



Welcome to Trident Machine Tools

Haas Lathe Setup and Operation





Haas Lathe Setup and Operation

- This half day course is designed to familiarize the user with the basic set-up and operation of a Haas CNC lathe. Machine settings, turret set-ups, machine offsets and tool presetters will covered in this course.

Schedule

- Introductions
- Classroom:
 - Machine components
 - Turret types
 - Control settings
 - Tool holders
 - Loading tools
- Break
- Showroom
 - Set-up Demo
 - Tool setting

Haas Lathe Overview

- This tutorial is designed to train the user with skills necessary to operate a Haas CNC lathe.
- 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
 - 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
 - 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 further than the part length in Z.
 - 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 chuck or turret.
 - Never remove chips with your hands.
 - Keep skin contact with cutting fluids to a minimum.





Machine Components

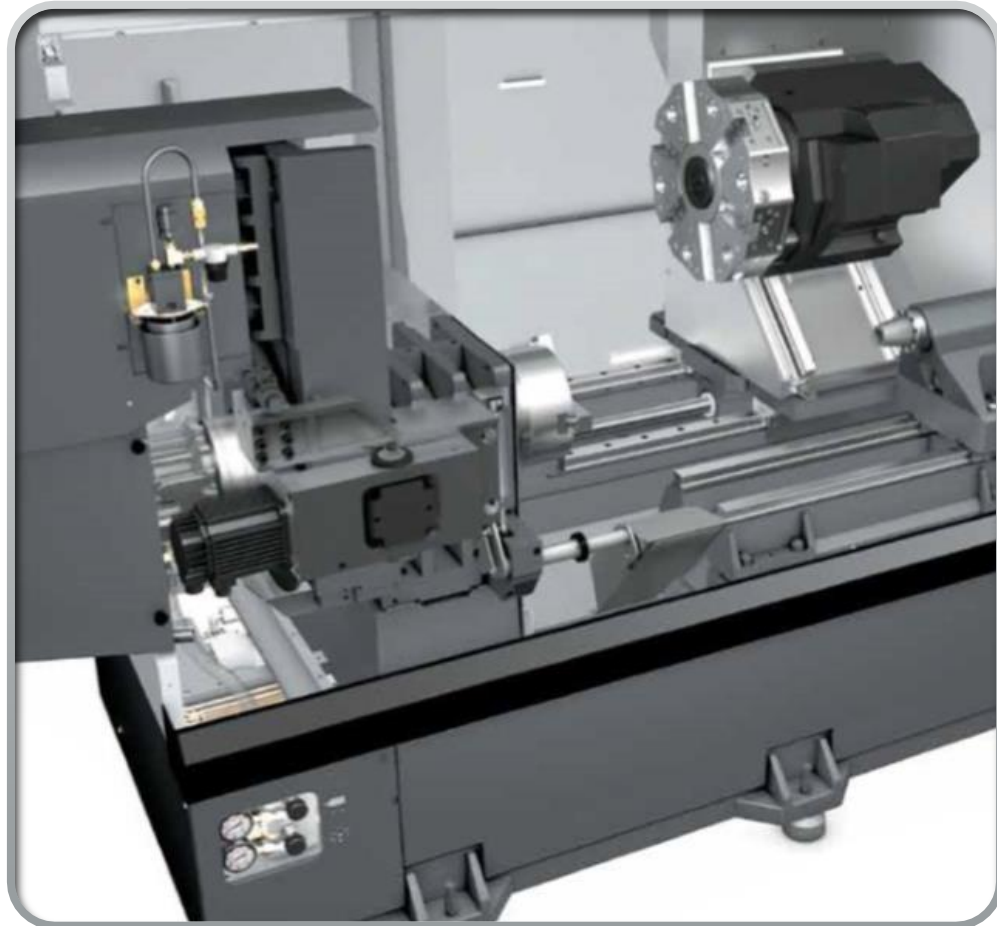
Lathe

Machine Components

- Major components of a CNC lathe
 - Frame
 - Bed
 - Headstock
 - Chuck
 - Turret
 - Control

Lathe Bed

- The bed is the base of the lathe. All the machine components are bolted to the bed. The machine is only as durable as the frame.



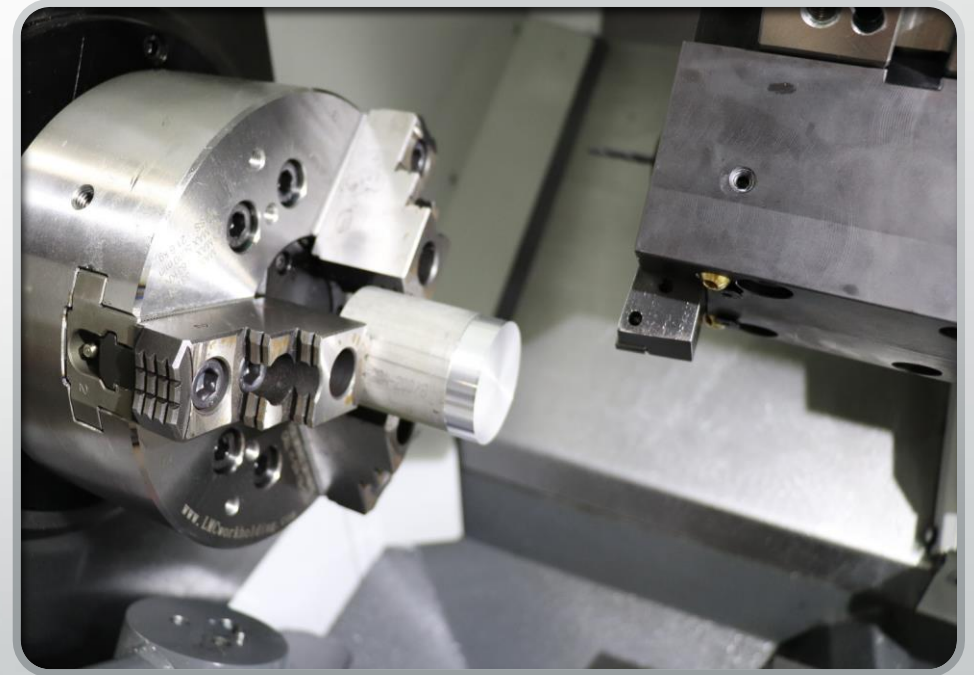
Head Stock

- The head stock consists of the spindle, spindle tube, spindle bearings and spindle motor. The work holding device, usually a chuck, attaches to the spindle tube.



CNC Turning Centers

- When turning is performed, the part rotates and the tool is stationary.
 - The work piece is held in a chuck, which spins at RPM's up to 4000 or 6000 depending on the lathes chuck size.
 - Smaller chucks can rotate at higher RPM. Larger chucks operate at lower RPM because of the rotating mass.
- Lathes are often defined by their chuck size.
 - Chucks are often found in 2" increments starting at 6" diameter.



Work holding

- CNC lathes use a hydraulic chuck to hold the work piece.
- Chucks range in size and are sized to the CNC lathe.
- The hydraulic operation allows the chuck to be operated with a foot pedal or programmed with M-codes.
 - This allows for automatic operation of the machine when parts are run in production.
 - This allows for the use of a bar puller of bar feeder to be used in a production setting.



Work holding

- The chuck operates on hydraulic pressure from a pump on the machine.
 - The pressure can be adjusted based on the operation being performed. Roughing operations can use holding pressure up to 300 or 400 psi.
 - Max clamping pressure is dependent on the machine model, the max pressure of the machine is listed on the side of the machine.
 - Finishing operations may use as little as 100 psi, this depends on the features of the part.



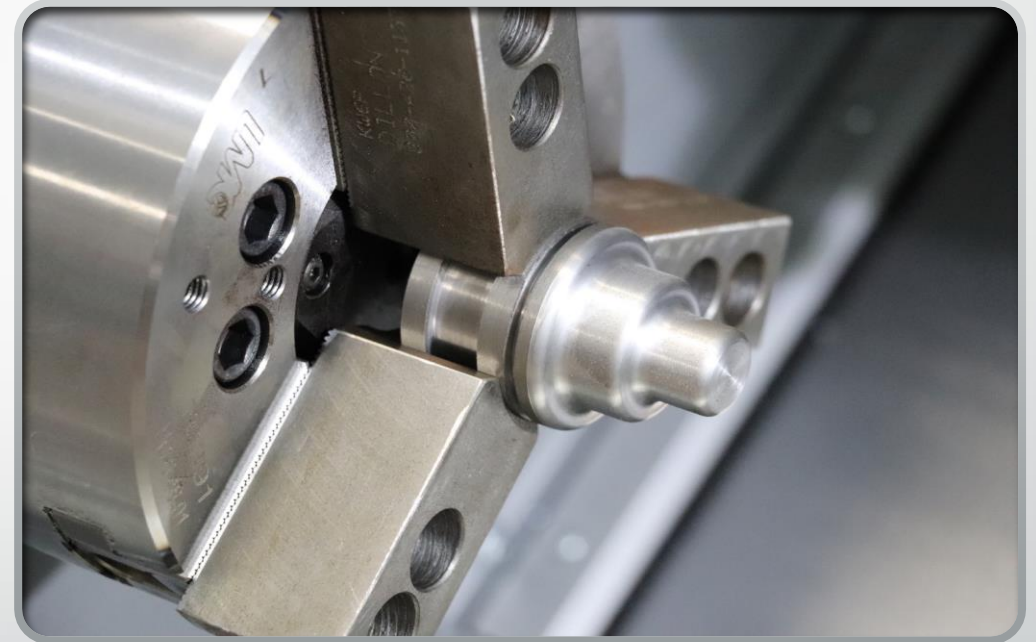
Work holding

- The hydraulic chucks have a limited stroke, or travel, of the jaws.
 - Because of this, the removable jaws need to be adjusted to match the size of the part being held.
- The jaws on a chuck are located with $60^\circ \times 1.5\text{mm}$ serrations. These serrations interlock the master and removable jaws when fastened together.
 - The jaws are held on with a t-nut and two cap head screws.
 - Remember to grease the chuck as specified by the maintenance schedule.



Types of jaws

- Jaws for a CNC lathe can be broken into two categories:
 - Hard jaws
 - these are a hardened, ground jaw that can be used for both OD and ID set-ups.
 - These jaws are ideal for set-ups that hold on rough stock. The jaws could dig into turned surfaces and may have runout.
 - Soft jaws
 - These jaws can be made from soft steel or aluminum .
 - They can be used for both OD and ID work.
 - Jaws can be bored to hold a specific diameter. This makes the jaws application specific and ideal on 2nd operations.
 - After boring, the jaws will clamp on the part without ruining a turn surface.
 - The boring operation will ensure the part runs concentric to the spindle.



Installing Jaws

- Before installing the jaws, all of the mating pieces should be clean and free of debris.
- The stroke of the chuck must be factored into the jaw installation process. Installing the jaws with hand tight screws and checking to see if the part fits.
 - Counting the number of teeth exposed on each master jaw or measuring jaw protrusion is a good way to make sure all three jaws are put on in the same position.
 - Jaws are labeled and should correspond to the correct number on the chuck.



Installing Jaws

- Once the jaws are in the correct position, tighten the jaws.
- A wrench can be used on the jaw to hold the chuck stationary while tightening the screws. Doing this ensures there is no added pressure to the mechanical components while the jaws are being tightened.
- After the jaws are tight, an indicator can be placed on the part to check for runout. Hard jaws will have runout because they have not been cut true to the spindle.
 - If a jaw is off one serration there will be .060" runout. Finding the high spot will indicate which jaw is off.



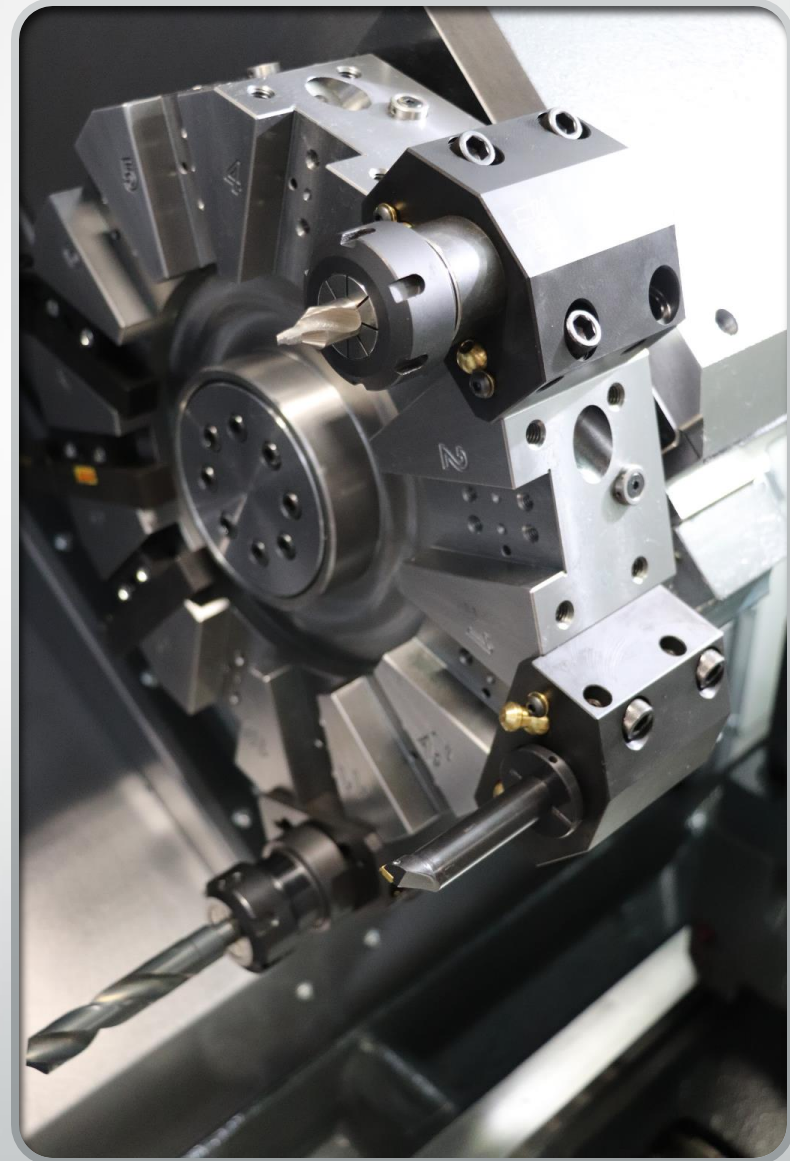
Turret

- The turret allows for the storage and use of multiple tools to complete a job. This also allows for uninterrupted operation of the machine in automatic mode.
 - There are three types of turrets
 - Bolt on tool turret (BOT)
 - VDI turret
 - Base mount tool turret (BMT)



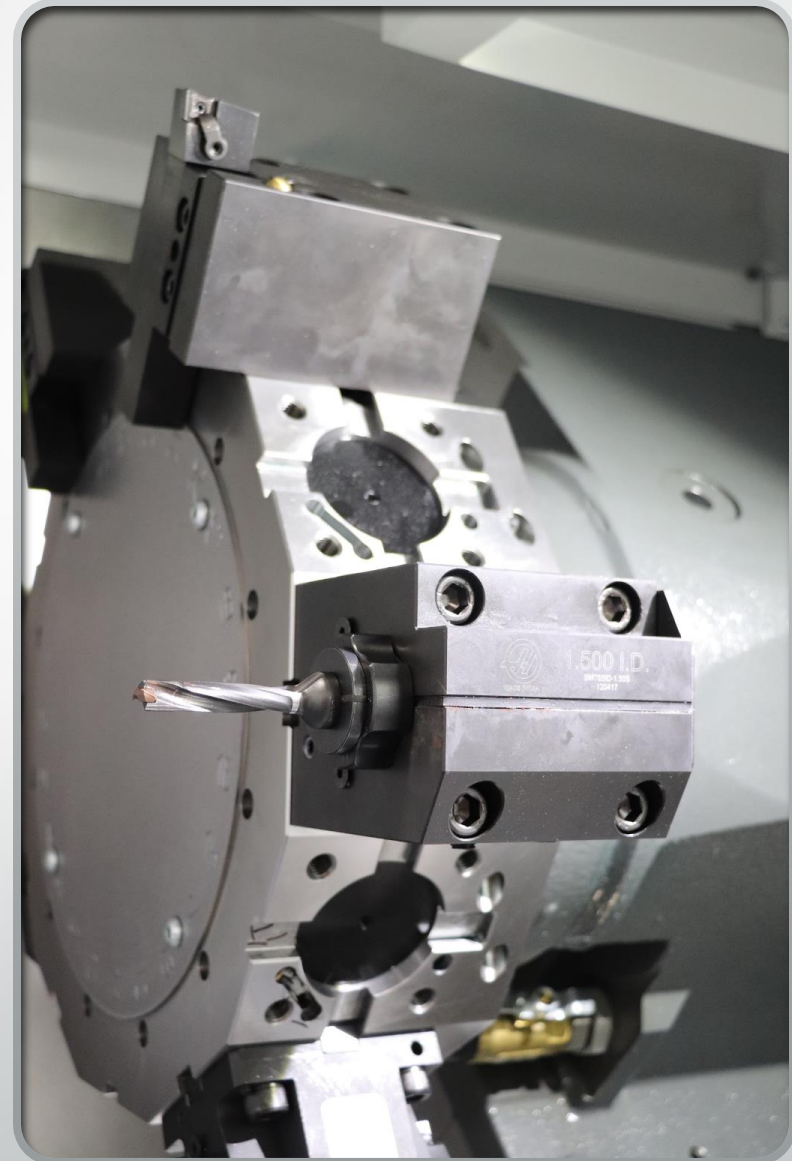
Bolt on Turret

- Bolt on turret (BOT)
 - BOT turrets have a space at every position for an OD tool to bolt on with a basic wedge system.
 - BOT turrets can also have an ID holder bolted onto the turret in any position.
 - This turret style is the base style and normally comes with the ability to hold 12 tools.



Base Mount Turret

- Base Mount Turret (BMT)
 - BMT turrets require all tool holders to use a tooling block, either OD or ID.
 - These holders are keyed and interlock with the turret.
 - This makes setting up of tools fast and accurate.
 - These turrets allow for the use of live tooling that is driven internally, through the turret.
 - These turrets come in sizes with 12 and 24 positions.



VDI turret

- VDI turrets are a quick change tooling system. These turrets allow for rapid change over of tooling between jobs.
 - VDI turrets have 12 tool positions



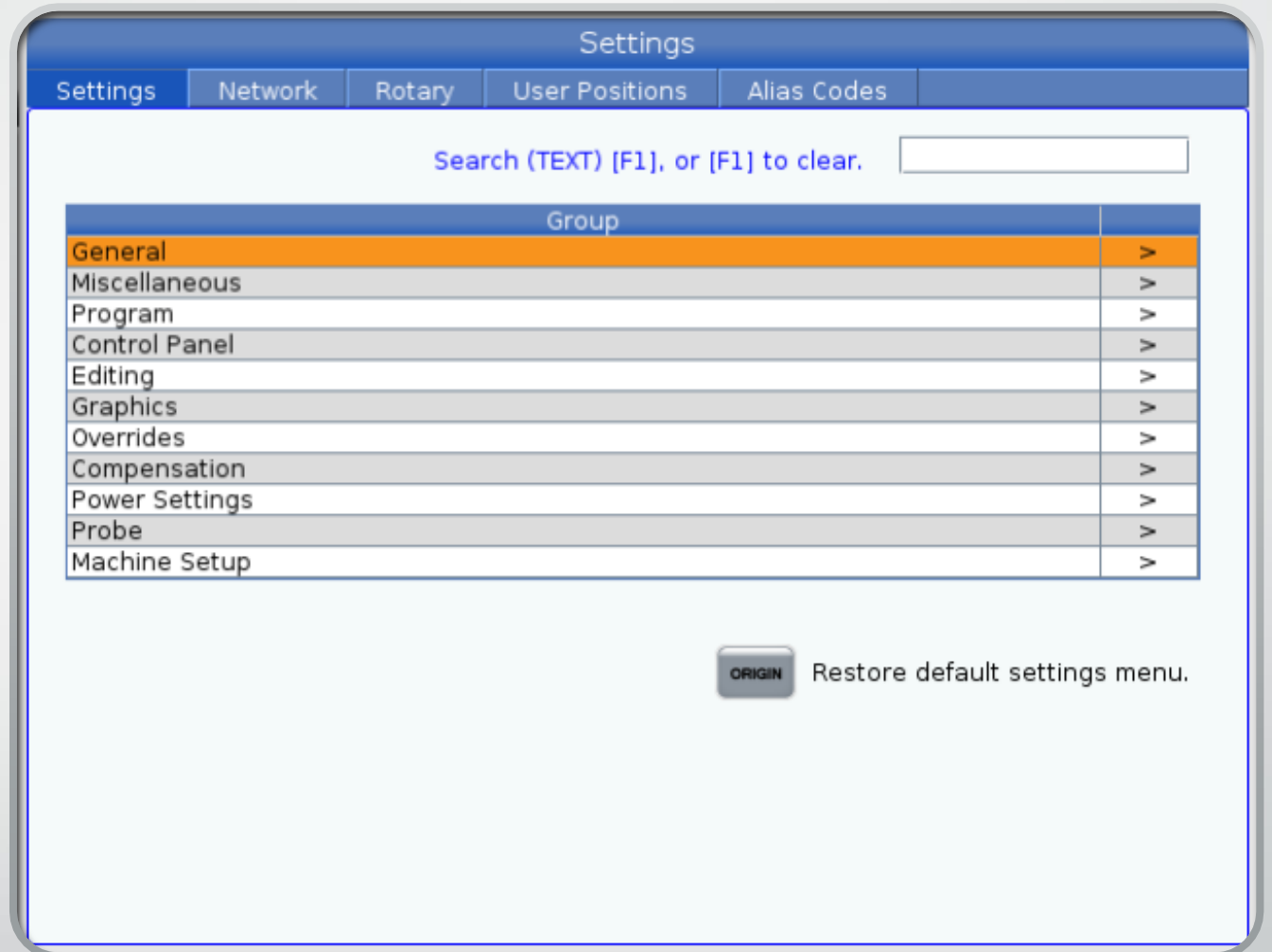
Control Pendant

- The control pendant is the operator's interface with the machine



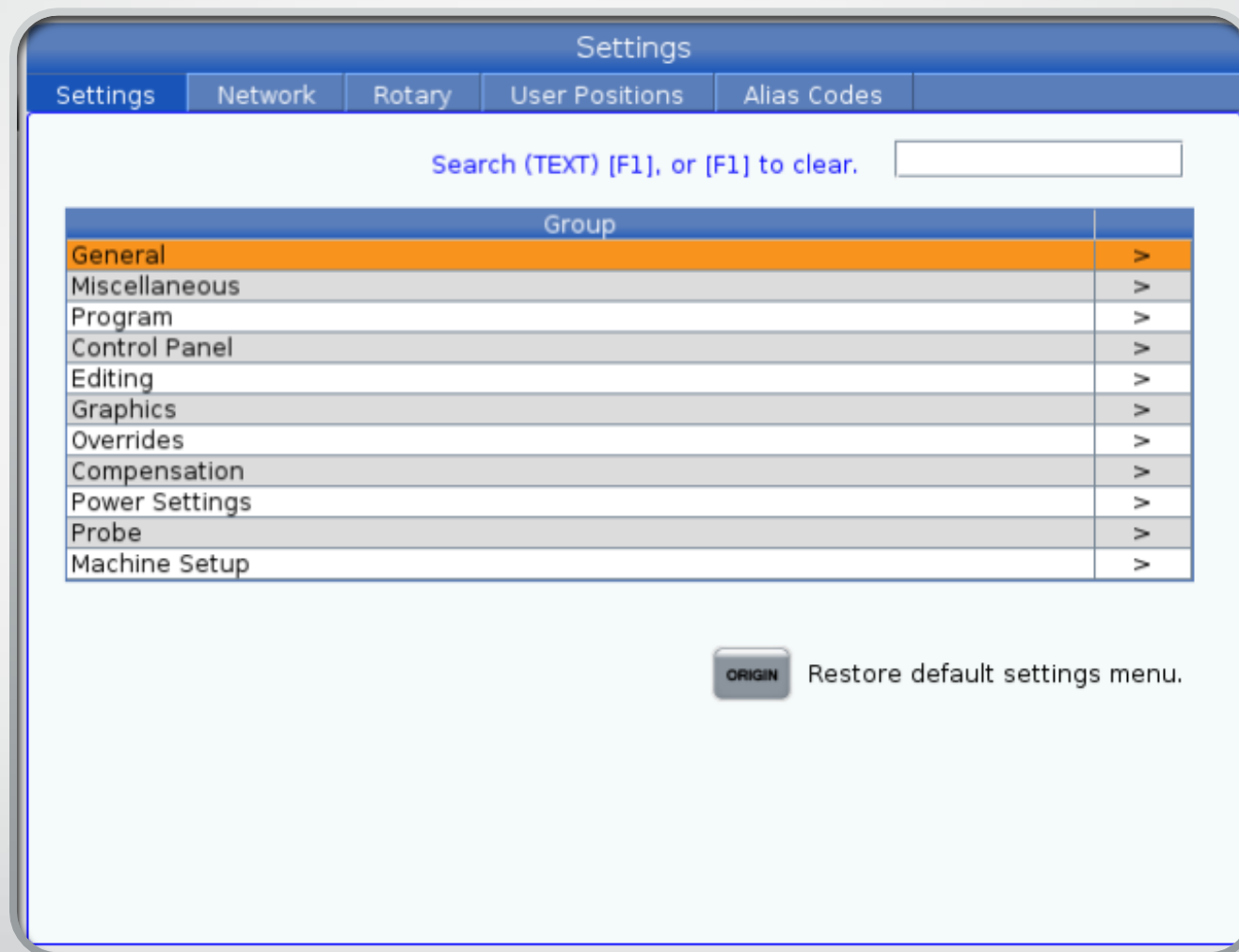
Settings on a Haas Control

- Settings are default values that can change the way the machine preforms.
- 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.
 - 119 – Offset lock – This can turn off the ability to make offset adjustments.
 - 132- Jog or home before tool change- If the turret is not at the home position the control will display a message asking if tool change is ok at current position.

Settings

SettingsNetworkRotaryUser PositionsAlias Codes

◀

Group Listings

Search (TEXT) [F1], or [F1] to clear.

	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	

ORIGIN

Restore default settings menu.

9 - Dimensioning

This setting selects between inch and metric mode. When it is set to INCH , the programmed units for X and Z are inches, to 0.0001" . When it is set to MM , programmed units are millimeters, to 0.001 mm. ...

HELP

[View full text.](#)

Program Settings

- Common Program settings that are changed:
 - 39 – Beep @M00, M01, M02, M30 – This controls the beeping sound at any of these codes.

Settings

SettingsNetworkRotaryUser PositionsAlias Codes

◀

Group Listings

Search (TEXT) [F1], or [F1] to clear.

	Group	Name		Value	Unit
29	Program	G91 Non-Modal	>	Off	
31	Program	Reset Program Pointer	>	On	
32	Program	Coolant Override	>	Normal	
34	Program	4th Axis Diameter		1.000	IN
35	Program	G60 Offset		0.300	IN
36	Program	Program Restart	>	Off	
39	Program	Beep @M00,M01,M02,M30	>	Off	
42	Program	M00 After Tool Change	>	Off	
43	Program	Cutter Comp Type	>	A	
44	Program	Min F In Radius CC %		50	%
45	Program	Mirror Image X Axis	>	Off	
46	Program	Mirror Image Y Axis	>	Off	

ORIGIN

Restore default settings menu.

Unavailable

HELP

View full text.

Program Settings

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

Settings

SettingsNetworkRotaryUser PositionsAlias Codes

◀

Group Listings

Search (TEXT) [F1], or [F1] to clear.

	Group	Name		Value	Unit
43	Program	Cutter Comp Type	>	A	
44	Program	Min F In Radius CC %		50	%
45	Program	Mirror Image X Axis	>	Off	
46	Program	Mirror Image Y Axis	>	Off	
47	Program	Mirror Image Z Axis	>	Off	
48	Program	Mirror Image A Axis	>	Off	
52	Program	G83 Retract Above R		0.000	IN
56	Program	M30 Restore Default G	>	Off	
57	Program	Exact Stop Canned X-Y	>	Off	
58	Program	Cutter Compensation	>	Fanuc	
63	Program	Tool Probe Width		0.000	IN
64	Program	T Ofs Meas Uses Work	>	Off	

ORIGIN

Restore default settings menu.

43 - Cutter Comp Type

This controls how the first stroke of a compensated cut begins and the way the tool is cleared from the part. The selections can be A or B ; refer to the Tool Nose ...

HELP

[View full text.](#)

Miscellaneous Settings

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

Settings

SettingsNetworkRotaryUser PositionsAlias Codes

Group Listings

Search (TEXT) [F1], or [F1] to clear. Miscellaneous

	Group	Name		Value	Unit
90	Miscellaneous	Max Tool Offsets To Display		200	
114	Miscellaneous	Conveyor Cycle		0	Min
115	Miscellaneous	Conveyor On-Time		2	Min
117	Miscellaneous	G143 Global Offset		0.000	IN
118	Miscellaneous	M99 Bumps M30 Cntrs	>	On	
131	Miscellaneous	Auto Door	>	Off	
142	Miscellaneous	Offset Chng Tolerance		0.250	IN
143	Miscellaneous	Machine Data Collect		0	
155	Miscellaneous	Load Pocket Tables	>	Off	
156	Miscellaneous	Save Offset With Prog	>	Off	
242	Miscellaneous	Air Water Purge Interval		5	Min
243	Miscellaneous	Air Water Purge On-Time		0.500	Sec

Range: 6 to 200

ORIGIN Restore default settings menu.

90 - Max Tools To Display

This setting limits the number of tools displayed on the Tool Offsets screen. The range of this setting is 6 to 99.

HELP View full text.

Control Panel

- 163 – Disable .1 jog rate – this can disable the .1 jog feed command.

Settings

SettingsNetworkRotaryUser PositionsAlias Codes

◀

Group Listings

Search (TEXT) [F1], or [F1] to clear.

Control Panel

	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	%

ORIGIN

Restore default settings menu.

6 - Front Panel Lock

When set to ON , this Setting disables the Spindle [FWD] / [REV] keys and [TURRET FWD] / [TURRET REV] keys.

HELP

[View full text.](#)

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.

Settings

SettingsNetworkRotaryUser PositionsAlias Codes

◀

Group Listings

Search (TEXT) [F1], or [F1] to clear.

Power Settings

	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

Range: 0 to 7200

ORIGIN

Restore default settings menu.

1 - Auto Power Off Timer

This setting is used to automatically power-down the machine after a period of idle time. The value entered in this setting is the number of minutes the machine remains idle until it is powered down. The ...

HELP

[View full text.](#)

Loading Tool Holders

- Toolholders are used to fasten tools to the turret. There are multiple types of tool holders for different turrets, but most clamp onto the tool externally with fasteners.





Loading Tools

Loading tools is either done directly in the turret or in a toolholder that is installed on the turret. The type of turret will determine how this is done.

This is a VDI holder, which is installed in the turret with a large fastener and clamps onto a tool.

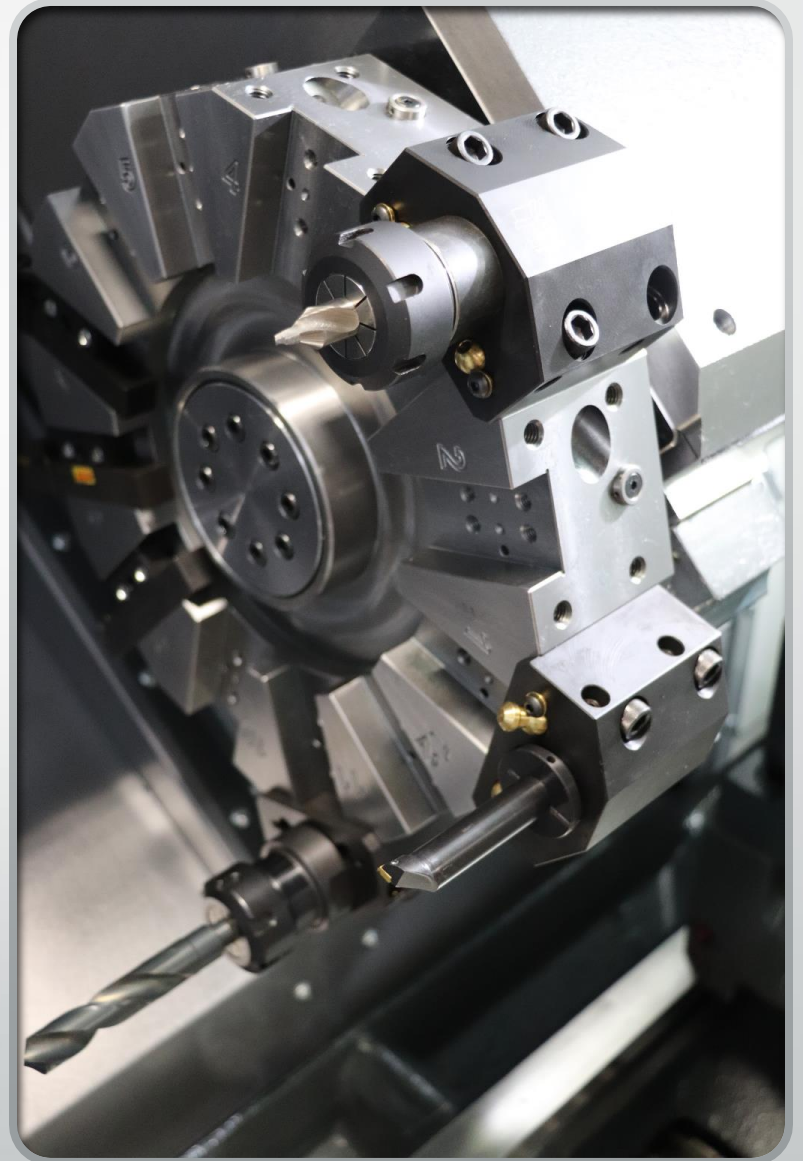
A close-up photograph of a Bolt On Turret (BOT) tool holder assembly on a lathe. The tool holder is a dark, heavy-duty metal component with multiple tool slots. A yellow label with the text "SANDVIK" is visible on one of the tool slots. The tool holder is bolted directly to the turret. The background shows the complex mechanical structure of the lathe, including the turret and various adjustment mechanisms. A red diagonal line is overlaid on the image, separating the title and text from the main image.

Loading Tools

- This is a Bolt On Turret (BOT). This is an example of the tools being bolted directly to the turret. ID toolholders are bolted to the outside of the turret to align the tools on centerline.

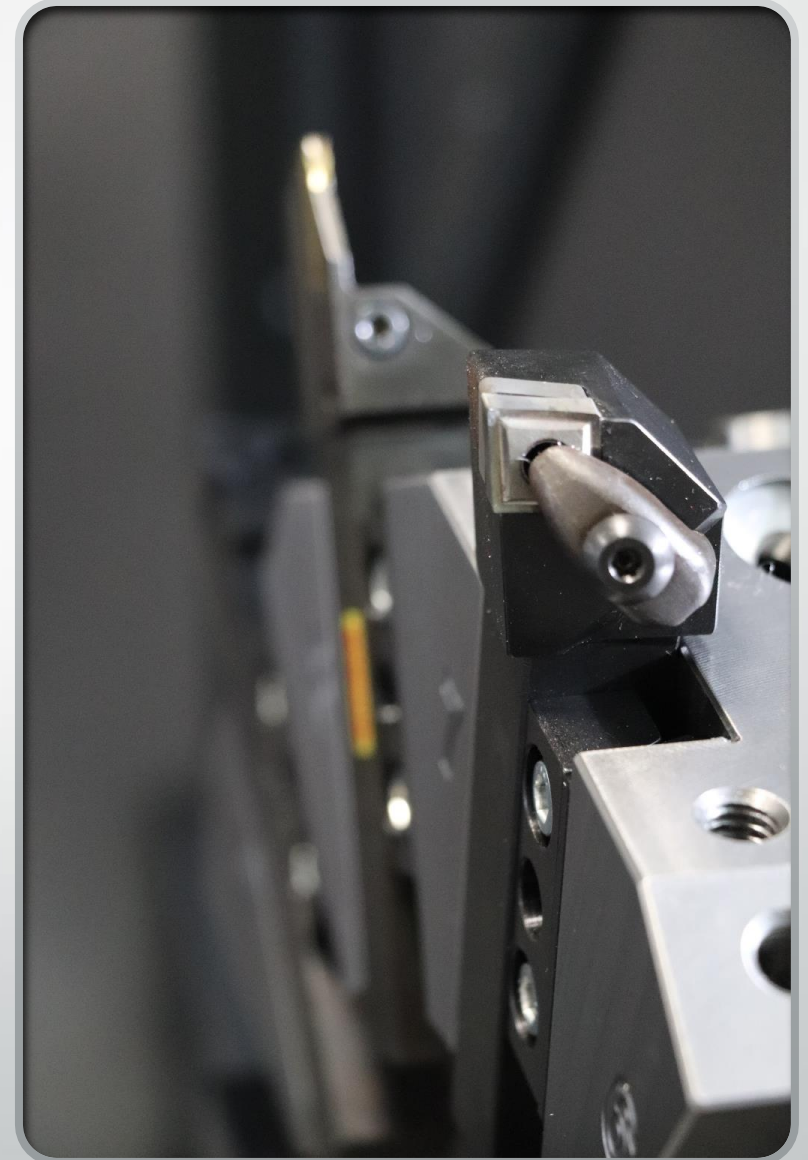
Installing Tools in a Turret

- Tools must be mounted on the turret before they can be used.
- OD and ID tools have specific holding methods:
 - OD tools are typically held in with a wedge style clamp.
 - ID tools use a holder with a bore and special lathe collets to adjust for the diameter of the tool.



OD Tool install

- OD turning, grooving, threading, and parting tool are all held on with a wedge system.
- This simple wedge system uses a clamping force to hold the tool in place.
 - When tools need to be removed, the center screw can be used to push the wedge apart.
- Tool should be kept to the centerline of the turret.



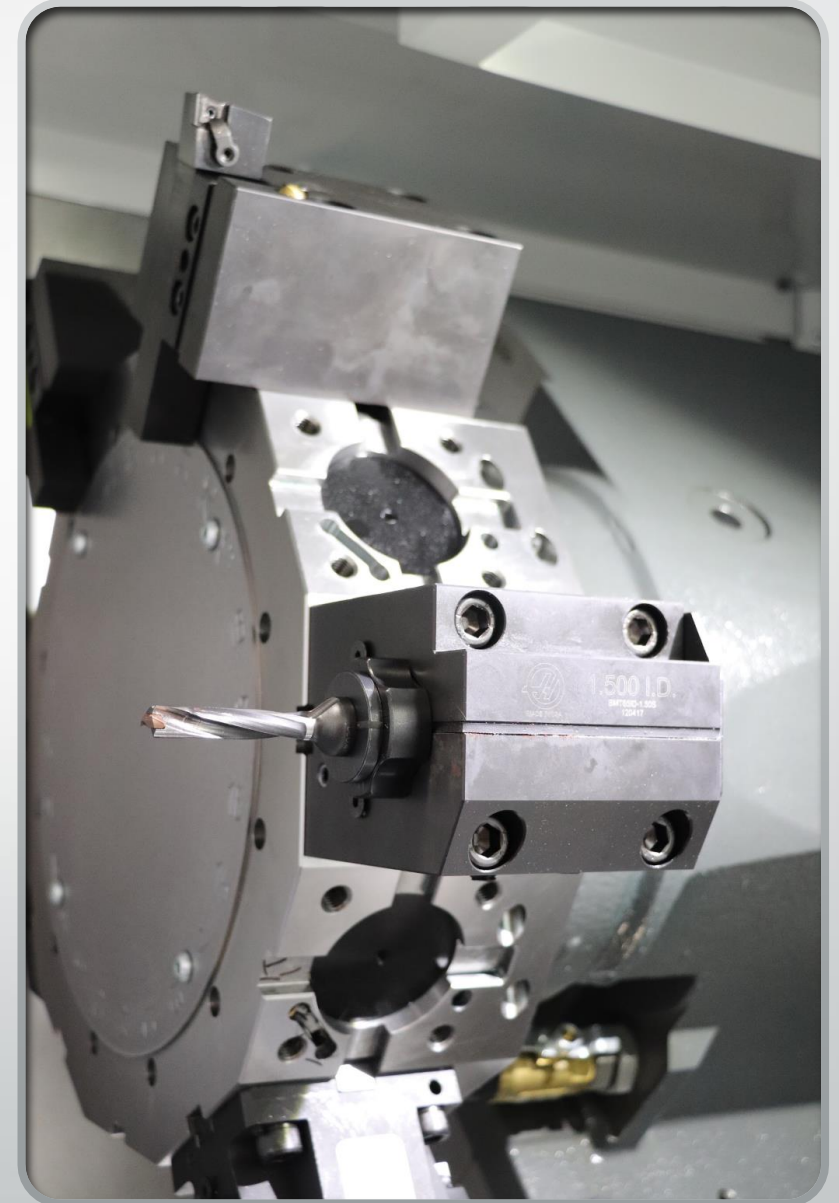
ID Tool holders

- ID tool holders bolt onto the turret.
- The holders have a round hole and come in nominal sizes: 1", 1.25" and 1.5".
- Tools that are smaller than these diameters require the use of a sleeve to reduce the bore in the holder.
- Space ID/axial tools by at least every other tool position to avoid adjacent tools from colliding with the machine.



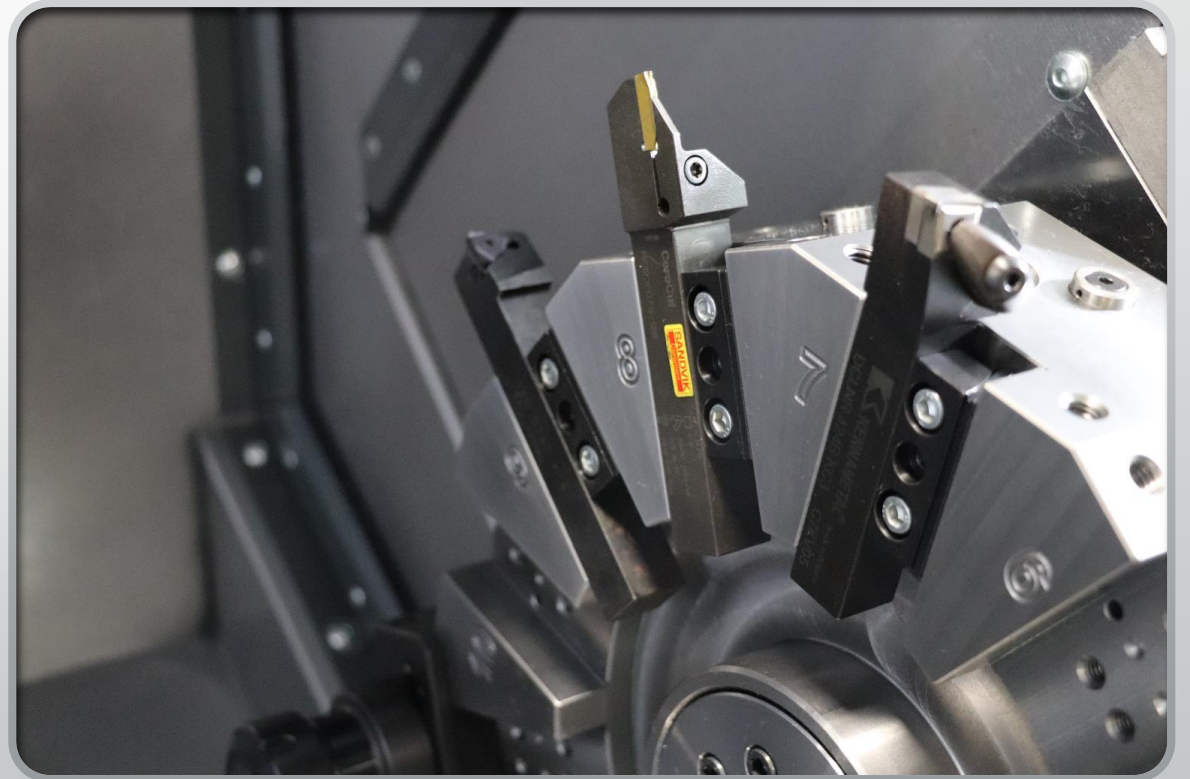
Cutting Tools

- The cutting tools on a lathe can be subdivided into two categories:
 - Outer Diameter (OD)
 - These tools work on the OD of the part. Examples of OD tools are:
 - Turning tools
 - Grooving tools
 - Threading tools
 - Parting tools
 - Inner Diameter (ID)
 - These tools work on the ID of the part. Examples of ID tools are:
 - Drills
 - Taps
 - Reamers
 - Boring bars
 - Grooving bars
 - Threading bars



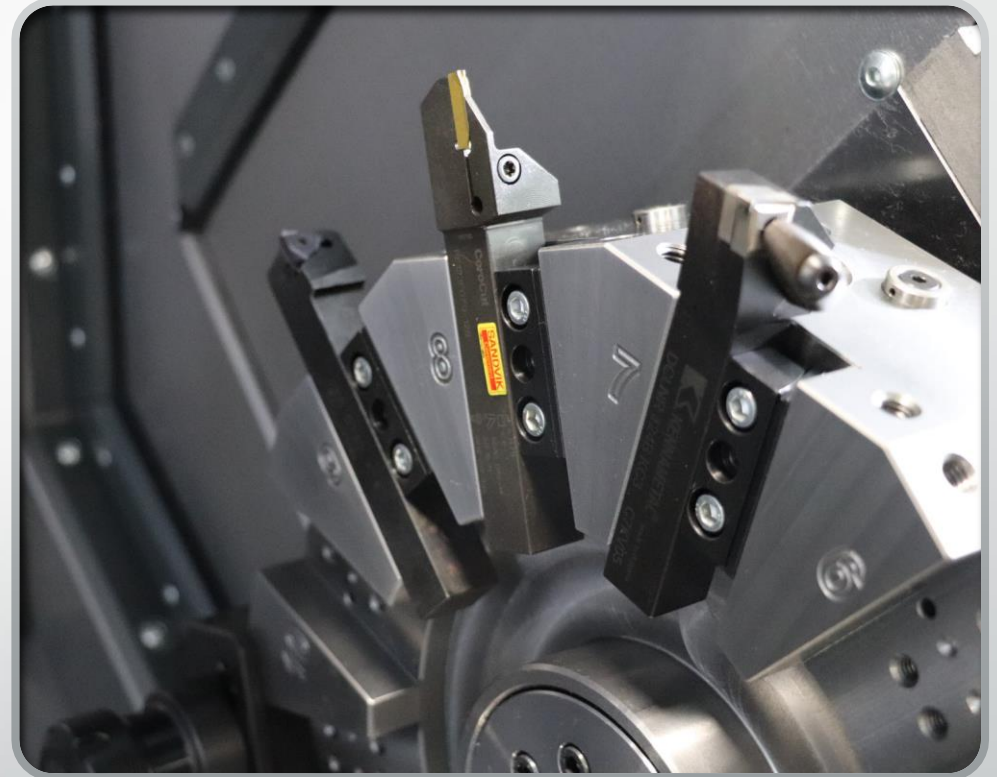
Cutting Tools

- The majority of cutting tools utilize a steel body, which is held in the turret, and a carbide cutting insert.
- This makes the overall cost of the tool cheaper to produce while adding flexibility to the tool.
 - Inserts can be replaced when worn .
 - Different insert grades can be used when material changes.



Turning Tools

- Turning tools are used to remove stock from the face and OD of a part.
- These tools have a steel body and a carbide insert, this reduces the cost of tool and holder while increasing the flexibility of the tool.
 - The tool is held in the turret by the steel body.
 - The insert can be replaced when the cutting edge is worn out.
 - The insert can also be replaced to best suit the material being turned.



Turning Tools

- These tools come in a variety of insert shapes and configurations.
 - The shapes used in this series are 80 degree and 55 degree diamond.
- Inserts are held on with a screw or clamp.
 - These are an allen or a torx style screw.
- There can also be a seat between the insert and holder.
 - This helps reduce vibration from cutting.
 - The seat can also be replaced in the event of a crash.



Boring Bars

- Boring bars are a tool that turns material on the ID of the part.
 - Boring bars can not create a hole in a part, they are used after a hole is made.
- Boring bars are used to bring the ID of a part to a specific diameter.
- Profiles can be programmed for the boring bar to cut.



Boring Bars

- A boring bar is a steel bar with a replaceable carbide insert.
- The bar should always be as large of a diameter as possible. Hole diameter is the limiting factor when choosing a boring bar.
- Boring bars come in a variety of insert shape and options, much like OD turning tools.
 - Much like OD turning tools, there are endless insert shapes and configurations that can be used. The operation and material being cut determine the insert that should be used.



Grooving Tools

- These tools make grooves or a notch in the part.
- Grooves can be cut on the OD, ID or face of the part.
 - OD grooving tools are held like OD turning tools.
 - ID grooving tools are held in a bar, like a boring bar.
 - Face grooving tools look like a OD turning holder but have a curved insert support. Face grooving requires a special tool holder. The holder is secured with a wedge clamp but needs to be positioned on the turret like a boring bar.
- Grooves can be produced with many different shapes.



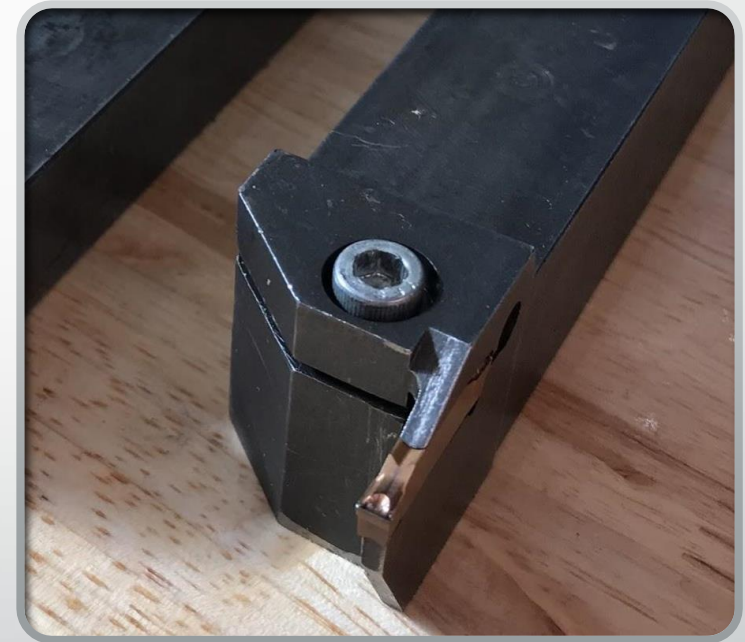
Grooving Tools

- Grooves can be produced with many different shapes.
 - Grooving tools can be purchased to a specific groove size, programmed to take multiple cuts to make a wide groove, or a specialty insert can be purchased to make the groove.
- Grooving tools can be used to cut profiles on a part that regular turning tools or boring bars can not reach.



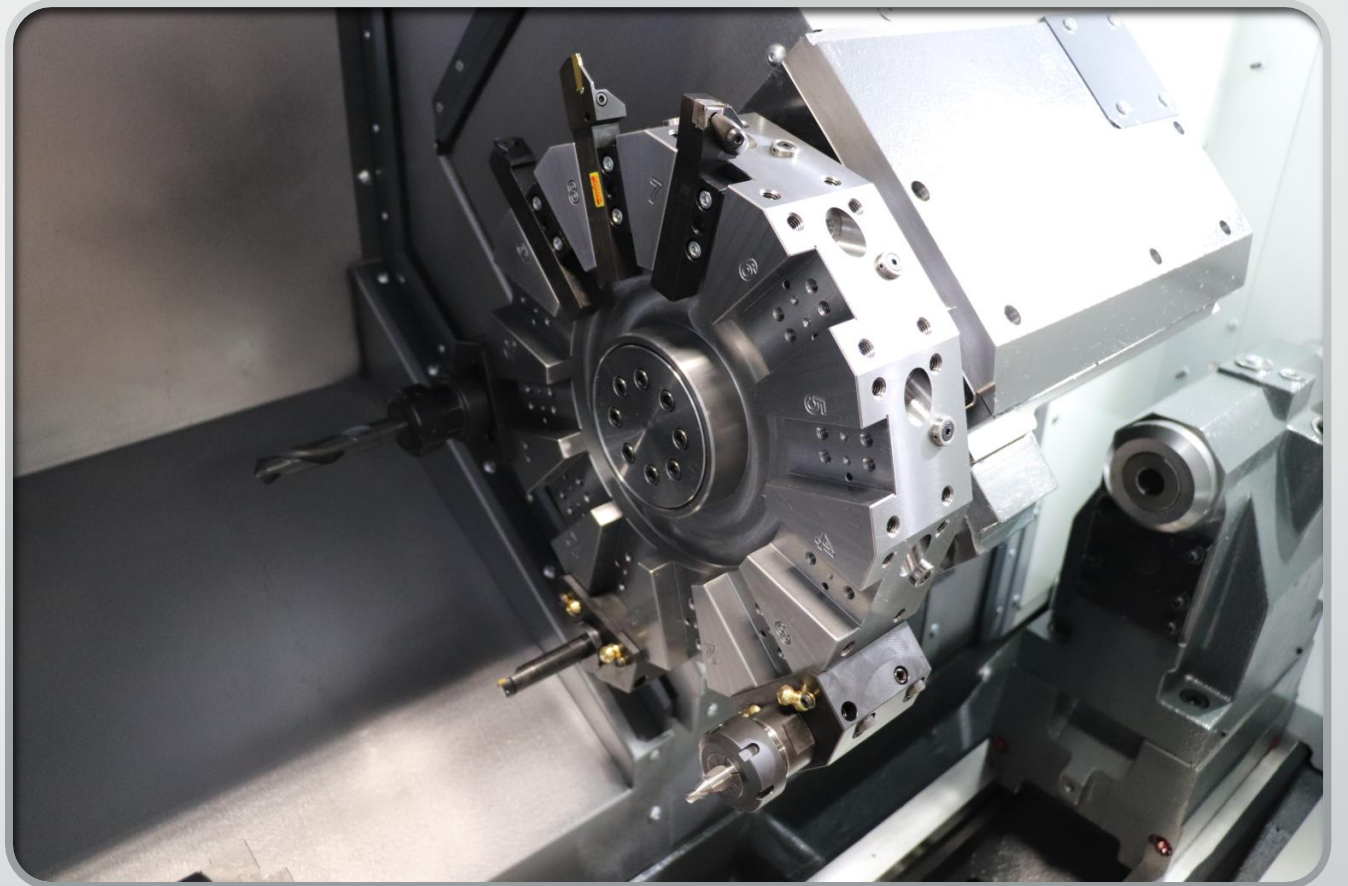
Grooving Tools

- Much like turning tools, the grooving tools consist of a steel body and a carbide insert.
- These inserts are held in with a top clamp.
- Inserts with V's on the top and bottom are used to profile a part.
 - The V's hold the insert in place when side load is applied to the part.



Machine Zero Position

- The machine zero position (MZP) is the machine's home position. This is where the turret homes to upon start up.



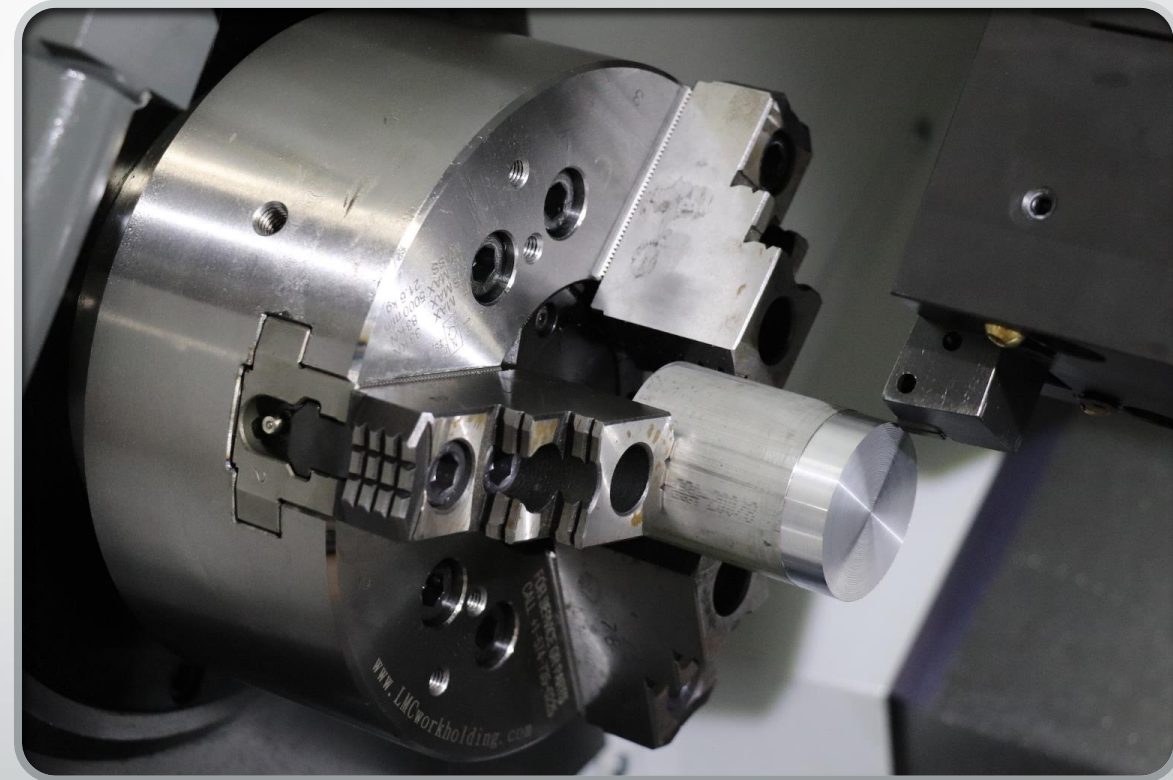
Tool Offsets

- Tool offsets are the distance from the tool at machine zero position to the part PRZ. It is common practice to use just the tool offset without any work offsets.
- These offsets are stored in the tool offset registry page of the machine



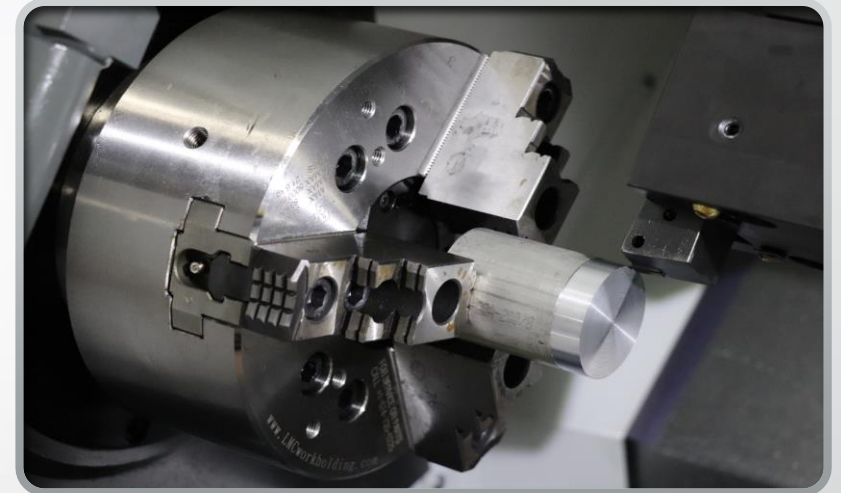
Offsets for OD tool offsets

- After the turning tool is installed the X and Z offsets can be set.
- In order for the offsets to be set manually, the part must first be faced and turned. Doing this ensures there is no runout from the part when the tools are touched off.
 - If there was runout and the tool was touched off the high or low of the part, the tools offset would then be incorrect. If the offset is not correct the part will not be cut to the programmed dimension.



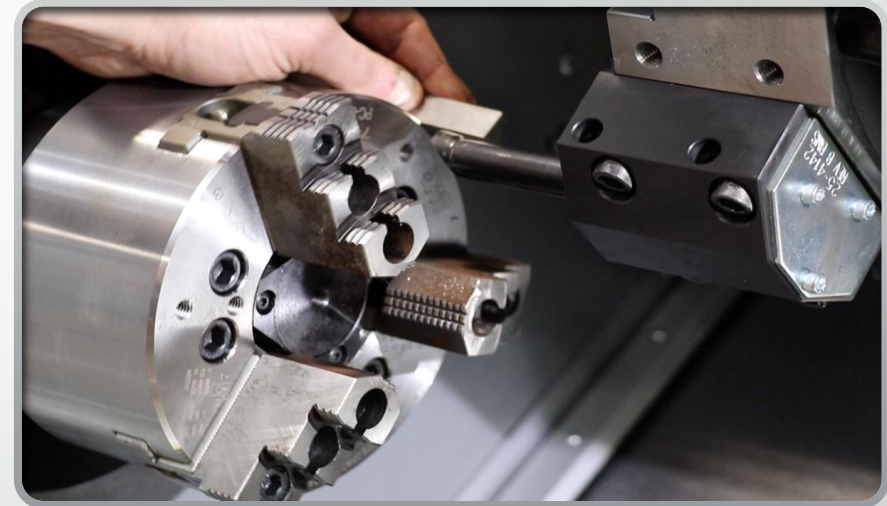
Offsets for OD turning tools

- After the surface has been faced and turned, the OD tools can be brought close to the part surface.
- Shim stock can be used as a gage, the tool should approach the part while sliding the shim, checking for pressure on the shim.
- Once there is drag on the shim, the offset is then set.
 - Z Face Measure is used on the control to store the Z offset in the control.
 - X Diameter Measure is used on the control to store the X offset. Once the offset is taken, the control will ask for the diameter. This is the diameter the tool was touched off from. Entering this diameter will adjust the X tool offset to the centerline of the spindle.



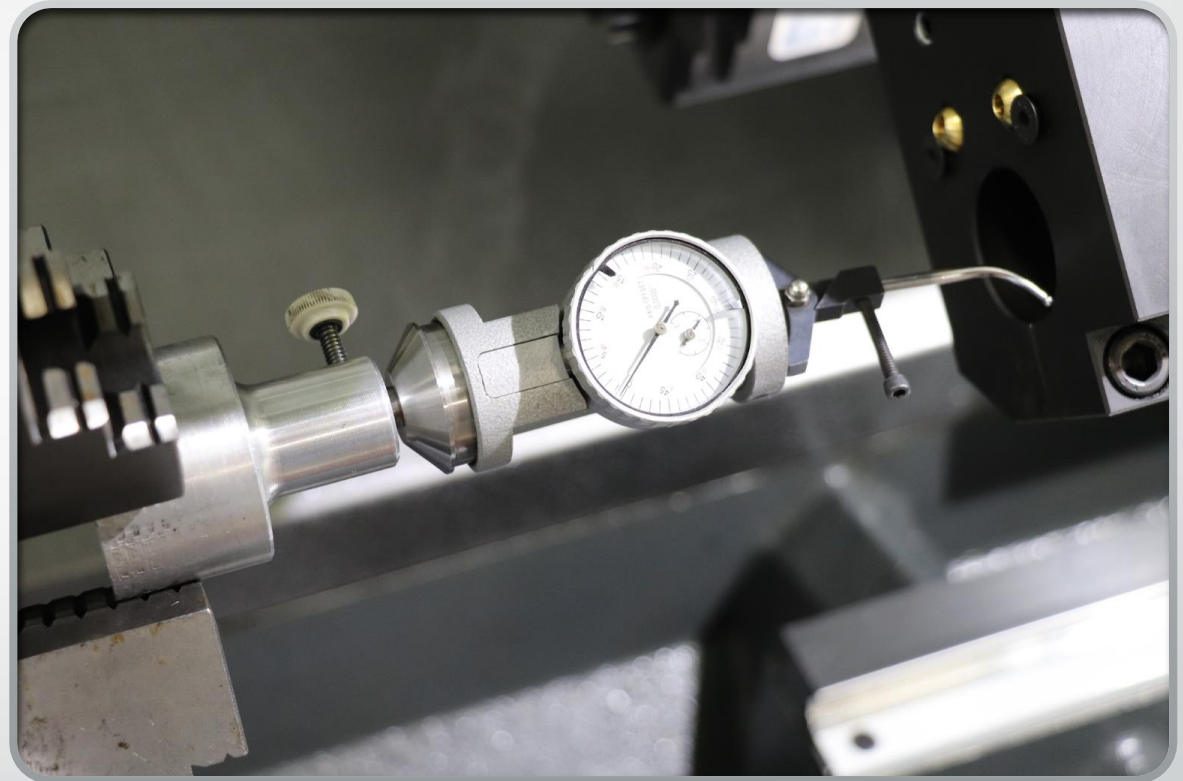
Offsets for ID turning tools

- ID tools are touched off the face of the part, the same way the OD tools are set.
- Finding the X offset for ID tools differs from that of OD tools. The insert on ID tools often faces up, not allowing for the insert to be set by touching the turned OD surface.
- Common practice is to use a gage block held on an OD surface of known diameter. The gage block on the diameter is rotated while the ID tool is moved in the X positive direction. Once contact is felt, the X diameter measure button is used to set the tool offset.
 - The OD of the chuck could be used for the ID tool touch off. A part with a turned OD would also work for setting ID tools.
- Remember that after using the X diameter measure button, the diameter must be entered to shift the tool offset to the spindle center line.



ID holders

- Tools like drills, reamers, and taps need to be in line with the spindle centerline.
- Before installing the tool, the bore of the tool holder must be indicated. Finding this X value will allow the tool to work on the same centerline as the spindle.
 - Once the centerline is found, X diameter measure can be used and the diameter entered will be o.o.
- A coaxial indicator can be used to find the centerline of a toolholder.



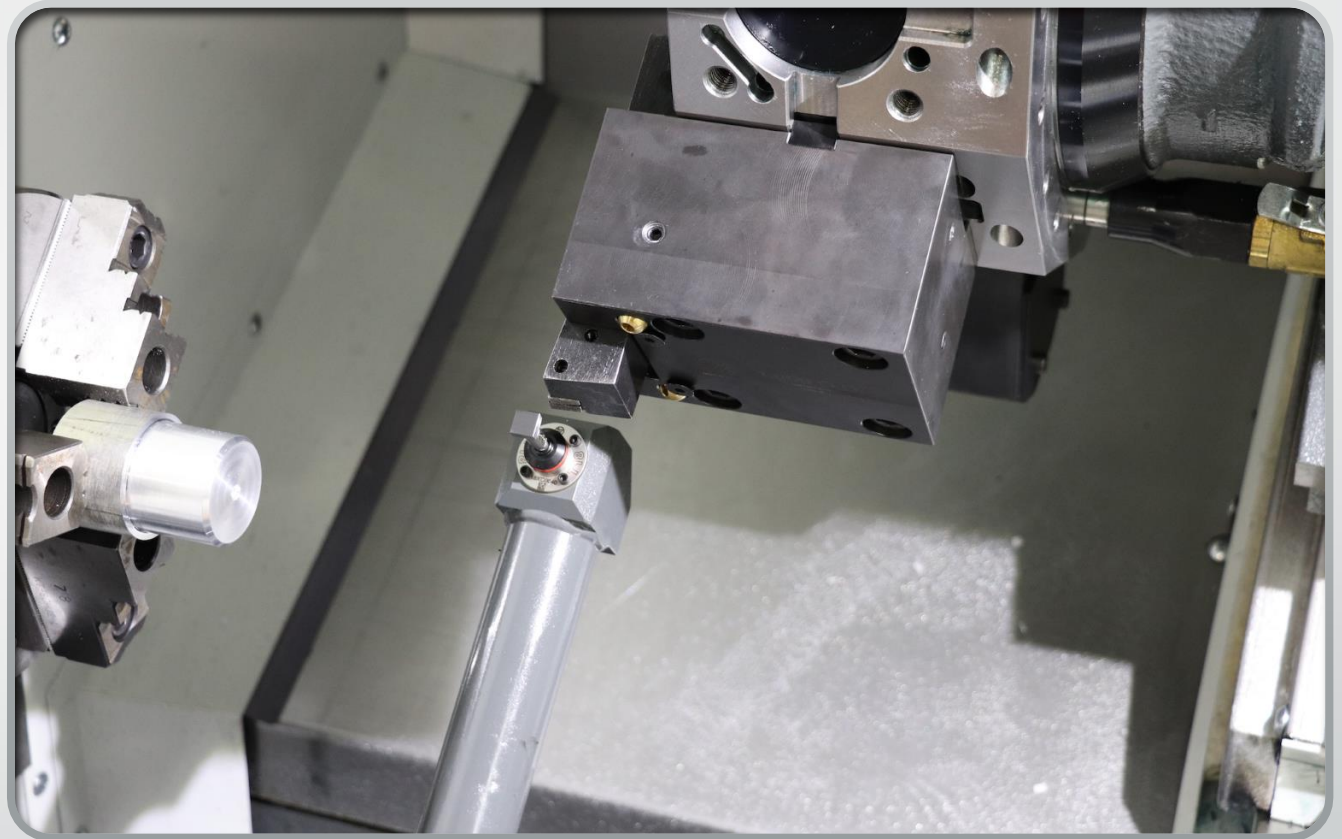
Tool Using the Tool Presetter

A tool setter can automatically set the tool offsets. By using VPS on the control to answer a series of questions, a program will be generated to run the probing cycle. This will set the tools X and Z offsets.



Tool Offsets Using VPS

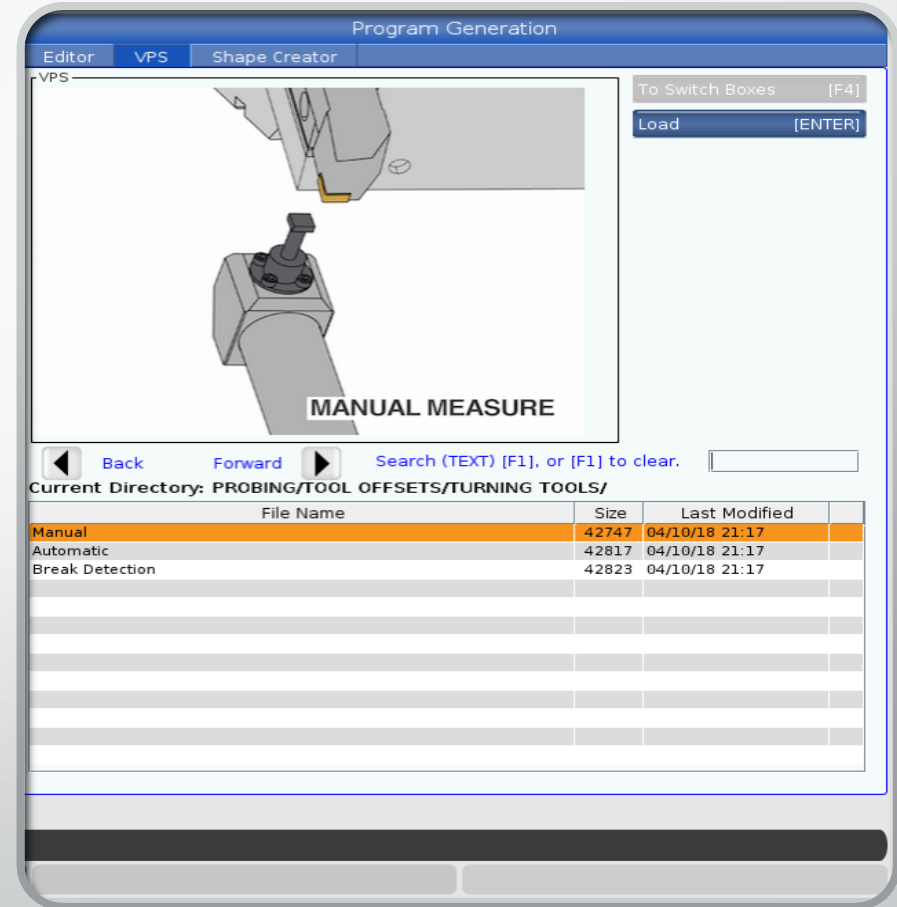
- The tool presetter uses a tool probe to measure the X and Z offsets. Once the tool touches the stylus, the positions are recorded and stored in the tool offset page.
- The probe arm is stored by the chuck. It extends out in front of the chuck and retracts back when not in use.



Using a Tool Presetter

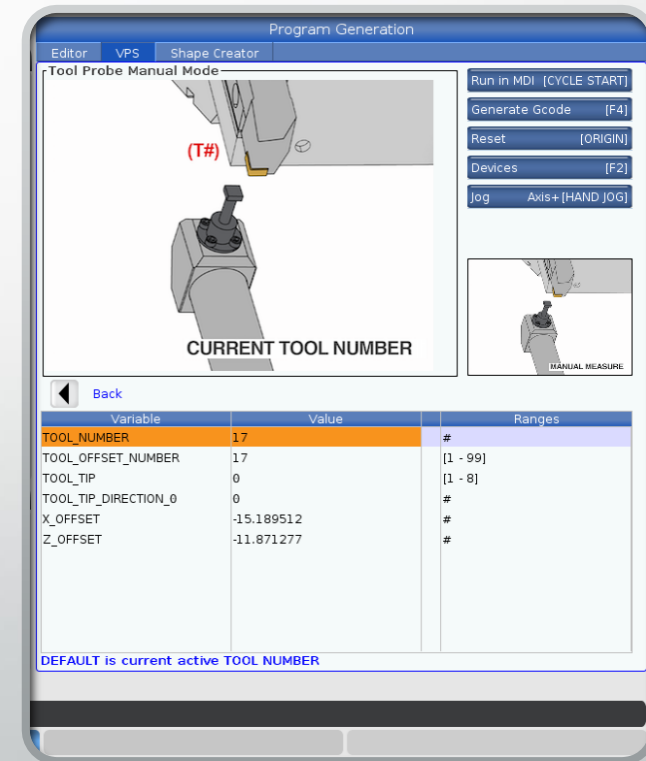
The presetter is programed through VPS.

- To program the tool setter:
 - Select "EDIT" from the modes
 - Cursor over to VPS, then cursor down to enter
 - Select "Probing", then cursor over to enter
 - Cursor over to enter tool offsets
 - Select "Turning Tools", then cursor over
 - Select "Manual Offsets", then cursor over



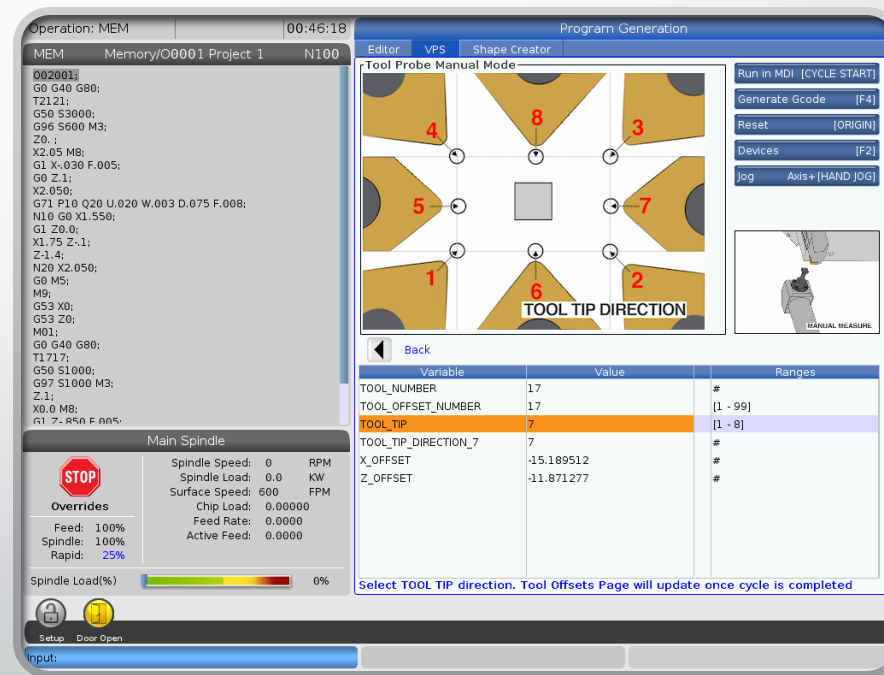
Tool Setter Inputs

- Once in manual offset, there are variables to input:
 - Tool Number – Tool number to be probed
 - Tool Offset Number – Tool offset to store the value
 - Tool Tip – Orientation of the tools insert
 - Tool Tip Direction – Corresponds with the tip direction
 - X Offset – Current X offset
 - Z Offset – Current Z offset



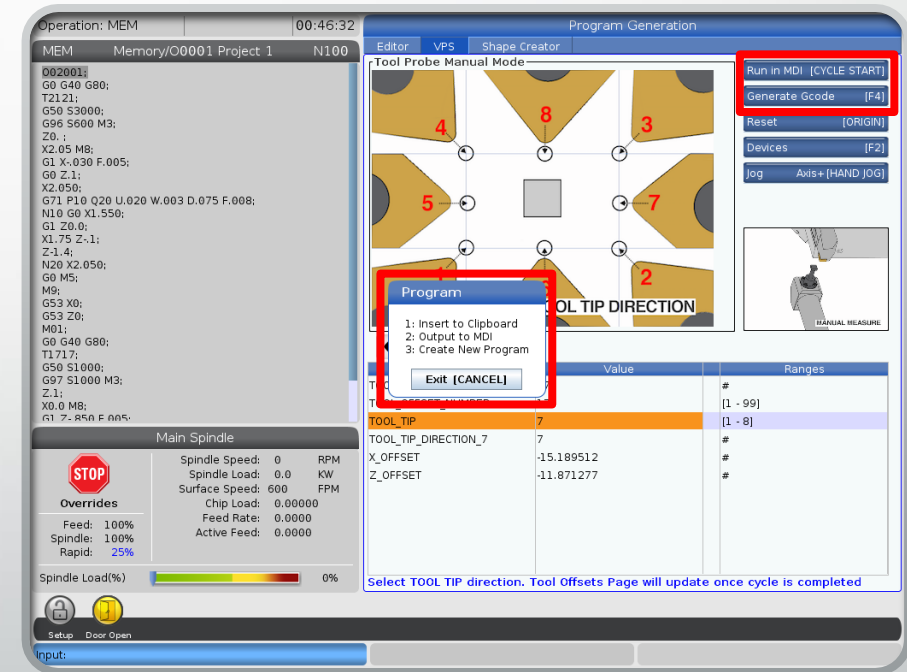
Tool Tip Direction

- The tool tip direction must be selected. This direction is selected based off the orientation of the tools insert.
- The direction determines the travel of the tool to touch the probe. This tool tip direction is also used for Tool Nose Radius Compensation.
 - Without this direction, the control does not know how to apply to the proper tip comp or tool touch off.



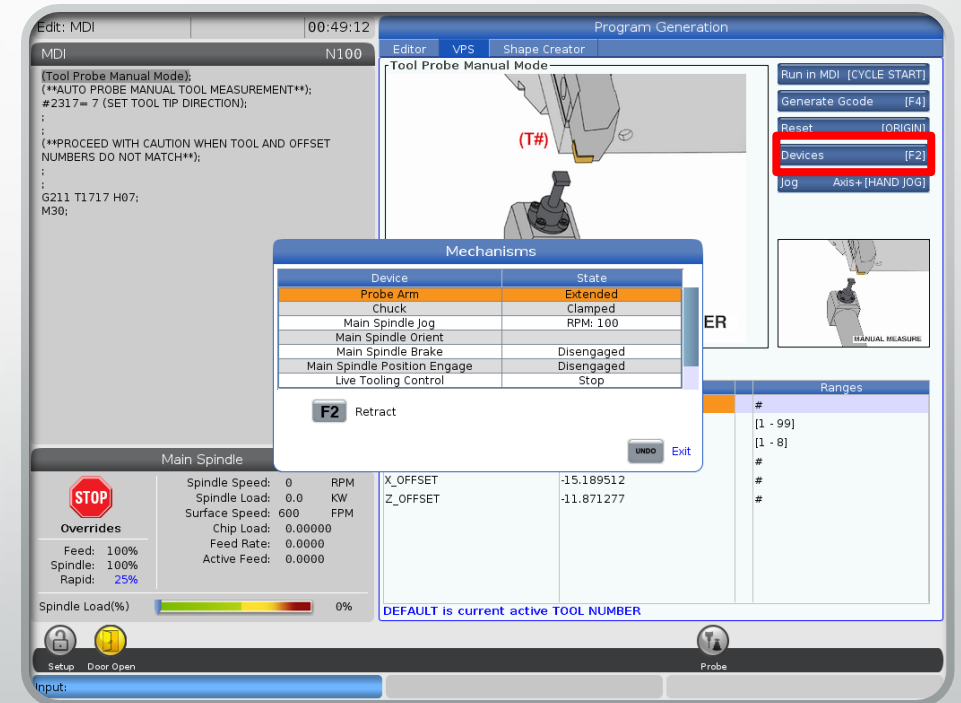
Creating the Program

- Once the variables are input, the program can be output. In this case F4 is used to generate the code, then 2 is selected to output the file to MDI.
- Other options include:
 - Inserting to the clipboard, this can then be inserted into a turning program.
 - Creating a new program, stored in memory, for the probing cycle.



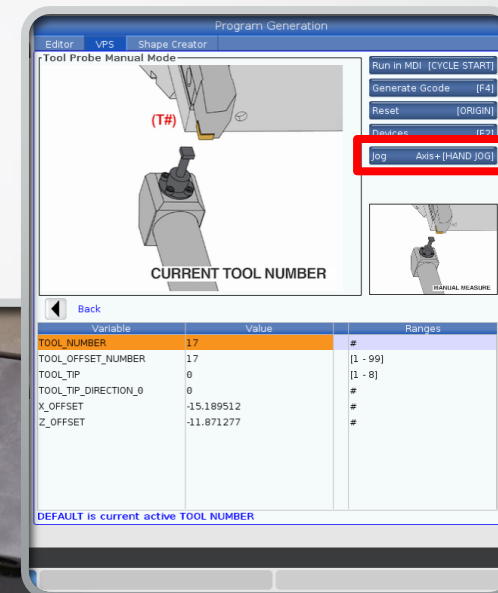
Lowering the Arm

- In the manual tool offset page, F2 can be selected to see the device menu. Once in the menu, select "Probe Arm", then F2 can be used to lower the arm. Once lowered, select the UNDO button to exit the device screen.



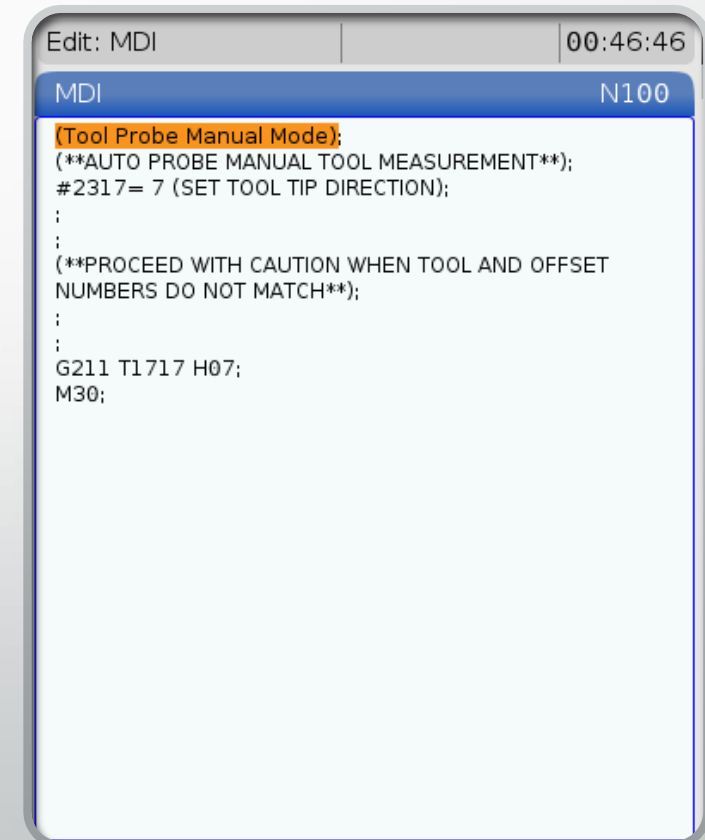
Jogging the Tool

- After the arm is down, the tool can be jogged into position. The tool should be brought down where it clears the stylus in both the X and Z axis.



Running the Cycle

- With the “MDI” mode active, select cycle start to run the probe cycle. The routine will run and record the results. The tool offsets will automatically be updated for the tool.

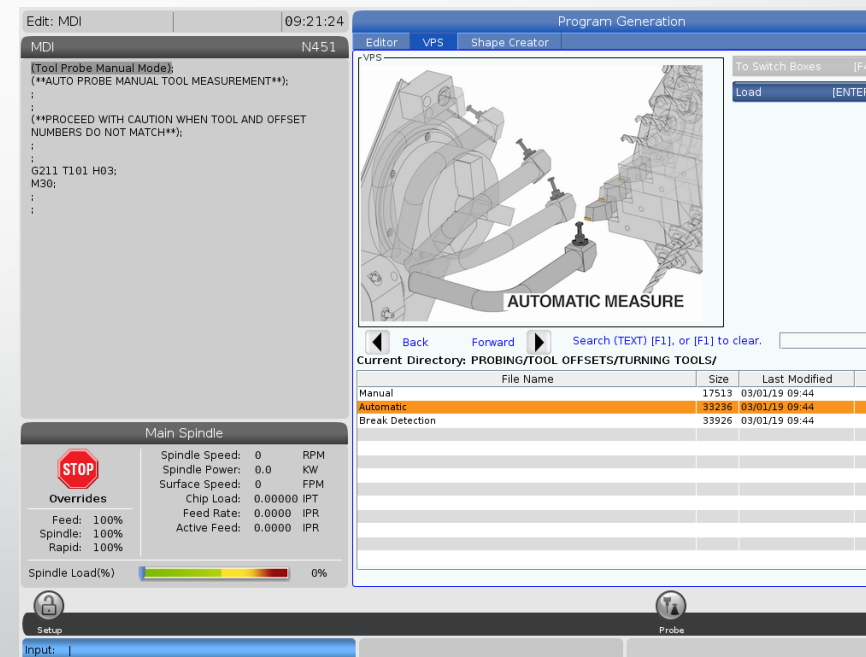


The screenshot shows a CNC control interface with a top status bar displaying 'Edit: MDI' and a timer '00:46:46'. Below this is a blue header bar with 'MDI' on the left and 'N100' on the right. The main area contains a program listing for a manual tool probe cycle. The first line is highlighted in orange. The program includes comments in double asterisks, a tool tip direction setting, a cautionary note, and a G211 command for tool measurement.

```
Edit: MDI 00:46:46
MDI N100
(Tool Probe Manual Mode);
(**AUTO PROBE MANUAL TOOL MEASUREMENT**);
#2317= 7 (SET TOOL TIP DIRECTION);
:
:
(**PROCEED WITH CAUTION WHEN TOOL AND OFFSET
NUMBERS DO NOT MATCH**);
:
:
G211 T1717 H07;
M30;
```

Automatic Tool Probe

- After a tool has been manually probed, the automatic tool probe option can be used. The variable screen will be the same as the manual screen.
- The difference is that in automatic the probe will come down on its own and the turret will move to position without the operator manually positioning it.
 - This is often used after an insert has been changed.



Broken Tool Detection

- Haas lathes can detect broken or worn tools using a probe arm.
- A tool just needs to be touched off the arm, and the machine will detect how much variation there is from the original tool offset.
- The operator can set how much variation to allow before the machine alarms out.

Tool Breakage Detection

- In the tool breakage screen the only variable that is new is “wear tolerance”. This is the maximum amount of deviation from the original tool offset.
- When the tool is probed, an alarm will be given if the measurement exceeds this tolerance.

The screenshot displays a CNC control interface with the following components:

- MDI Window:** Shows the text "(Tool Probe Manual Mode); (**AUTO PROBE MANUAL TOOL MEASUREMENT**); ; (**PROCEED WITH CAUTION WHEN TOOL AND OFFSET NUMBERS DO NOT MATCH**); ; G211 T101 H03; M30; ; ;".
- Main Spindle Window:** Displays a "STOP" button and "Overrides" for Feed (100%), Spindle (100%), and Rapid (100%). It also shows spindle parameters: Spindle Speed (0 RPM), Spindle Power (0.0 KW), Surface Speed (0 FPM), Chip Load (0.00000 IPT), Feed Rate (0.0000 IPR), and Active Feed (0.0000 IPR). A "Spindle Load(%)" bar is at 0%.
- Tool Probe Break Detection Mode Window:** Features a diagram of a tool and workpiece with a red circle highlighting the "TOOL WEAR TOLERANCE" area. Below the diagram is a table of variables.

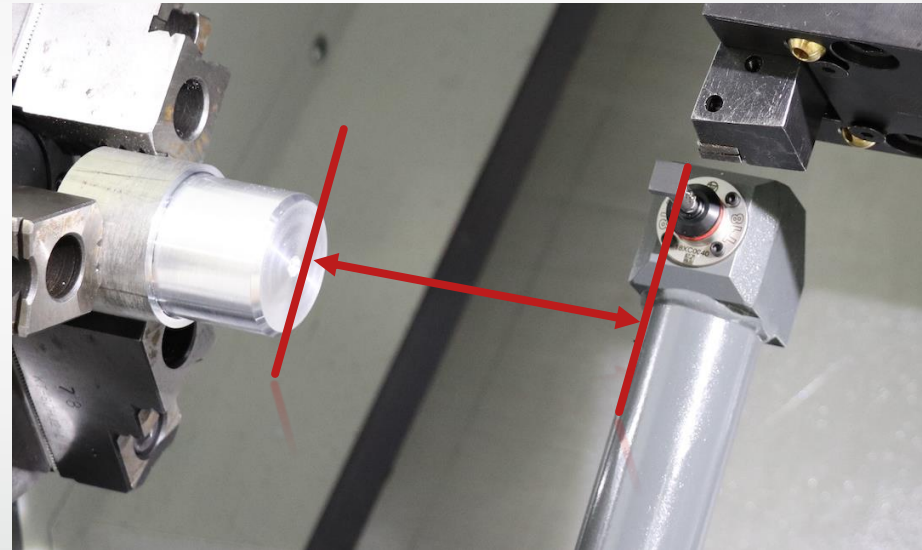
Variable	Value	Ranges
TOOL_NUMBER	1	[1 - 24]
TOOL_OFFSET_NUMBER	1	[1 - 99]
CALIBRATION_DIRECTION_3	3	
TOOL_TYPE_NONE	NONE	
WEAR_TOLERANCE	0.005	[0.0 - 0.1]
X_OFFSET	-17.9363	
Z_OFFSET	-9.3449	

Below the table, it says "Enter a wear tolerance to determine if the tool is broken".

Work Offsets

Using VPS

- After the tool offsets are probed, the Z axis work offset will need to be established.
- This is the distance from the probe to the Z zero of the part.
- This is found by touching one tool off manually at the parts Z zero position. After the position is found the Z Face Measure button can be used to record the Z work offset.
 - The correct Z work offset must be active for the offset to properly record.



Work Offsets

- Work Offset values are then stored in the work offset page.
- G54 thru G59 are the standard work offset numbers that are used.
- The control has 99 additional offset, P1-P99. This makes for a total of 105 work offsets on the control.

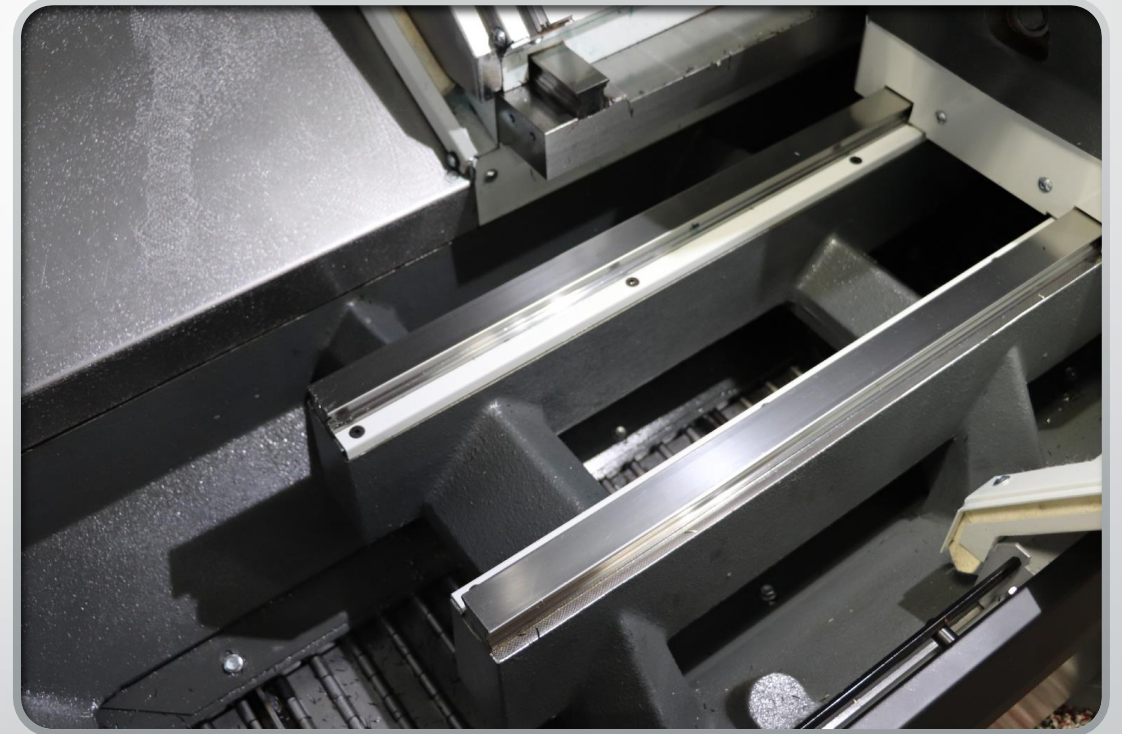
Tailstock

- When a tailstock is needed it can be positioned on the lathe bed close to the part, then locked in place.
- The tailstock has a hydraulic quill that can advance and retract to engage the part.
 - The tailstock can be activated with buttons on the control or with M-codes within a program.
- The quill operates from the same hydraulics as the chuck, quill pressure can be adjusted the same way chuck pressure is adjusted.



Chip Conveyor

- Chip conveyors are used to remove the chips made from the machining operation. The conveyor sits above the coolant tank. When the chips fall, they are carried off in the conveyor and out of the machine. As the chips are moving, the coolant can drain down into the coolant tank.
- Chips are transferred into a bin that can be removed when it fills.
- Conveyors allow the machine to run without the build up of chips.
 - M31 will start the chip conveyor.
 - M33 will stop the chip conveyor.



Parts Catcher

- Part catchers are used when the parts are cut off from the bar stock. To prevent the part from bouncing around inside the machine, a shoot can be programmed to raise up as the part is cut off. The part then falls into the shoot and into a holding bin on the door of the machine.
- The parts catcher can be raised and lower with M codes (M36/M37). These codes can be used within a program.



Spindle Liners

- Spindle liners are used when stock is placed in the spindle tube. The liner prevents the unsupported part of the bar from vibrating in the spindle tube while spinning.
- Spindle liners should be sized just larger than the bar stock, which should slide through the liner freely.

