Mill Tool Life Troubleshooting - Tap

Scan code to get the latest version of this document

Translation Available

Incorrect Tapping RPM / Feedrate - Mill

You can tap inch and metric threads in both inch and metric modes (Setting 9). Be sure to use the correct feedrate for Setting 9 and the threads that you are tapping.

The tapping feedrate (**F**) is based on the tap RPM (**S**). **F** and **S** work together to time the Z-Axis feed with the spindle position to create precise threads. Refer to the tap manufacturer's recommendations for spindle speeds to use.

Typically, you program tapping in G94, feed per minute. G94 is the machine's default mode. You can also tap in G95 mode, but remember to return the control to G94 before milling.

Corrective Action:

Use these formulas to determine the correct RPM and feedrate to use:

Feedrate Formulas for Setting 9 = **Inch**

Inch Tap		**Metric Tap (pitch in mm)**	
G94 F	RPM / TPI	G94 F	Pitch * RPM / 25.4
G95 F	1 / TPI	G95 F	Pitch / 25.4

Feedrate Formulas for Setting 9 = **mm**

Inch Tap		**Metric Tap (pitch in mm)**	
G94 F	RPM / TPI * 25.4	G94 F	Pitch * RPM
G95 F	1 / TPI * 25.4	G95 F	Pitch

Pitch is the distance from one thread to the next. This value is in millimeters for metric taps. For example, in a M6 x 1. thread, the pitch is 1.0.

TPI is the threads per inch on Inch (unified) taps. For example a 1/4-20 tap makes 20 threads per inch.

In inch mode, round feedrates to (4) decimal places. In mm mode, round feedrates to (3) decimal places.

Coolant Issues

Incorrectly aimed coolant nozzles or obstructions in the stream can prevent coolant from reaching the cutting area. Adjust your coolant nozzles to deliver coolant to the cutting area.

Be sure to use the recommended coolant mixture concentration in your applications. If your concentration is too lean, the reduced lubricity can negatively affect your tool life and surface finish.
There are many different coolants for different applications and materials. Contact your coolant dealer for advice.

Refer to the Machine Tool Coolant Series page for videos and articles about maintaining your coolant system.

The Rigid Tap Option is not Activated

The Rigid Tap option will synchronize the spindle with the axis feed to produce an accurate thread when a tapping canned cycle is commanded.

To verify this option is turned on, check the value of the Rigid Tap parameter (**57 bit 4**):

1. Press **[PARAM/DIAGNOS]**.
2. Type **57** and press the **[DOWN]** cursor arrow.
3. Look at the value of the **RIGID TAPPING** bit.

This bit has a value of **1** if the Rigid Tap option is activated.

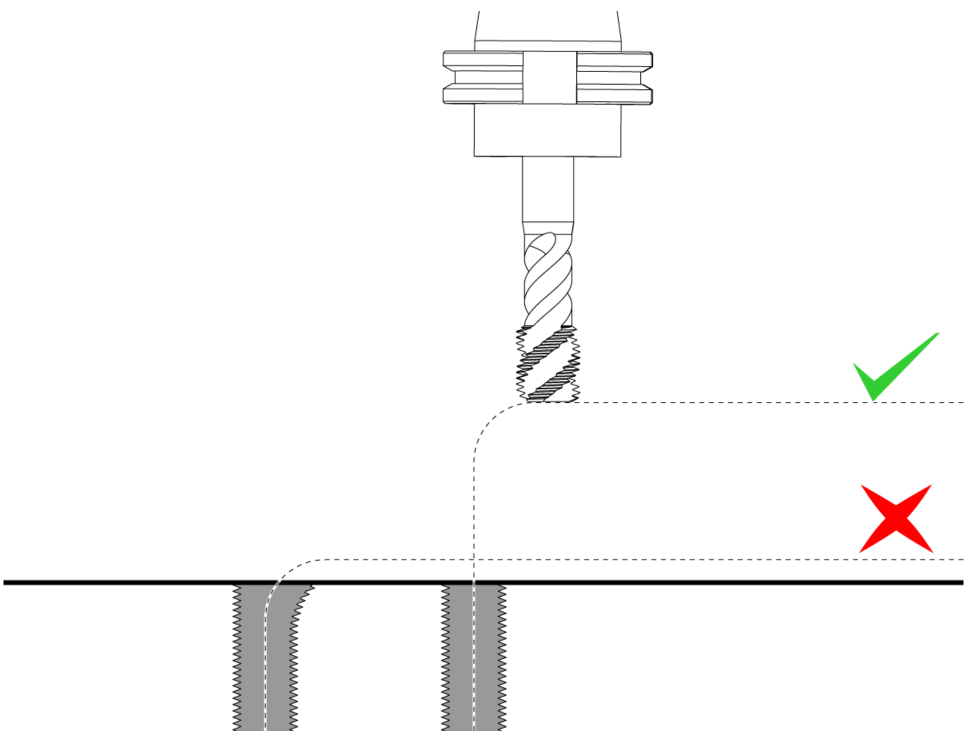
If the bit has a value of **0**, purchase the Rigid Tap option from parts.haascnc.com.

The Tapping R Plane is Too Close to the Workpiece - Mill

The positioning moves between hole locations in a canned cycle are rapid movements. If the R plane is too close to the workpiece, the tool can begin to feed down before the axes reach the correct hole location. This can cause the tool to hit the edge of the hole as it enters, or it could start drilling/tapping at the wrong location.

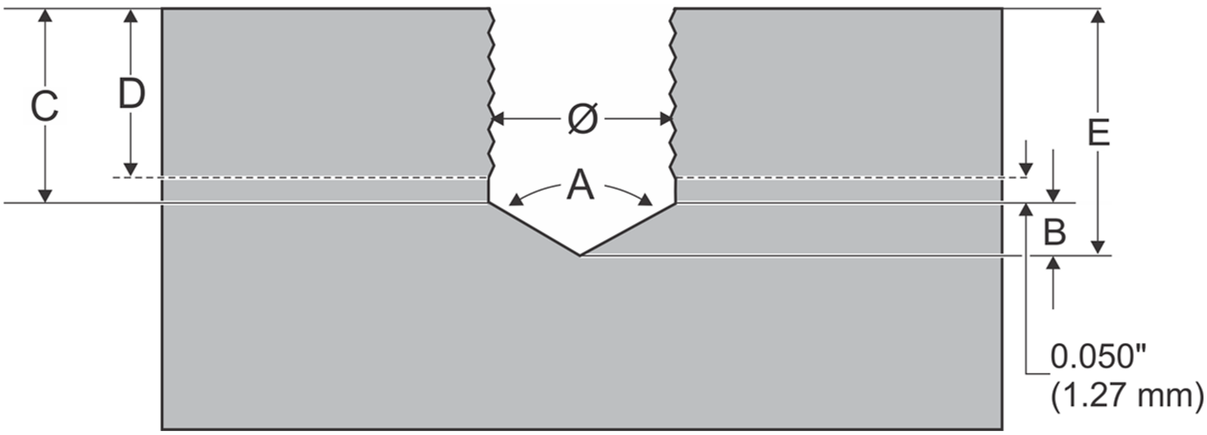
Corrective Action:

Increase the R plane to a minimum of .100" (2.54mm) above the surface so that the axes are in position before the canned motion starts.



The Drilled Hole is Too Shallow

The hole may not be drilled deep enough.



The full-diameter depth [C] of the hole is shallower than what is programmed because of the drill tip angle on the drill [A]. Program the full-diameter depth [C] at least 0.050” deeper than the tap depth [D].

Corrective Action:

Your programmed drill depth [E] should compensate for the drill tip depth [B] plus an additional .050" (1.27mm) clearance.

To calculate drill tip depth for a drill (Ø denotes the drill diameter):

Drill Point Angle [A]	Drill Point Depth [B] Formula
60°	0.866 X Ø
82°	0.575 X Ø
90°	0.500 X Ø
118°	0.300 X Ø
120°	0.288 X Ø
135°	0.207 X Ø

Example: To calculate for a 118-degree drill tip depth, multiply the drill diameter by 0.3, e.g.:

0.250 drill diameter x .3 = 0.075 drill tip depth

RPM or Feed Rate Too High - Mill

The commanded RPM may be too high for the spindle and axis motion to remain synchronized during the tapping cycle.

Haas Mills can rigid tap at up to 2000 RPM or 150 IPM (3810 mm/min)

Corrective Action:

Reduce the programmed RPM and feedrate to a maximum of 2000RPM or 150IPM (3810 mm/min) to ensure the spindle and axis remain synchronized.

Note: DT- and DM-series mills can tap up to 5000 RPM.

Incorrect Hole Size

Make sure that the drilled hole is the correct diameter for the tap. Refer to theHaas Shop Notes handbook for tap drill charts and tap drill size formulas.

Note: Cut taps and form taps require different-sized holes to tap the same thread diameters.

Excessive Tap Runout - Mill

Excessive runout causes an inconstant load on the tool as it rotates. This can lead to premature tool wear and issues with accuracy and surface finish.

Tool runout should not exceed 0.0003" (0.076 mm). To check the tool runout, place a dial indicator on the tool, and then rotate the spindle by hand.