Haas Technical Documentation

Lathe Threading Chatter Troubleshooting

Scan code to get the latest version of this document

Translation Available

Excessive Tool Wear

If your tool has excessive wear, the cutting forces generated by the cut will increase. These increased cutting forces can lead to chatter during the cut.

Inspect your tool and replace it if necessary.

It is normal for tools to wear over time. In a stable machining process the tool wear is predictable. This will allow you to use the Tool Life Management systems that come standard in your Haas control. The tool life information can be input in to the Haas control to alert the operator to replace the tool before it negatively affects your machining process.

Refer to the Tool Life Management section for details on how to use these systems.

Incorrect A or P Value with G76

Use the optional A and P codes with a G76 threading cycle to control the infeed angle (A) and cutting method (P).

The A value specifies the infeed angle, or the tool nose angle, for the thread. This value can range from 0 to 120 degrees; do not use a decimal point. If you do not specify an A value, the control assumes zero.

The P value specifies the cutting method. The options are P1, P2, P3 and P4. These control whether the depth of cut is a constant depth with each pass, or a constant cutting amount that reduces the depth of cut the deeper into the thread it goes. The P value also specifies single edge or double edge cutting. If you do not specify a P code, the control selects the P code designated in Setting 232.

P1 Single edge cutting, cutting amount constant
P2 Double edge cutting, cutting amount constant
P3 Single edge cutting, cutting depth constant
P4 Double edge cutting, cutting depth constant

Helpful Hint:

To reduce chatter while threading, use an A value 1-3 degrees less than the included angle of the thread; for example, use an**A57, A58 or A59** to cut a 60-degree included angle thread. This allows for clearance on the back side of the insert so it does not contact the thread form until the final depth cut.

Refer to G76 Threading Cycle, Multiple Pass (Group 00) - Lathefor more details on threading.

Threading Tool Selection

The insert geometry, insert grade and/or insert shim are incorrect for the application.

Corrective Action:

Consult with your tooling manufacturer for recommendations on proper tool selection for your application, and make adjustments where necessary.

Helpful Hint 1:

Threading inserts come in many different shapes and sizes. Topping inserts are the most common threading inserts used on a CNC machine. Topping inserts cut one full thread form at a time, and they are designed for a specific pitch.

Be sure to use the correct insert for your application.

Helpful Hint 2:

Check under the insert for a seat, sometimes referred to as a shim. This seat is ground to an angle. Different thread diameters require different seat angles. Check with the insert manufacturer to be sure that you are using the correct seat for the thread being cut.

Settings 86 and 99 Have Incorrect Values

Setting 99 (THREAD MIN CUT) and setting 86 (THREAD FINISH ALLOWANCE) control the minimum roughing depth-of-cut and the final finish pass depth-of-cut for a G76 threading cycle.

Corrective action:

Use an infeed chart from the tooling manufacturer to set the values for Setting 86 and 99.

This chart specifies the minimum and final cut values for a specific thread size, as well as the total number of depth cuts for roughing.

The Threading Tool is Too Long

The OD or ID threading tool is too long, or it is not properly supported.

Corrective Action:

Adjust the OD stick tool holder so that the tool holder's projection length from the turret is as short as possible. If you must use a longer stick tool for clearance, consider using a Haas Twin Turn or Extended Twin Turn BOT holder to fully support the stick tool. You can get these holders on parts.haascnc.com.

Adjust the ID boring bar holder so that the tool's projection length from the tool older is as short as possible. The material type of the bar will affect its stability. A steel boring bar is stable up to a stick-out of 3 times the diameter. A carbide boring bar is stable at a length up to 5 times its diameter. If you must exceed these limits, you must compromise your cut parameters to compensate for the reduced stability—reduce the depth-of-cut, or spindle speed to compensate.

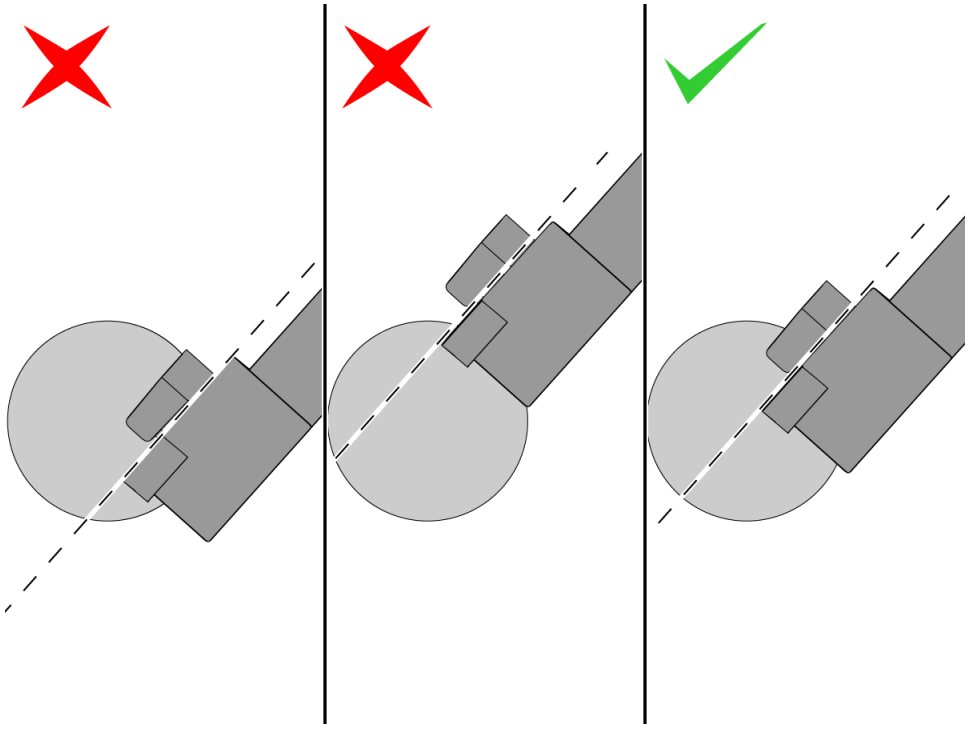
Helpful Hints:

If you need to use a bushing for the boring bar, use a split bushing to increase bar support.

Special vibration-dampening boring bars are available when you need an extreme length-to-diameter ratio to machine an ID thread.

The Tool is Not On the Spindle Centerline

If the cutting edge of your tool is not on the spindle centerline, excessive cutting forces can cause chatter, accuracy, and tool life issues.



Corrective Action:

Make sure that the stick tools are the correct size for your turret or tool holder.

Make sure that the seat under the insert is the correct thickness.

On Y-axis lathes, you can use a Y-axis tool offset to bring the cutting edge to the spindle centerline.

Inspect and correct any alignment errors in your machine tool.

Insufficient Support on the Workpiece

If the workpiece is not properly supported, it will begin to vibrate and introduce chatter into the cut.

As a general rule, if the portion of the workpiece that extends past the chuck exceeds a diameter-to-length ratio of 3:1, use a tailstock to stabilize the cut.

If the length-to-diameter ratio of your workpiece exceeds 10:1, you may need to use the additional support of a steady rest, different workholding, or a different machining strategy to stabilize the cut.

Helpful Hint:
Consider using the Spindle Speed Variation (SSV) feature to disrupt chatter.

For more details on tailstocks watch Tailstock Fundamentals: How to use the Tailstock on a Haas Lathe.

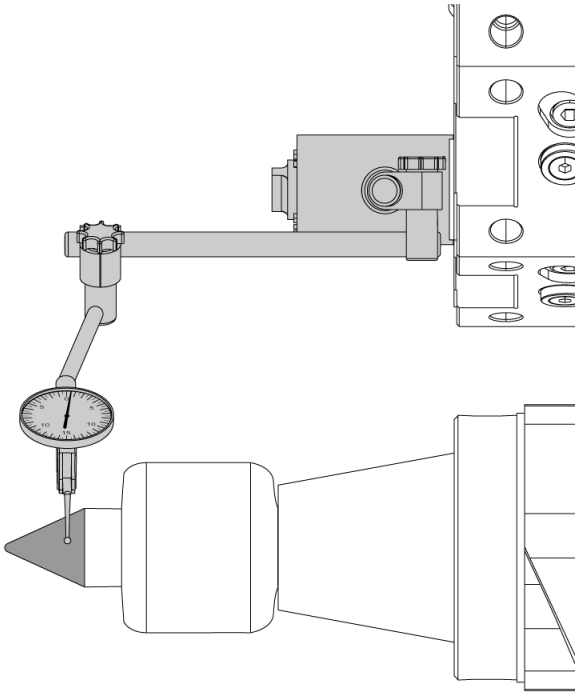
Worn or Damaged Live Center

A worn or damaged live center can introduce vibrations and let the part move. This can cause chatter, taper, poor surface finish, and tool life issues.

Inspect live centers for excessive runout and damaged bearings while they’re still in the machine.

Check runout by placing an indicator on the 60 degree point, then gently rotate the center's point. The TIR should be within the

manufacturer’s specification.

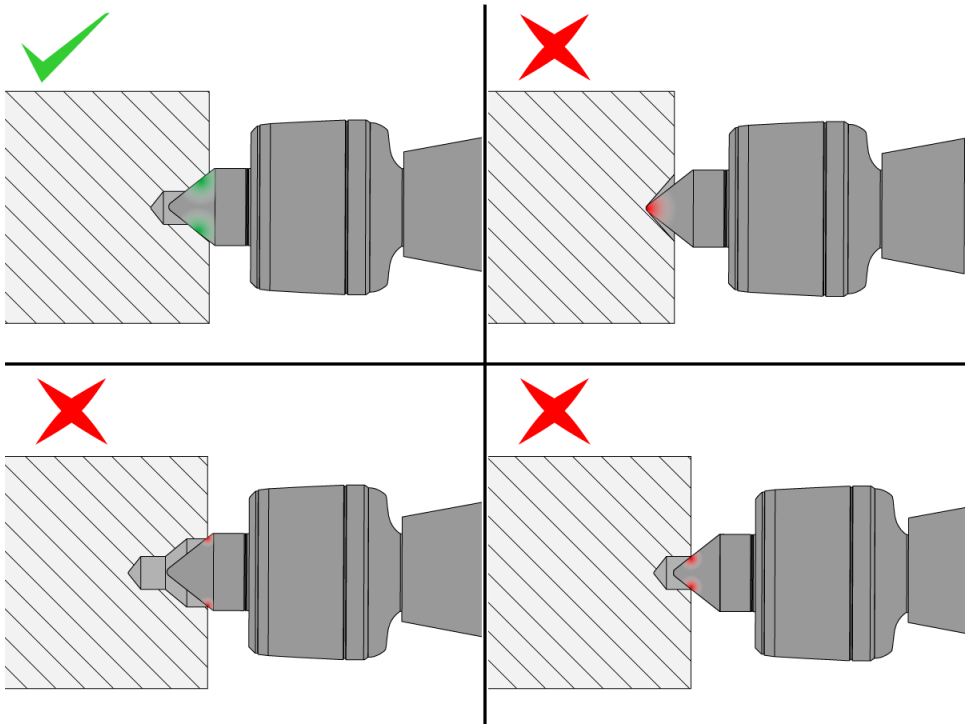


Check for bearing wear by gripping the point firmly and rotating it in one direction. The spindle should turn freely, if you can feel hesitations or roughness this indicates bearing wear.

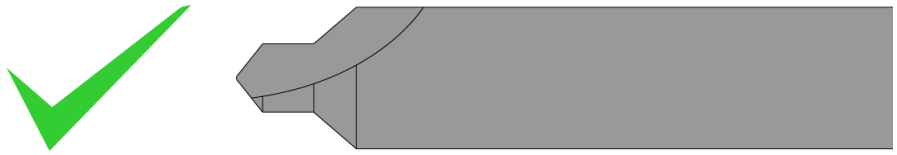
Note: Live centers have a service life and a maintenance schedule. Refer to the live center manufacturer's documentation for details on your live center.
Tip: Excessive tailstock pressure can prematurely wear out your live center.

The Center-Drilled Hole is Incorrect or Damaged

If the center-drilled hole has the wrong angle, is too small, too shallow, too deep, or is damaged, the live center will not have sufficient contact with the workpiece to properly stabilize the cut.



Be sure to use a 60° center drill tool. Countersink tools do not have the tip relief required for the live center.



Inspect the center-drilled hole. If necessary, machine it again.

For more details center-drill requirements watch this video on Tailstock Fundamentals.

Coolant Issues

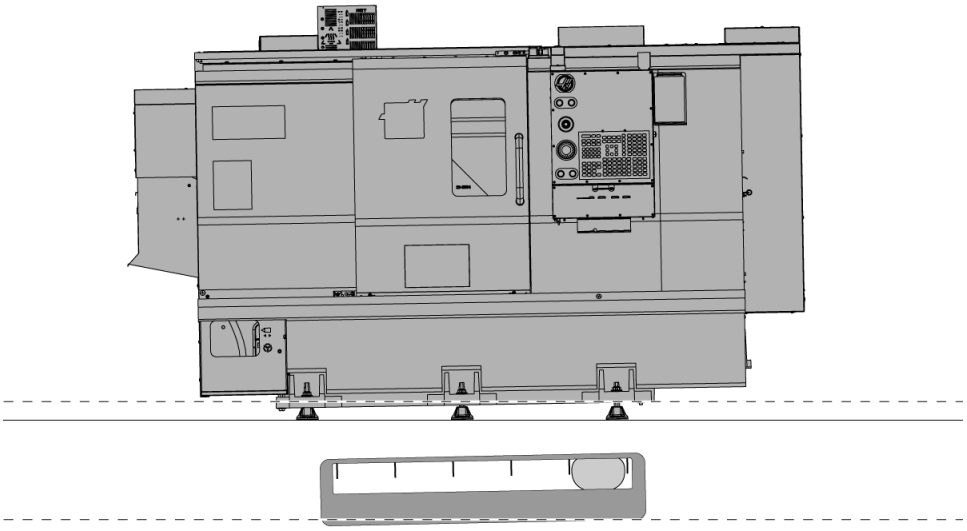
Incorrectly aimed coolant nozzles or obstructions in the stream can prevent coolant from reaching the cutting area. Adjust your coolant nozzles to deliver coolant to the cutting area.

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 Machine is Not Level - Lathe

To operate correctly, the machine must be level. An out-of-level machine can have problems such as poor surface finish, tapered parts, accuracy and repeatability issues, out-of-round circular motion, and out-of-true linear motion.



Watch the Lathe Leveling video to learn how to properly level your Haas lathe, or contact your Haas Factory Outlet to have your machine's level checked.

Inadequate Foundation

The machine must sit on a solid and stable foundation. Refer toPre-Installation Information for a full description of the foundation requirements.

If the foundation is badly cracked, move the machine to a location with a solid foundation, or repair the foundation.

The machine should sit on one continuous slab of reinforced concrete. If the machine straddles more than one slab, you may need to move the machine to a single, continuous slab.

Example of a Machine Set Across Multiple Concrete Slabs