



Welcome to Trident Machine Tools

5th Axis Overview





Haas 5 axis overview

- This overview is designed to provide the user with a familiarity of Haas 5 axis machine options. During this half day class participants will be exposed to the different axis configuration, set-up considerations, programming changes and workholding options.

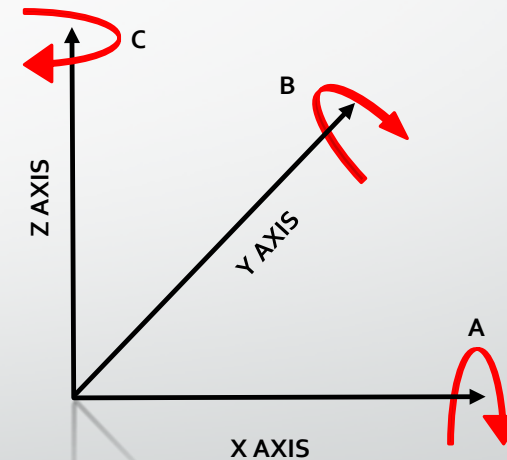


Schedule

- Introductions
- 5 axis machine configurations
- Pivot points
- Break
- Set-up tolerances
- Dynamic work offsets
- Tool center point control
- Workholding considerations

Adding Two Rotational Axis

- X,Y, & Z all have a rotational axis
 - A – Rotates about X.
 - B – Rotates about Y.
 - C – Rotates about Z.
- If two rotational axis are added to a machine, it becomes a 5 axis mill.
 - Three linear axis.
 - Two rotational axis.



What is 5 Axis Machining?

- In summary, 5 axis machining is machining with a combination of 5 linear and rotational movements.

Types of 5 Axis CNC Mills

- 5 axis machines are built for specific scopes of work.
- There are two main types of 5 axis mills:
 - Table/Table rotation
 - Table/Head rotation

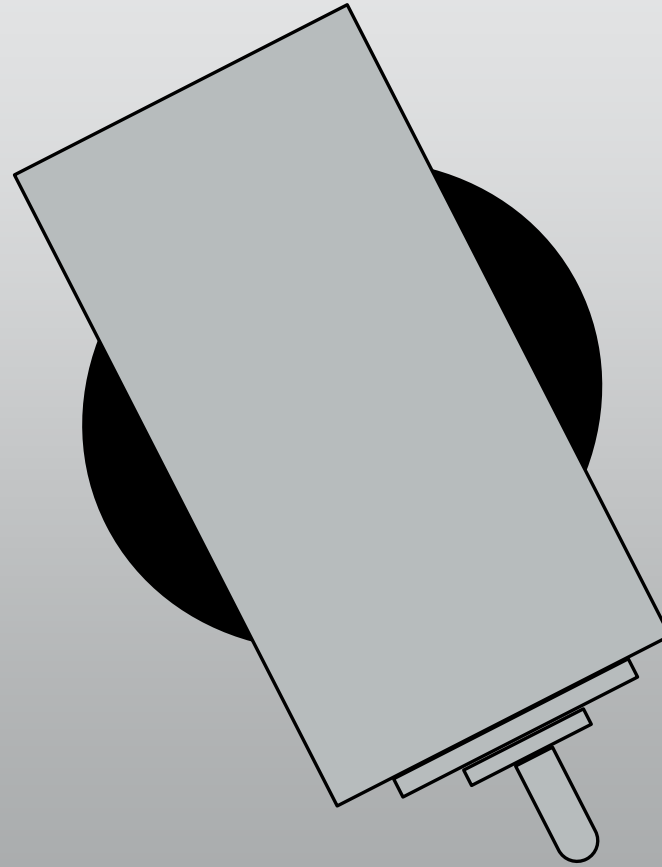
The Haas UMC 750 is a 5 axis CNC mill with dedicated B and C rotational axis built in the table. This would be a table/table machine.



Types of 5 axis CNC Mills

Table/Head Machines

One example of a table/head machine is a 4th axis rotary (A axis) and a tilting head (B axis).



Types of 5 axis CNC Mills

Table/Table Machines

- These machines have both rotational axis built into the table. The table can both rotate and tilt.
- These machines work well for smaller parts.



Types of 5 axis CNC Mills

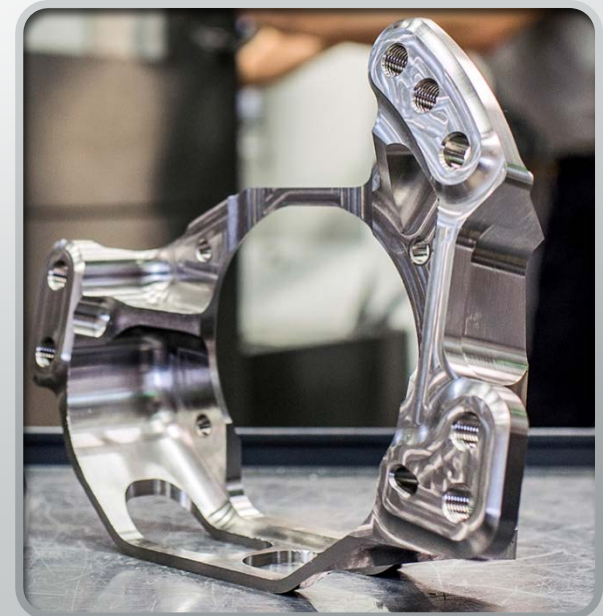
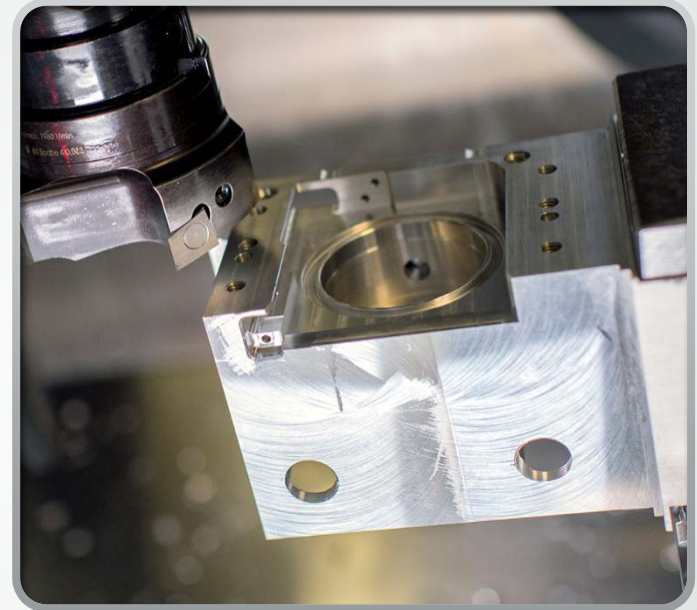
Table/Table Machines


- An affordable table/table machine is a bolt on trunnion style machine.
- These machines utilize a 3 axis mill.
- When needed, the trunnion can be placed on the table.
- Once the servos are enabled, the mill now has five axis capabilities.



Types of Positioning


- Much like 4 axis machining, there are two main types of cutting with 5 axis machining.
 - **Axis Positioning** (top) – this is often called 3+2 machining.
 - These are parts that are not complicated but are put on a five axis mill to reduce the number of set-ups.
 - **Simultaneous** (bottom) – both rotational axis and any of X,Y, or Z axis are moving at the same time while cutting.





3+2 Machining

- 5 axis parts that are positioning or 3+2 programs work as follows:
- **Axis Positioning** (top) – this is often called 3+2 machining.
 - Unclamp Axis.
 - Position Part.
 - Clamp Axis.
 - Cut Part.
 - Repeat on another side of the part.



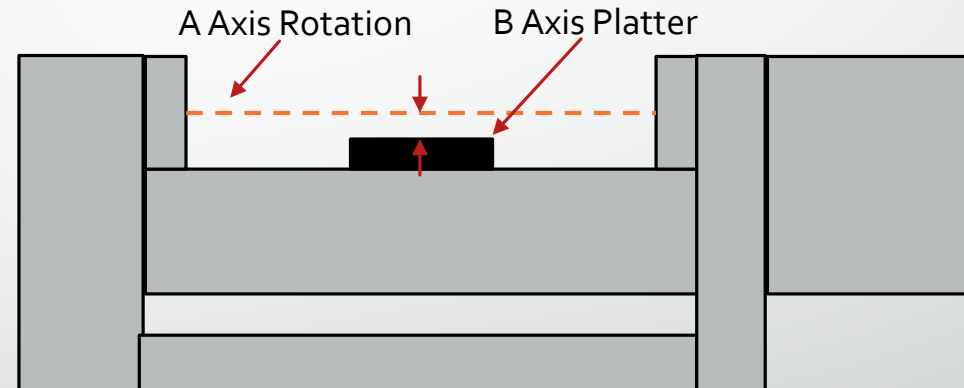
Simultaneous Machining

- 5 axis parts that are simultaneous work as follows:
 - Unclamp axis.
 - Cut Part.
- Note that this is less rigid because the rotational axis are left unclamped.

Pivot Point on 5 Axis Machines

Table/Table Machines

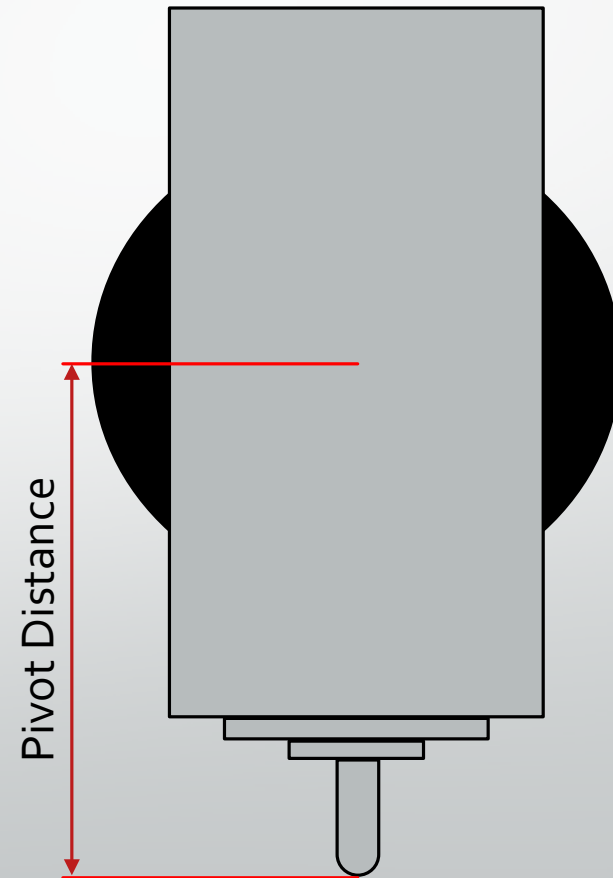
- Machines with both axis of rotation in the table have a pivot point that is different.
- The picture to right shows where the pivot point would be located on a trunnion style 5 axis.
- This distance must be built into the CAM program for the 5 axis moves to be posted correctly in the NC code.



Pivot distance is the difference between the center of the A axis of rotation and the top of the B axis platter.

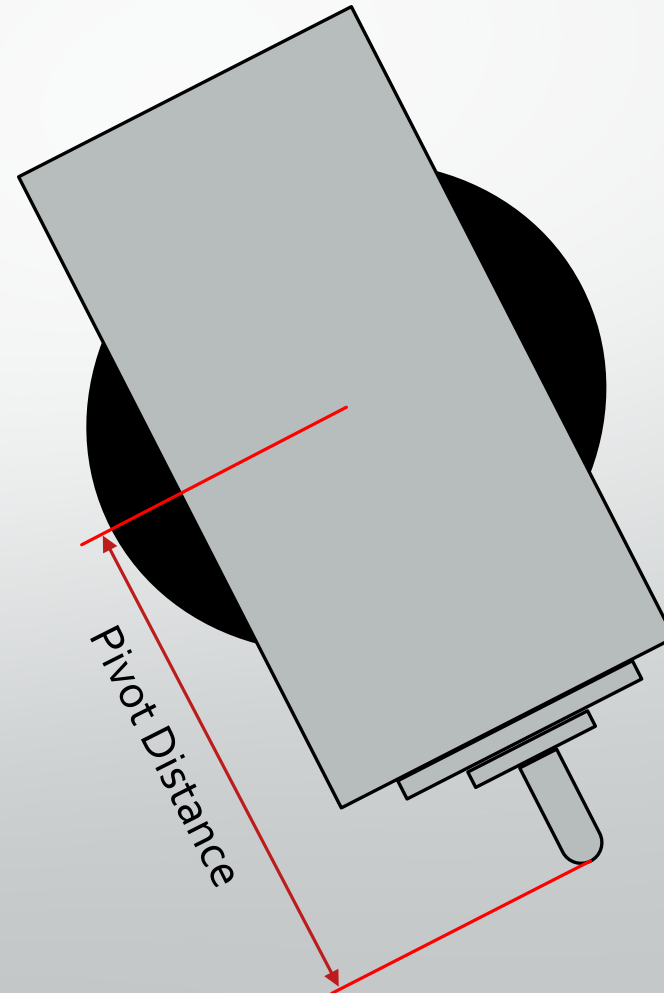
Pivot Point on 5 Axis Machines

- All 5 axis machines have a pivot point.
- On rotating head machines, this distance is calculated one of two ways:
 - From the center of rotation to the tip of the tool.
 - From the center of rotation to the gage line of the spindle.



Pivot Point on 5 Axis Machines

- This distance must be entered in the CAM system before posting the NC code.
- If this is not done, the machine will not properly cut the part.

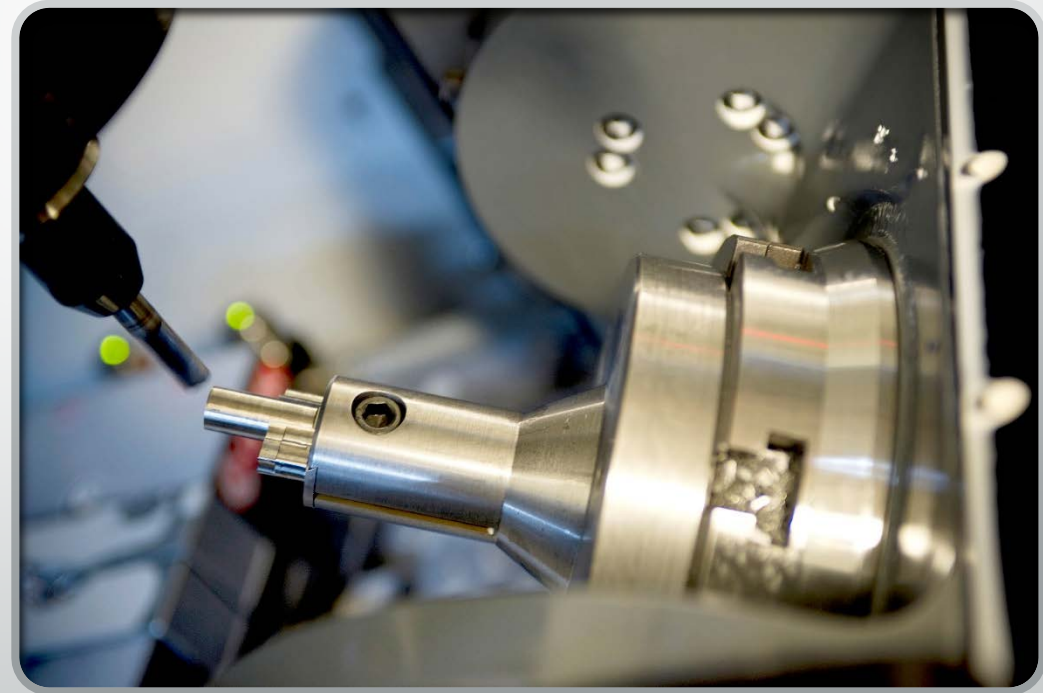


Set-up Tolerances

- Adding two axis of rotation will create a compounding tolerance issue when the part is rotated.
- Because of this, the set-up must have little to no deviation.
- A rule of thumb:
 - A 3 Axis set-up can have 20% of part tolerance used in the set-up.
 - If a part has a $\pm .005$ tolerance for a total of $.010$ deviation, the set-up (possibly a vise) could have $.002$ of run out.
 - A 4 Axis set-up can have 10-15% of the part tolerance used in the set-up.
 - That same $.010$ deviation on a part could only have $.001-.0015$ deviation in the set-up.
 - A 5 axis part can have 5-10% of the part tolerance used in the set-up.
 - This would make the set-up deviation $.0005-.001$.

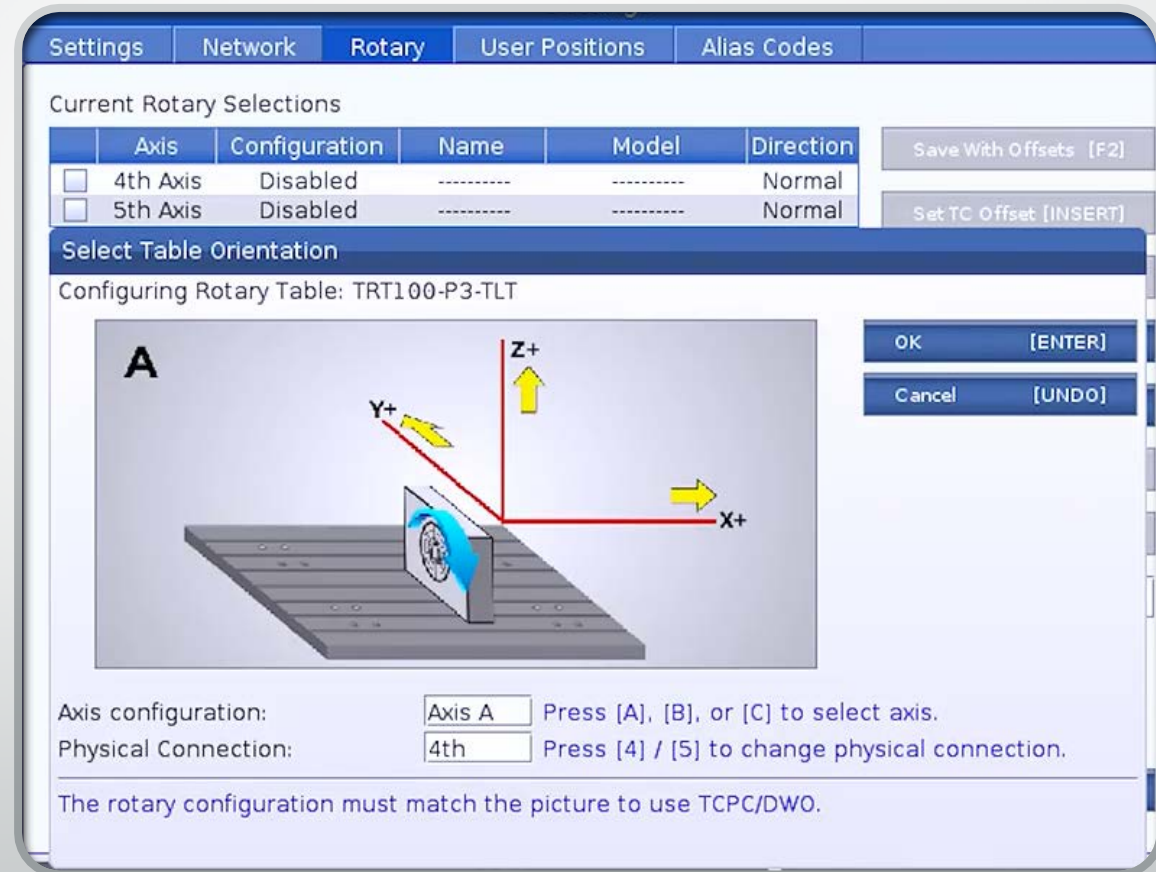
Set-up Tolerances

- A 5 axis part can have 5-10% of the part tolerance in the set-up.
- If a part has a .005 tolerance, this leaves .0005 to .001 tolerance for the entire set-up.
- This tolerance includes:
 - Setting the trunnion parallel to the X axis.
 - Leveling the A axis.
 - Finding X and Y centerline.
 - Finding the Pivot Point.
 - Setting Z height offsets.
 - Setting the B axis offset.



Installing the Trunnion

- Before a part can be made, the trunnion will have to be installed on the machine.
- To do this, the trunnion must be:
 - Placed in the machine.
 - Servo wiring and air connected.
 - Servos enabled on the control.
 - Level the A axis.
 - Indicate the trunnion parallel to the X axis.



Leveling the trunnion A axis

- The A axis must be leveled before any offsets can be recorded.
 - This can be done using a dial indicator to move across the platter in the Y axis.
- If the A axis is not level, any offsets recorded will be off.
 - This could cause the parts produced on the set-up to be made out of tolerance.



Finding Center for X & Y

- The NGC controlled machines utilize the dynamic work offsets if the option is purchased.
- If DWO is not used, it is easier to locate the part to the center of rotation.
 - Fixtures are designed to hold the part on centerline.
 - Programs are easier to make.
 - CAM systems must match the set-up values to post the correct rotational movements.



Finding the Pivot Distance

- The sequence to find the pivot distance on a trunnion:
 - Level the A axis of rotation.
 - Find XY axis center of the platter.
 - Record these values in the work offset you will be using.

Hand Jog				
	Position: (IN)		Jog Rate: 0.0100	
	Work G54	Distance To Go	Machine	Operator
X	-1.0477	17.5610	-11.8800	0.0000
Y	4.7072	2.3800	-5.9300	0.0000
Z	19.0948	0.0000	0.0000	0.0000
B	0.000	0.000	0.000	0.000
C	-2.358	0.000	0.000	0.000

Record the XY machine position

Finding the Pivot Distance Cont.

- The sequence to find the pivot distance on a trunnion:
 - Rotate A 90° .
 - Edge find the Platter.
 - Write down the Y machine position after the edge finding is complete.



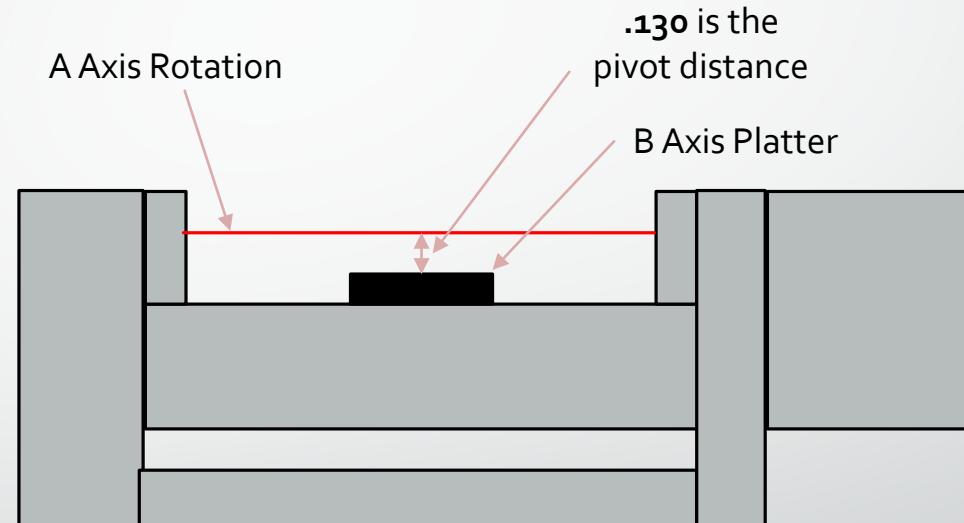
Edge find the platter with the A axis
@ 90°

Hand Jog				
	Position: (IN)		Jog Rate: 0.0100	
	Work G54	Distance To Go	Machine	Operator
X	-1.0477	17.5610	-11.8800	0.0000
Y	4.5772	2.2500	-6.0600	-0.1300
Z	19.0948	0.0000	0.0000	0.0000
B	0.000	0.000	0.000	0.000
C	-2.358	0.000	0.000	0.000

Record the Y axis position after the edge
finding is complete

Finding the Pivot Distance Cont.

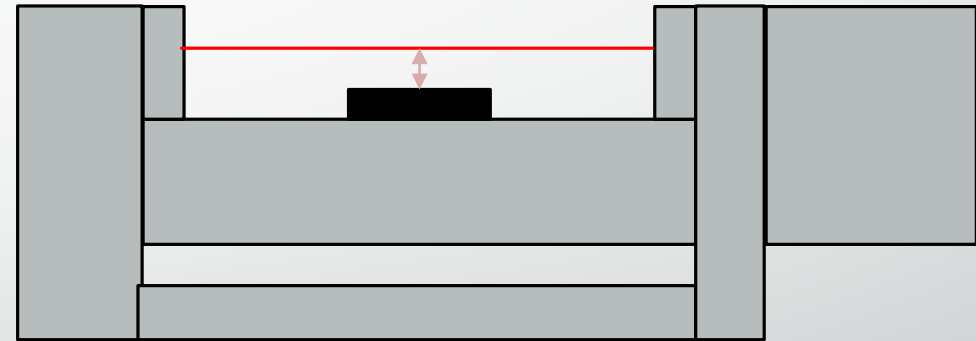
- The sequence to find the pivot distance on a trunnion:
- Subtract the Y axis machine position when the platter was @ 90° from the Y centerline machine position.
 - In this case $10.0057 - 9.8757 = .130$
 - $.130$ would be the pivot point, or the distance the A axis of rotation is above the platter.



Pivot distance is the difference between the center of A axis of rotation and the top of the B axis platter

Setting the Z Axis height

- Z heights will need to be set to the center of the A axis of rotation.
 - This is where the CAM program has Z 0.0 located.
- This can be done by:
 - Leveling the A axis.
 - Finding the Z offset from the top of the platter.
 - Adding the pivot distance distance to the Z offset distance .



The Z axis height offset can be set by finding the Z machine distance to the top of the B axis platter and adding the pivot distance.

Inverse Time Feed

- *G93* is generally used in 4 and 5 axis work when the program is generated using a CAM system. *G93* is a way of translating the linear (inches/min) feedrate into a value that takes rotary motion into account. When *G93* is used, the *F* value will tell you how many times per minute the stroke (tool move) can be repeated.
 - **F** - Feedrate (strokes per minute)
This G code specifies that all *F* (feedrate) values are interpreted as strokes per minute. In other words, the time (in seconds) to complete the programmed motion using *G93* is 60 (seconds) divided by the *F* value.
 - When using the *G93*, the max programmable feedrate is 4500 for inverse time feed.

Inverse Time Feed

- If a line of code is programmed with a high feed rate of $F1000$. or $.05$ seconds, this means the linear feed moves at the programmed feed while the rotary moves over the given time. This means the axis traveling on that line of code will arrive at a precise time.
 - The user has finer control of the toolpath and generally can achieve more precise results as the inverse feed creates a "time" based path as opposed to a constant feed per "x".
 - When $G93$ is used, feedrate (F) is mandatory for all interpolated motion blocks. Therefore, each non-rapid motion block must have its own feedrate (F) specification.

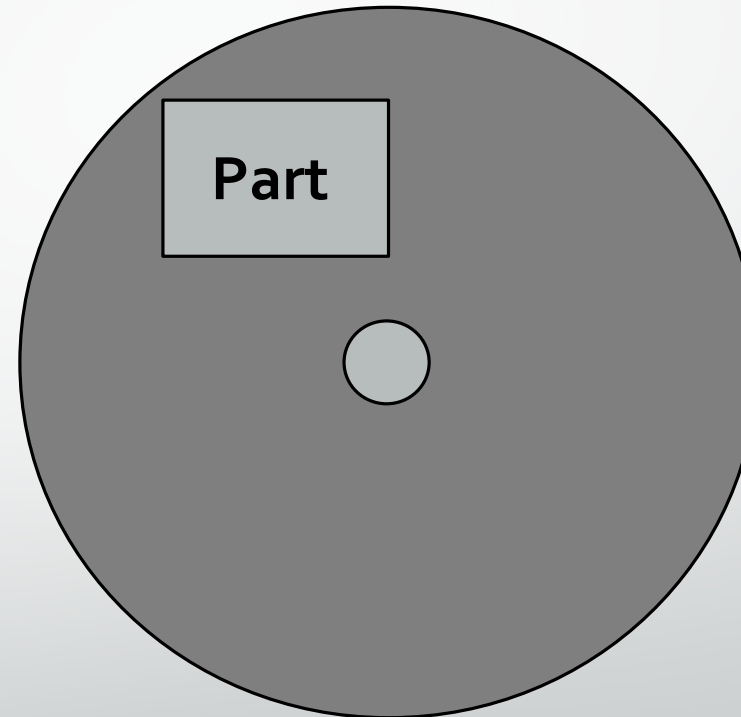
Rotary Axis Calibration Tool

- This rotary-axis calibration tool simplifies the process of setting your machine's rotary zero point (MRZP) offsets, which are used by the Dynamic Work Offsets/Tool Center Point Control (DWO/TCPC) option to make 4 and 5 axis setup and calibration easier. MRZP is a set of control parameters that define the centers of rotation for the rotary axes relative to the home positions of the linear axes. Note: The taper of the master gauge-length tool will match the taper of the machine for which the option is ordered.
 - Includes a precision calibration ball, master gauge-length tool, all necessary hardware, and a heavy-duty case.
 - Allows recalibration of the rotary zero point offsets to compensate for environmental factors, or a machine crash.
 - Improves the overall positioning accuracy of the rotary axes.



Dynamic Work Offsets

- Any Haas UMC or Trunnion machine with a NGC control can utilize DWO.
- This means that a part can be placed anywhere on the table.
 - The CAM program is still made from the centerline of rotation.
- The distance from the center of rotation to the part is recorded in the machine offsets.
 - The machine will make the calculations to machine the part where it sits on the table.

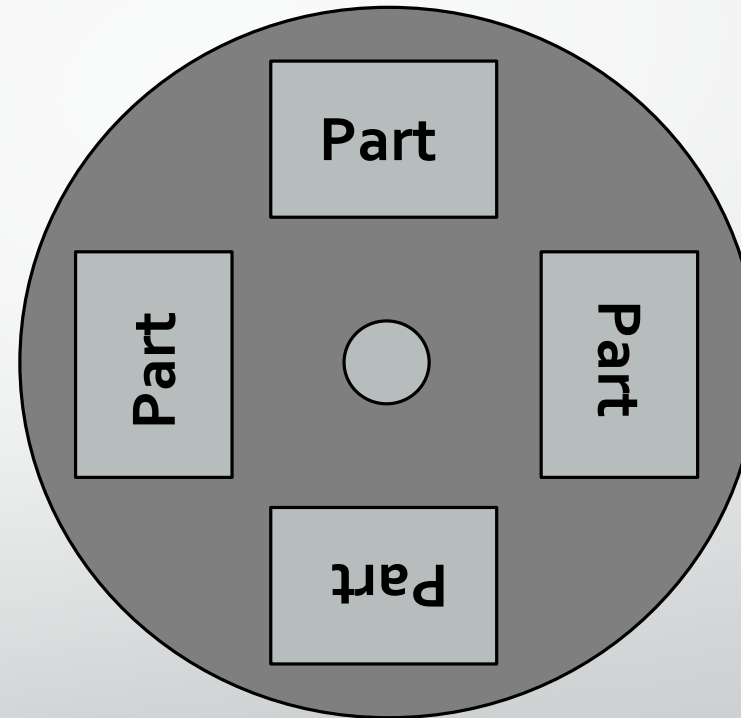


Top view of the machine
table

Dynamic Work Offsets

- Dynamic offsets allow numerous setups to be used on one trunnion, as long as there is enough space.
- The Haas mills have 105 work offsets.

[WATCH VIDEO](#)

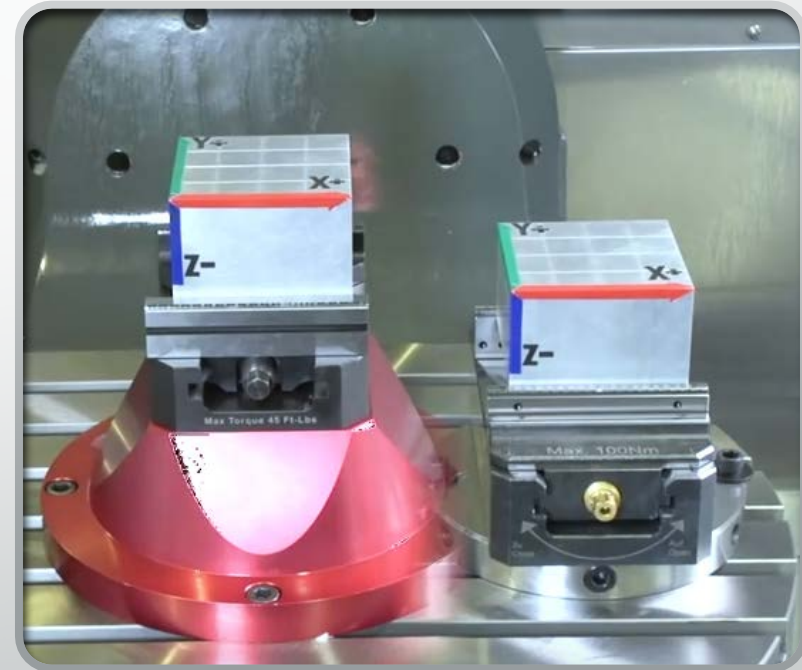


Top view of the machine table

Tool Center Point Control

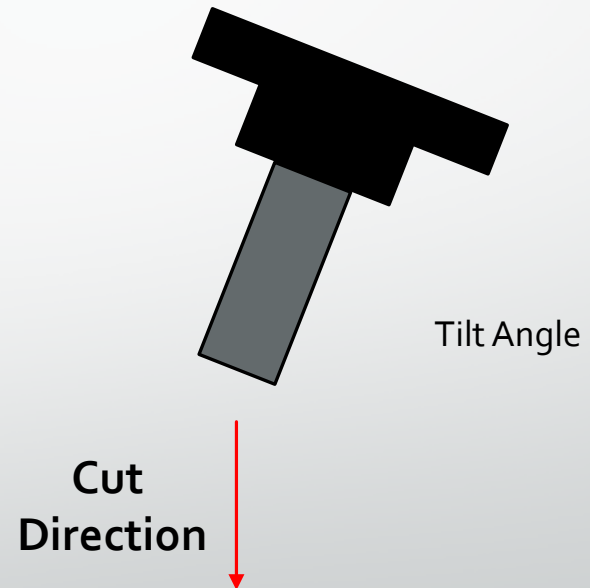
- Tool center point control (TCPC) is activated with a G234 code. Once activated, the option allows for a multi-axis contouring program to be run on any spot on the table, regardless of where it was programmed.
- TCPC combines the machine's MZRP and the part's work offset to adjust the program to the part's actual position on the table.
 - G243 replaces the G43 in the CNC code.
- TCPC essentially allows multiple setups to use the same program regardless of the position in the machine.

WATCH VIDEO



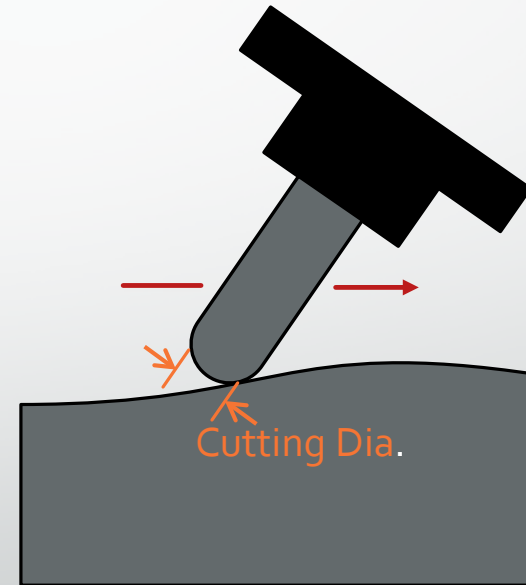
Tilt Angles

- 5 axis programs can have tilt angles applied to the endmill.
- These angles can be used to angle the tool to the side.
- This can also change the cutting pressure or entry angle of the end mill.



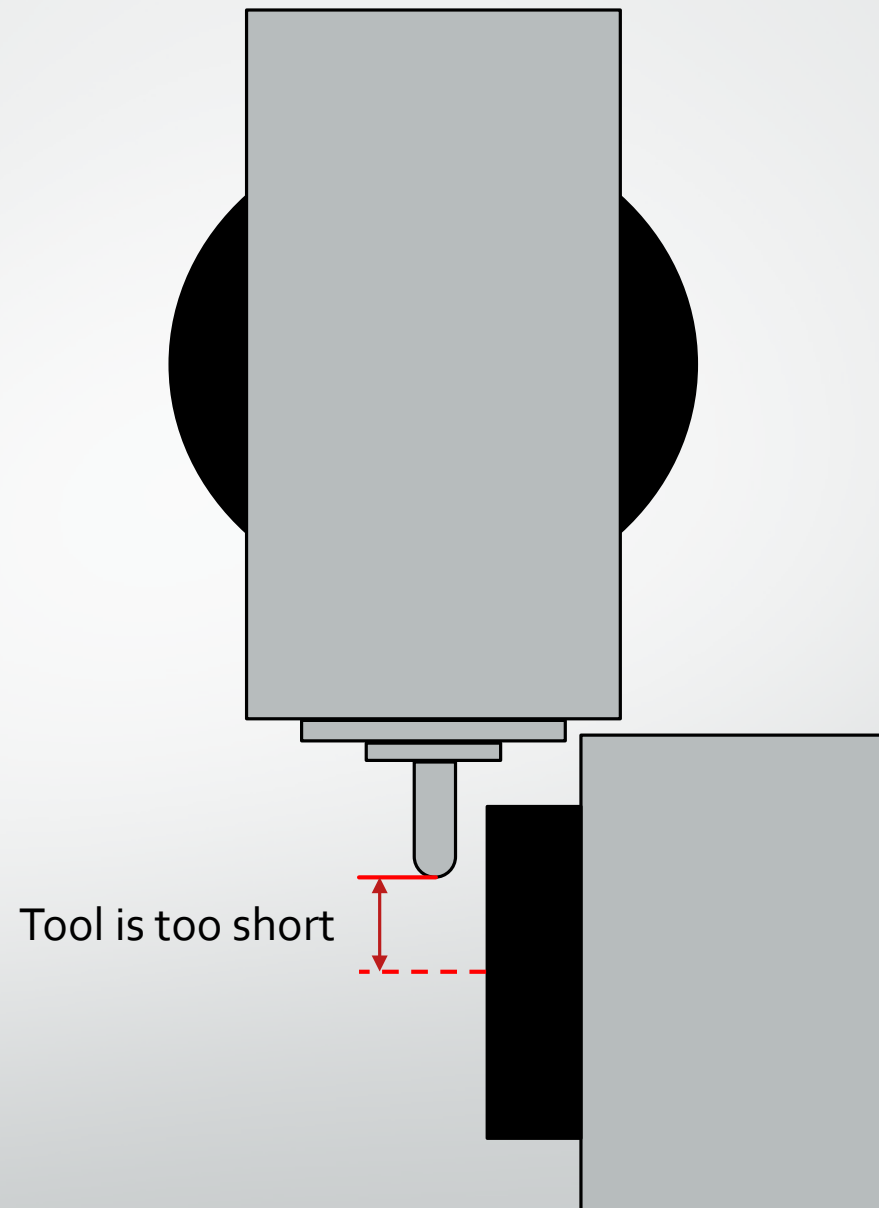
Tilt Angles

- Tilting the tool also helps during surfacing since the tool can cut with the larger cutting diameter, as opposed to cutting with the centerline of the tool.

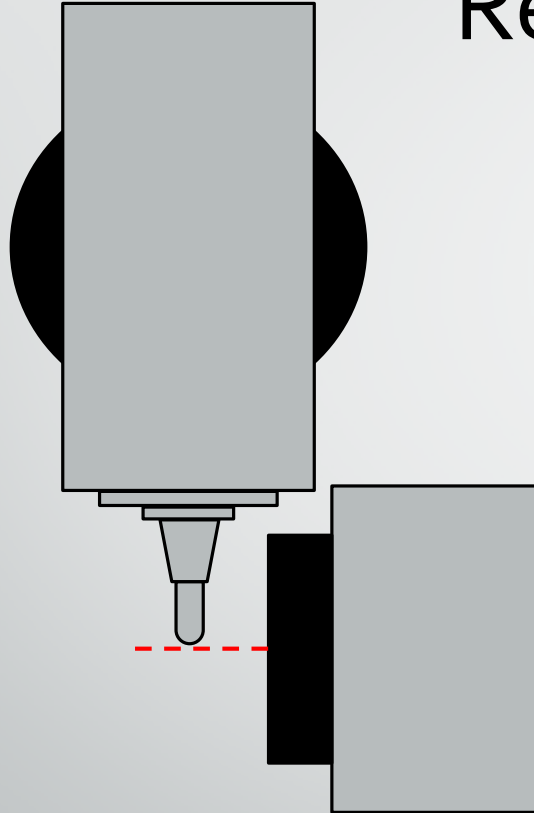


Restricted Work Area

- Since 5 axis machines use wide trunnions, this often restricts the cutting area.
 - In order to reach X and Y zero when table is rotated, the tool has to be able to extend the radius of the trunnion platter, or the workpiece has to be offset from the trunnion.

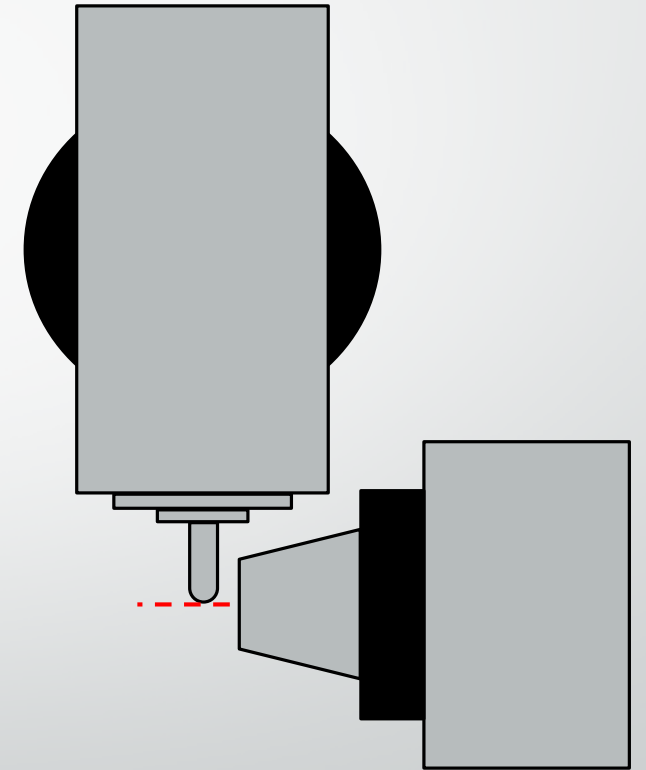


Restricted Work Area



The Solution

- (Right) Oftentimes, special fixtures are used to offset the workpiece from the trunnion for better navigation. This is the preferred method out of the two options since tools should be as short as possible.
- (Left) It is common for machines with 4th or 5th axis setups to use extended tooling to avoid the trunnion. This also helps when dealing with complex geometry and getting into tight spaces.





5 Axis Fixturing

- There are multiple forms of workholding in 5 axis setups. The following setups are examples of common solutions for avoiding work area restrictions while still providing a rigid setup.

Dovetail Fixturing

- Dovetail workholding is very popular with 4th or 5th axis machining. It has many benefits such as:
 - Holding force – Dovetail fixtures have high holding forces due to the aggressive grip.
 - More surfaces – Instead of clamping the outside of the part in a vise, taking up two surfaces, only one side of the part is used.
 - Discreet – Dovetail fixtures fit neatly under the workpiece. They also extend the part away from the trunnion.
- Stock can be cut for these fixtures, but blanks can also be purchased with pre-cut dovetails.



5 Axis Vises

- 5 axis vises generally sit higher than standard vises or sit on a riser. They also have a smaller profile.
- The jaws of these vises are serrated or even have clamps. This helps hold the part with great force with minimal grip.
- These vises are also self-centering, meaning both jaws move. This allows multiple parts to be clamped on the same centerline regardless of part geometry.



5 Axis Modular Solutions

- Workholding setups benefit from modular fixturing. This is where multiple setups can be indexed using a fixture that accepts multiple replaceable plates.
- Identical fixtures can be replaced with minimal variation in the setup to reduce down time.
- Different fixtures such as vises, collets, and dovetail fixturing can be changed out and setup on the same modular plate.

WATCH VIDEO

