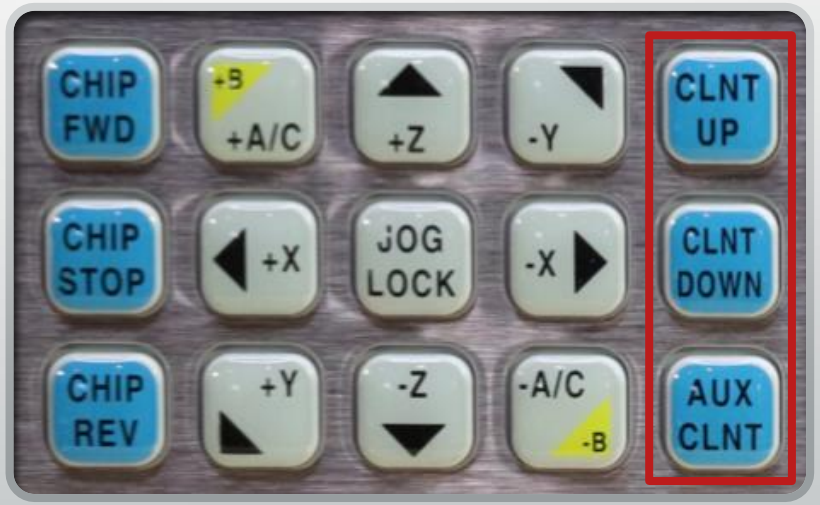
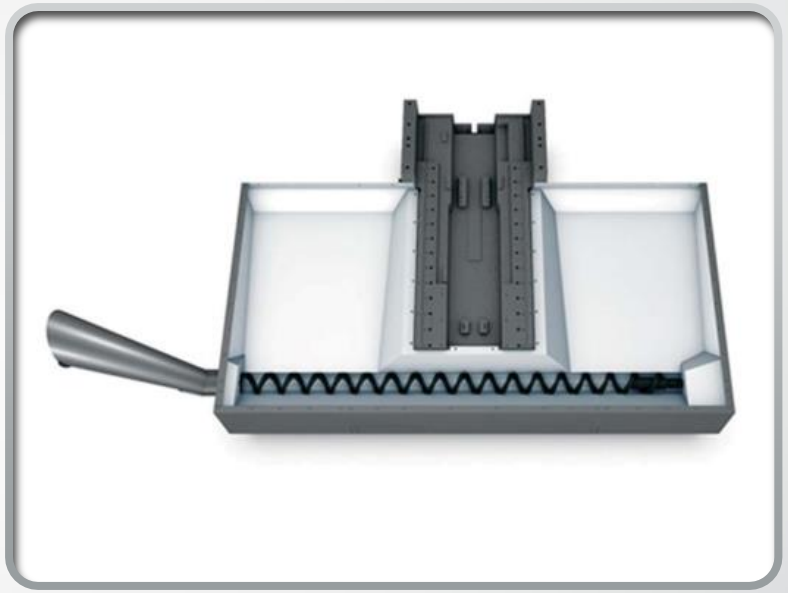


Photo courtesy of Haas Automations Inc.
https://www.haascnc.com/service/troubleshooting-and-how-to/troubleshooting/programmable-coolant---troubleshooting-guide.html

Chip Auger

- The chip auger is used to evacuate the cut chips from the machine. The screw style auger pushes the chips out of the machine, up the shoot, and then into a chip bin.
- The auger can go in reverse if it becomes obstructed.
- The auger can be activated on the control panel with the three highlighted buttons. The auger can also be programmed (M31) to run during a cutting cycle.



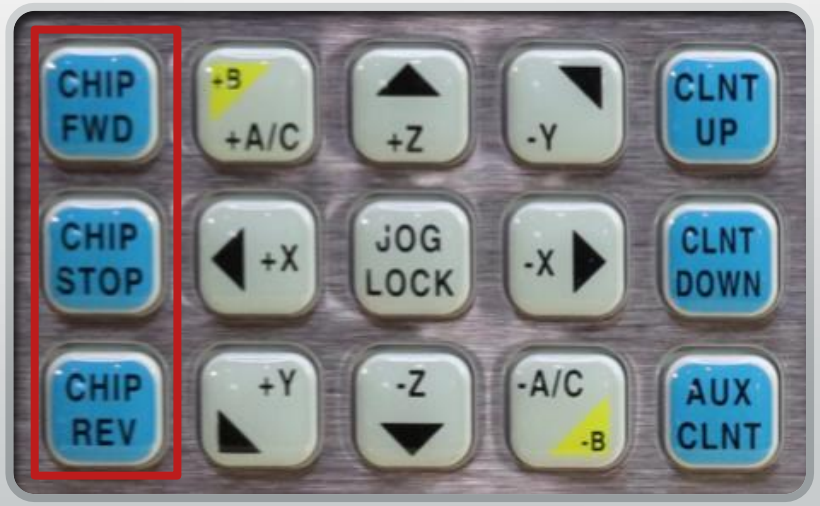


Photo courtesy of Haas Automations Inc.
https://www.haascnc.com/productivity/chip-and-coolant-management/auger.html