

Tip of the Day Quick-Pick List



| Tip of the Day Title | Sequence Number | Mill or Lathe | Skillset Level | Reason to view this Tip of the Day | Video Length |
|---|-----------------|---------------|----------------|--|--------------|
| Entry Level | | | | | |
| What is G-Code? | 1 | M/L | Entry | Learn the basics of G codes in this TOD. This video will teach you G and M code basics and the general organization required to effectively use the G code language. | 6:18 |
| How to Square and Indicate a Vise on Your CNC Mill | 2 | M | Entry | Entry level skillset. This skill is important for the machinist to align workpiece with Machine Axis. This is the initial Tip of the Day to watch. | 8:01 |
| How to: Set Tool Lengths and Work Offsets | 3 | M | Entry | This Tip of the Day shows the machinist how to program Tool Lengths and enter work offsets into the G Code program, manually. This is a must see video! | 12:00 |
| How To Set a Work Offset with an Edge Finder and an Indicator | 4 | M | Entry | Entry level skillset. This skill is important for the machinist to establish a datum/work coordinate on the edge of a work piece. This is the second TOD to watch. | 10:22 |
| How To Manually Pick Up a Bore or a Hole with an Indicator | 5 | M | Entry | Entry level skillset. This skill is important for the machinist to be able to establish a datum/work coordinate from the center of a bore (or hole) . This is the third TOD to watch. | 7:43 |
| Reduce Setup Time with the Haas "Next Tool" button | 6 | M | Entry | Part of the machine setup is to establish the 'length off-sets' for each tool. This TOD shows you how to take advantage of the Haas "Next Tool" button to speed the process of setting these off-sets. | 2:26 |
| Avoid Disaster Using the Haas Single Block Function | 7 | M | Entry | The 'Single Block' function allows the machinist to execute their program one line at a time, for the purpose of program verification. This function reduces the chances of a 'crash' based on the single line verification process. | 4:28 |
| Easily convert Your MDI Program to Memory | 8 | M/L | Entry | All Programs for CNC's are 'files'. The MDI program function allows the machinist to write simple commands into the Haas Control. This TOD shows you how to save those simple commands to a file in memory for future use. | 2:13 |
| Copy your CNC Programs Quickly and Easily | 9 | M/L | Entry | Learn how to quickly copy your existing CNC program | 1:06 |

| Tip of the Day Title | Sequence Number | Mill or Lathe | Skillset Level | Reason to view this Tip of the Day | Video Length |
|--|------------------------|----------------------|-----------------------|--|---------------------|
| Quick and Easy Backup of Your Haas CNC Programs | 10 | M/L | Entry | As with any computer, you always want to back up your data. This TOD shows the machinist how to simply back-up their programs. | 3:49 |
| How To Take a Screenshot on Your Haas Control | 11 | M/L | Entry | There are many reasons that a machinist may need to 'Take a Picture' of the control screen, such as: service or application trouble shooting. This TOD shows how to take a screenshot. | 3:39 |
| Easily Program a Tap in Inch Mode on your Haas VMC | 12 | M | Entry | This TOD teaches the machinist how to program tapping cycles in 'Inch' mode. | 7:25 |
| Easily Program a Tap in Metric Mode on Your Haas VMC | 13 | M | Entry | This TOD teaches the machinist how to program tapping cycles in 'metric' mode. | 7:27 |
| CNC Engrave Made Easy using G47 | 14 | M | Entry | This TOD explains how to engrave using the G47 engraving cycles. | 4:04 |
| Troubleshoot Your Lathe G71 and G72 Roughing Cycles Quickly | 15 | L | Entry | This TOD provides the solution and a way to troubleshoot an alarm that might occur during lathe roughing cycles. | 6:35 |
| Intermediate Level | | | | | |
| Troubleshoot Your Haas Probe, Part 1 | 1 | M | Intermediate | This TOD discusses the most common root causes for probing difficulties and how to get the machine back up and running. | 7:39 |
| Troubleshoot Your Haas Probe, Part 2 | 2 | M | Intermediate | This TOD discusses more probe troubleshooting tips. | 6:59 |
| Stringy Chips Wrapped Around Your Tools? Mark has a Solution! | 3 | M/L | Intermediate | This TOD discusses the problems with chip build up on cutting tools. This video shows the solutions that Haas has created and suggestions to ensure the tools are chip free. | 4:22 |
| Plot Your Cutting Path and Clamp Locations using Your Haas CNC | 4 | M | Intermediate | This TOD demonstrates a simple way to plot out your toe clamp positions on difficult part setups. Also shows how to create a very useful tool for any machinist's tool box. | 6:55 |
| How Canned Cycles Work with G98 and G99 | 5 | M | Intermediate | This TOD teaches you how to avoid possible crashes, program efficiently by explaining how 'not to cut air' and how to set your return planes from feature to feature | 7:49 |
| Simple Peck Tapping Using a G84 Tapping Cycle | 6 | M | Intermediate | This TOD explains what peck tapping is and how to peck tap with your Haas Mill. | 3:11 |
| Faster Tapping with Setting 130-CNC Milling | 7 | M | Intermediate | This TOD shows how to save valuable cycle time by retracting out of a tapped hole faster than you feed in. | 4:01 |

| Tip of the Day Title | Sequence Number | Mill or Lathe | Skillset Level | Reason to view this Tip of the Day | Video Length |
|--|-----------------|---------------|----------------|---|--------------|
| Easily keep Track of Your Parts using the Haas M30 counters | 8 | M/L | Intermediate | This TOD explains how to use the the M30 counters on the control to display to total number of parts completed. | 4:23 |
| Easily Engrave Sequential Serial Numbers | 9 | M | Intermediate | This TOD explains how to engrave sequencial serial numbers. | 3:32 |
| Easily Cylindrical Engraving and Machining: Use G47/G107 No CAM needed. | 10 | M | Intermediate | This TOD explains how to use the cylindrical mapping function and a rotary to wrap a 2D feature around a cylindrical part. | 4:44 |
| Get the Information that you need by customizing your Haas Control Home Screen | 11 | M/L | Intermediate | This TOD describes how you can customize the timers and counters section of the haas control to display whatever information you woul like. | 3:53 |
| Save Time on Your Haas Mill by Pre-Staging Tools! | 12 | M | Intermediate | This TOD shows you how to save some valuble cycle time with a side-mount-tool-changer by pre-staging tools | 3:24 |
| Let Your Haas CNC bring the tools to you! The Second Home Option. | 13 | M | Intermediate | This TOD explains how to set up the second home option on the Haas control. | 2:50 |
| Programming Shortcuts on Your Haas Control | 14 | M/L | Intermediate | This TOD shows you a few programming tricks that save valuble time and extra key strokes when writing by hand. | 2:33 |
| Set Up Live Tools Properly on Your Haas Lathe | 15 | L | Intermediate | Learn how to set up BOT as well as Axial and Radial Live Tools on your Haas Lathe. Extensive video! | 13:18 |
| Machining magic: Rotate Your Part with G68 | 16 | M | Intermediate | This TOD shows how to use the coordinate rotation and scaling option to simplify part setups. | 4:11 |

| Advanced Level | | | | | |
|---|---|-----|----------|---|------|
| How to Command a Safe Tool Change Position to Avoid Fixtures and Rotaries | 1 | M | Advanced | This TOD shows you how to avoid costly errors by using alias codes to create a safe tool change position | 5:43 |
| Display the Axes You Want to See on Your Haas Control | 2 | L | Advanced | This TOD shows you how to customize the control screen on a lathe to display all the axes that you care to see. | 2:29 |
| Limiting Block Look-Ahead when using Macros | 3 | M/L | Advanced | This TOD explains the most import thing you need to know when using macros on the Haas control. | 5:15 |
| Easily Create Your Own Custom M and G Codes | 4 | M/L | Advanced | This TOD discusses how and why a machinist might want to use 'Alias' G and M codes to save themselves time while programming | 4:39 |
| Use Your Insert Drill as a Boring Bar! 2 operations from 1 Tool! | 5 | M/L | Advanced | This TOD shows a simple lathe trick that allows you to use an indexable drill as a drill and a boring bar saving set-up time and tools. | 2:00 |

| Tip of the Day Title | Sequence Number | Mill or Lathe | Skillset Level | Reason to view this Tip of the Day | Video Length |
|--|------------------------|----------------------|-----------------------|--|---------------------|
| Quickly Pick up a Work Offset and an Angle with Your Haas Probe | 6 | M | Advanced | This TOD shows you how to pick up a work offset and an angle with your Haas probe. This helps to save valuable time in a part setup. | 8:02 |
| Unwind Your Haas Rotary Back to Zero Quickly | 7 | M | Advanced | This TOD teaches you how to save valuable cycle time when machining with a rotary unit by utilizing the quick rewind feature. | 2:57 |
| Simplify 3+2 and 5-Axis Machining with DWO/TCPC | 8 | M | Advanced | This TOD shows how to use the the dynamic work offset and tool center point control feature to make 4 and 5-axis programs simple. | 4:57 |
| Setting Up a Right-Angle Head on Your Haas: G17, G18 and G19 Explained | 9 | M | Advanced | This TOD discusses how to drill and tap with a right angle heads on different planes and angles. | 13:55 |
| Machine Your Own Gifts on Your Haas! | 10 | M | Advanced | This TOD shows a fun christmas gift idea that you can make on your Haas Machine | 4:21 |