Foreword

This guide is intended to assist AUTHORIZED USERS in machining facilities equipped with Haas CNC machine tools.

**This document does not replace supervised training.**

The photographs (and supplemental videos) used in this guide are taken from a variety of vintages of the Haas Controller some may be from controllers newer than the one you are using, and some from an older version. One of the beauties of the Haas controller is that it has been a stable platform from its inception. If you know how to use one vintage controller you can use any of them with little or no additional training.

As indicated by the title this book is intended as a quick reference guide and is not a replacement for the operators manual or other training materials available from Haas Automation, available at:

Acknowledgements and Authors

This guide has been compiled over a period of several years of teaching. The guide has grown and evolved over these years and I hesitate greatly to claim authorship. The first revision of this document was distributed in the spring of 2006; it was 14 pages long and I was the sole author.

Since this first version there have been dozens of WPI students and staff who have contributed in one way or another. It would be impossible for me to list all of them here but I will try to list some of the most significant contributors below. If you are reading this and think your name should be on this list you are probably correct. Please contact me and I will make the correction.

Contributing Authors:

Wei Tian
James Loiselle
Adam Sears
David Ephraim
Aaron Cornelius
Kevin Arruda
Mikhail Tan
# Table of Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Foreword</td>
<td>iii</td>
</tr>
<tr>
<td>Acknowledgements and Authors</td>
<td>iv</td>
</tr>
<tr>
<td>Table of Contents</td>
<td>v</td>
</tr>
<tr>
<td>Safety</td>
<td>1</td>
</tr>
<tr>
<td>Operations Checklist</td>
<td>3</td>
</tr>
<tr>
<td>Startup Procedures</td>
<td>4</td>
</tr>
<tr>
<td>Program Selection</td>
<td>8</td>
</tr>
<tr>
<td>Machine Operation Basics</td>
<td>14</td>
</tr>
<tr>
<td>Setting Tool Offsets</td>
<td>25</td>
</tr>
<tr>
<td>Setting Work Offsets</td>
<td>41</td>
</tr>
<tr>
<td>Program Operation</td>
<td>66</td>
</tr>
<tr>
<td>Clean Work Area</td>
<td>71</td>
</tr>
</tbody>
</table>
Safety

The safe operation of any manufacturing facility is the responsibility of everyone involved from the CEO to the person sweeping the floors.

In 1968, the bloodiest year of the Vietnam War, more Americans were killed in industrial accidents than died in combat. In 1971, OSHA was founded.

The owners and managers of your facility are responsible for establishing safe operating procedures. They are responsible with providing all personnel and equipment with functional safety systems and appropriate personal protective equipment (PPE.) If they fail to do this they will be subject to substantial fines, penalties, and criminal prosecution.

*No judge has ever been able to award an injured worker a new hand or orphaned children a new mother or father.*
Before operating any piece of equipment or using any tool:

- Review the Operator’s Manual: particularly the section on safety
- Inspect the tool for obvious signs of misuse or damage
- Make sure you have someone within sightline to assist you
- When in doubt, ask for help

Whenever you are in a manufacturing facility:

- Follow all safety policies
- Pay attention to what is going on around you.
- Stop any activity you think is unsafe
- Clean up spills and debris immediately
- Always use the appropriate PPE
- Report damaged machines, guarding, or other unsafe situations to your supervisor

---

**Yale senior dies in machine shop accident**

*By Jason Hanna, CNN*

*April 13, 2011 9:25 p.m. EDT*

*(CNN) -- A Yale University senior was killed in an accident in a chemistry laboratory’s machine shop late Tuesday or early Wednesday, officials at the Connecticut school said.*

*Michele Dufault of Massachusetts was working on a project in the Sterling Chemistry Laboratory's machine shop when*
# Operations Checklist

<table>
<thead>
<tr>
<th>Operator</th>
<th>ID#</th>
</tr>
</thead>
<tbody>
<tr>
<td>Signature</td>
<td>Machine Tool</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Start Time</th>
<th>Date</th>
</tr>
</thead>
<tbody>
<tr>
<td>End Time</td>
<td>Date</td>
</tr>
</tbody>
</table>

All users must complete this checklist anytime they operate a machine tool. All of the steps must be repeated under any of the following circumstances:

1. the program has been modified,
2. the user has altered the setup in the machine tool since the program was last run,
3. the user has been away from the machine for someone else to have altered the setup.

**Notes**

☐ Set Tool Offsets
This tells the machine about the tools that are installed and must be done before a program can be run.

☐ Set Work Offsets
This tells the machine about the location of the workpiece and must be done before a program can be run.

☐ Simulate in CAM Software
This ensures that you know what the program is intended to do.

☐ Simulate on Controller
This ensures that you know what the machine tool thinks the program should do.

☐ Reduce Rapid Rate
This gives you a chance to stop the machine tool before something bad happens if you have made a mistake in the previous steps.

☐ Check Distance to Go for Each Tool
This is your last chance to catch setup and possible programming mistakes. Almost every crash that happens can be stopped if we pay attention and complete this step.

☐ Check here if you are completing a non-standard operation that precludes completing one or more of the steps below. The current lab monitor must initial and approve.
Startup Procedures

Before you start the machine tool it is a good idea to check a few things:

1. Check the coolant level
2. Check the way oil level
3. Check that the power switch at the rear of the machine tool is turned on (shown below)

After this, follow these instructions to turn the machine on and prepare it for use.

Contents

- Power On .......................................................... 5
- Clear Alarms ...................................................... 6
- Power Up/Restart ............................................... 7
Power On

1. Press the green button (POWER ON) located on the upper left side of the controller

*If the machine fails to start it may indicate that it has been locked out, or that a room level emergency stop has been activated.*

*If this has happened, check with your supervisor before continuing.*

2. Reset the EMERGENCY STOP

*Twist it to the right and it will pop out (Pulling the button out may cause it to fail prematurely!)*
Clear Alarms

When you power on the machine tool there will be at least one alarm message (two if the emergency stop was activated before the machine was shut down)

1. Press the **ALARM** button to enter the alarms display, and check the displayed alarms to make sure that there’s nothing critical. Use the **ARROW KEYS** to scroll between them.

2. Press the **RESET** button for each alarm displayed to clear it.

3. On machines with servo auto doors, you will need to open and close the door to check the sensor.
Power Up/Restart

Because of the construction of most CNC machine tools it is necessary to reference (home) all of the axes before you run a program.

The process of referencing the axes involves the controller moving each axis over a limit switch at one end of its travel in turn.

1. Verify there are no obstructions blocking the travel of the axes from their present location to the home position

*If tall fixtures or long tools installed in the machine tool will cause a collision during the POWER UP/RESTART operation, STOP and complete Single Axis Return. See your Haas manual for details.*

2. Press the **POWER UP/RESTART** button and wait for the machine to complete operation
Program Selection

It’s important to make sure the machine has the right program selected. This section will cover basic methods of selecting and loading programs onto the controller.

Contents

Select Program From Memory ................................................. 9
Load from Floppy Disk ........................................................... 10
Load from USB ................................................................. 11
Load with RS232 ............................................................... 12
Drip Feed from Hard Drive or USB ........................................ 13

© Engineering Media Productions, LLC 2015-2016
Select Program From Memory

1. Press LIST PROGRAM

2. On newer controls, press WRITE/ENTER to enter the MEMORY tab. On older controls, this won’t be necessary

3. Use the arrow keys to highlight your program

4. Or type in the program number

eg “O02021”

5. Press SELECT PROGRAM to select that program
Load from Floppy Disk

1. Insert the floppy disk with the program into the controller
2. Press **EDIT**, then **F1** to activate the menus
3. Use the **ARROW KEYS** and **WRITE/ENTER** to select the I/O tab
4. Select Disk Directory and press **WRITE/ENTER**
5. Use the **ARROW KEYS** or **JOG HANDLE** to highlight the desired file, then press **WRITE/ENTER** to start loading the program

*New files are listed at the bottom*

6. If there’s already a program with that number, you’ll be prompted to overwrite it. Press **Y** for Yes and **N** for No
Load from USB

1. Insert the USB drive with the program into the controller
2. Press **LIST PROGRAM**
3. Use the cursor arrows to select USB DEVICE and press **WRITE/ENTER** to enter the tab. If the list of devices isn’t active, press **CANCEL**
4. Locate the program you are trying to load and press **WRITE/ENTER**
5. Press **F2**
6. Use the cursor arrows to select MEMORY from the list
7. If there’s already a program with that number, you’ll be prompted to overwrite it. Press **Y** for Yes and **N** for No
Load with RS232

1. Select **LIST PROG** to view listed programs
2. Press **END** to make sure **ALL** is highlighted
3. Press **RECV RS232**. The controller is now waiting to receive data
4. Send the program from your DNC software. It will automatically be loaded and selected on the controller

*Note that this option requires additional configuration to run correctly.*
Drip Feed from Hard Drive or USB

1. Load the program onto a USB drive or the machine’s hard drive

   Only newer model machines will have built-in hard drives

2. Press LIST PROGRAM

3. Use the cursor arrows to select USB DEVICE or HARD DRIVE and press WRITE/ENTER to enter the tab. If the list of devices isn’t active, press CANCEL

4. Highlight the desired program, and press SELECT PROGRAM
Machine Operation Basics

In order to operate the machine tool and to complete machine tool setup steps there are a few basic operations you should know how to do. In this section we show you how to:

Contents

Operate Tool Changer .......................................................... 15
Load Tools .................................................................. 16
Remove Tools ................................................................. 17
Handle Jog ................................................................ 18
Start and Stop the Spindle ............................................... 19
Fixturing .................................................................. 20
Fasteners and Machine Straps ......................................... 21
Using a Vise ................................................................. 22
Installing a Vise ............................................................ 23
Tramming a Vise ............................................................ 24
Operate Tool Changer

1. Check to ensure the tool changer can rotate without any of the tools impacting fixturing or parts (Handle Jog the table or turret as necessary to clear the tool change area)

2. Select MDI mode

3. Type “T” followed by the desired Tool Number. Do not press WRITE/ENTER

   **Example: For Tool Number 10, enter “T10”**

4. Press ATC FW or ATC REV
Load Tools

1. **Tool Change** to the desired tool number

2. Ensure **MDI** mode is selected

3. With your left hand hold the tool in the spindle with the slots lined up with the spindle dogs pressing up firmly

4. With your right hand press and hold the **TOOL RELEASE** button until the tool holder moves up into the spindle

*Look at the controller and verify that it’s in MDI mode before pressing this button*

5. Release the **TOOL RELEASE** button

*Ensure the tool is held securely in the spindle before releasing it from your left hand*
Remove Tools

1. **Tool Change** to the desired tool number.
2. Ensure **MDI** mode is selected
3. With your left hand hold the tool firmly
4. With your right hand press and hold the **TOOL RELEASE** button until the tool holder move down out of the spindle

*Look at the controller and verify that it’s in MDI mode before pressing this button*

5. Release the **TOOL RELEASE** button
Handle Jog

Handle Jog Mode is used to manually position the axis of the machine tool. You can use either the Jog Handle or the Jog Buttons

1. Select the HANDLE JOG mode
2. Select an Increment / Speed

*The top number indicates the distance the selected axis will move for each “click” of the Jog Handle, here in inches.*

3. Select the axis to jog (X, Y, Z) with the Jog Buttons
4. Either turn the Jog Handle or press and hold the appropriate Jog Button to handle jog

*The bottom number indicates the speed the axis will move (here in inches/min) if the Jog Buttons are held in, or JOG LOCK is used*
**Start and Stop the Spindle**

1. Go into MDI mode

2. Type in the spindle speed you want. Don’t press **WRITE/ENTER**

   **With the door open the machine will cap spindle speed at 750 RPM**

3. Press **CW** in the overrides panel. This will start the spindle at the entered speed

4. After the spindle has been started this way and the speed is set, you can start it again just by pressing **CW**

5. To stop the spindle, press **STOP** or **RESET**.
Fixturing

All fixtures must serve the following two purposes in CNC milling:

1. Locate the workpiece within the machine tool’s work envelope
2. Counteracting machining forces applied during the machining operations.

A third function that is common to many fixtures, especially in a production environment, is:

3. Relocating subsequent workpieces at the same location.

*Entire text books have been written on the topic of fixturing.*
*Following are some examples of common fixturing methods and uses*

<table>
<thead>
<tr>
<th>Fixture Types</th>
<th>Common Uses:</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fasteners and machine straps</td>
<td>Large, awkwardly-shaped workpieces</td>
</tr>
<tr>
<td>Vises</td>
<td>Rectangular prismatic workpieces</td>
</tr>
<tr>
<td>Vises with soft jaws</td>
<td>Non-rectangular prismatic workpieces</td>
</tr>
<tr>
<td>Vises with V-blocks</td>
<td>Round workpieces</td>
</tr>
<tr>
<td>Collet fixtures</td>
<td>Round workpieces, automatic clamping</td>
</tr>
</tbody>
</table>
Fasteners and Machine Straps

1. Slide a t-nut into the slot
2. Turn the threaded rod into the t-nut
3. Using a wedge to hold up one end of the strap place the other end on the workpiece
4. Thread a nut onto the end of the rod
5. Tighten with a wrench

Nuts should rotate freely along the whole thread

Straps should angle slightly down toward the workpiece
Using a Vise

*Ensure that the vice is clean and fastened securely*

1. Secure the vice to the table
2. Make sure the vise is properly trammed
3. Use parallels if necessary (make sure the workpiece sits parallel to the bottom of the vise)
4. Make sure the vise will NOT be cut unintentionally (check the maximum depth of operations)
5. Tighten the vice using the handle
6. Make sure the workpiece is secured to the vice

*DO NOT LEAVE THE VICE HANDLE IN THE MACHINE DURING THE OPERATION*
Installing a Vise

1. Make sure the bottom of the vise and the table where it is to be mounted are both clean. If necessary, stone the surfaces to eliminate burrs.

2. Put the vise on the table in the rough position that you want it.

3. Slide T-nuts under the bolt holes in the vise.

4. Spray bolts to go in the holes with anti-seize lubricant, and then install them.

   *If you are using socket head screws, be sure you use several stacked washers under them*

5. Get one bolt snug, and leave the other loose.

6. **Tram in the vise**
Tramming a Vise

The process of tramming a vice is to align the X and Y axes of the machine tool with the X and Y axes of the vice.

1. Attach a dial indicator to the spindle

2. Using Handle Jog position the indicator and the machine tool’s tables so that the indicator needle is in contact with the back vise jaw

   *As you contact the vise jaw the indicator needle should move through half of its full travel*

3. Jog the needle slowly across the jaw observing the motion of the needle

   *A slight back and forth motion of the needle indicates that the vise is installed square with the table*

4. If the needle motion trends in one direction or the other, loosen the nuts holding the vise down slightly and use a soft faced mallet or hammer to tap the vise body until it is square

   *Remember to tighten the nuts when you are finished and repeat step 3 to double check*
Setting Tool Offsets

Tool offsets tell the machine how to compensate for the size of the tool, either in length or in diameter. All offsets are set to some reference surface. It is very important that that surface be the same for all tools. There are two primary approaches.

1. You can manually set the tools using a reference surface and gauge that you select.

2. You can use the machine probe to set offsets. In this case the reference surface is the spindle gage line.

Contents

Set Tool Offsets Using a Reference Gauge and Surface 26
Choosing a Reference Gauge and Surface 27
Positioning the Tool Using a Reference Gauge 28
Set the Tool Offset 29
Set Tool Diameter Offsets 30
Adjusting Diameter Offsets 31
Set Tool Offsets Using the Probe 32
When you manually choose your offset, there are three main steps.

1. Select a reference gauge and surface
2. Position the tool relative to the reference surface using the gauge
3. Set the tool offset
Choosing a Reference Gauge and Surface

Reference gauges must be dimensionally accurate, consistent, and easy to handle. There are three main choices for gauges: cigarette paper, gauge blocks, dowel pins.

Dowel pins are optimal as gauges. They have the following advantages:

1. Greater dimensional accuracy and safety than paper
2. Lower cost and longer life than gauge blocks

Reference surfaces need to be flat, durable, large enough to gauge off of, and ideally fixed. Common surfaces include: top of the part, back of the vise/fixture, machine table. All have advantages and disadvantages.

<table>
<thead>
<tr>
<th>Reference Surface</th>
<th>Advantages</th>
</tr>
</thead>
<tbody>
<tr>
<td>Top of the part</td>
<td>No need to set Z work offset, good for fast setups</td>
</tr>
<tr>
<td>Back of the vise</td>
<td>Consistent between parts. Flatter and more stable than the part top</td>
</tr>
<tr>
<td>Machine table</td>
<td>Consistent between setups. Flattest and most consistent surface</td>
</tr>
</tbody>
</table>

Once you’ve chosen your references, use them to set the tool offset.
Positioning the Tool Using a Reference Gauge

1. **Handle jog** the tool towards the reference surface so the gauge does **NOT** fit

2. Jog the tool away in 0.001” increment, one click at a time, until the gauge just fits

   *Don’t jog down while the gauge is in place.*

3. Remove your gauge, jog down one step, and switch to 0.0001” increment. Repeat the process until the gauge fits.

4. With your tool accurately positioned, next **set your offset**
Set the Tool Offset

1. Press the **OFFSET** button until the Tool Offset screen is displayed.

2. Highlight the Length Geometry offset for the tool you’re setting

3. Press **TOOL OFFSET MEASUR**

   *This sets the offset at the machine’s current location. Now accommodate for the thickness of your gauge.*

4. With the offset still highlighted, type a negative sign, and then the thickness of your gauge.

5. Press **WRITE/ENTER** to subtract from the offset.
Set Tool Diameter Offsets

The tool diameter offsets are used for radial cutter compensation, which adjusts for tool size and deflection on the control. The tool will be moved further in and out from the programmed path.

1. Look at your setup sheet to determine which tools use cutter comp, and what the default cutter comp values are.

Haas controls by default use the tool diameter for cutter comp. Some CAM software, like shown here, may output the radius of the tool instead, so we use twice the number shown.

2. Press the OFFSET button until the Tool Offset screen is displayed.

3. Highlight the Length Geometry offset for the tool you’re setting

4. Type in the default diameter of the tool, and press F1.

5. After you run a part and measure the feature, adjust the diameter offset to bring the part in tolerance.
Adjusting Diameter Offsets

After the initial part has been run, you should measure the important features and adjust the tool offsets to ensure that they are to spec.

1. Look at the diagrams to determine which situation applies.

2. Determine how far out of spec your feature is.

3. Apply the appropriate adjustment to the tool diameter offset, based on how far off your part is.
   - For outside cuts, change the offset by the difference in size. Positive changes will make the cut web bigger.
   - For inside cuts, change the offset by the difference in size. Positive changes will make the cut pocket smaller.
   - For one-sided cuts, change the offset by twice the difference in size. Positive changes will make the cut web bigger.
Set Tool Offsets Using the Probe

The tool setting probe can be used to automatically measure tool lengths, setting the offsets relative to the spindle gage line. Tool probing commands can be run through Visual Quick Code, a conversational programming interface.

During the probing of the tool length, the machine tool will usually drive the spindle in the Z-direction so the tip of the tool approaches and eventually comes into contact with the top surface of the stylus.

VQC provides the following tool setting cycles.

- Accessing VQC for Tool Offsets 33
- Auto Length Only 34
- Manual Length Only 35
- Automatic Length Rotating 36
- Automatic Length and Diameter 37
- Auto Length Only Sequential Tools 38
- Auto Length Only Random Tools 39
- Tool Breakage 40
Accessing VQC for Tool Offsets

Older Haas Controls

On older controls, VQC is accessed through the EDIT display.

1. Press EDIT, then F1, and use the cursors to highlight the HELP tab
2. Highlight “VISUAL QUICK CODE” and press WRITE/ENTER
3. Select “TOOL SETTING ” and press WRITE/ENTER

You will see the V QC tool setting page.

New Model Controls

On newer generation controls, VQC has been moved to the conversational programming display. To access it:

4. Press MDI, then PRGRM CONVRS
5. Scroll over to the VQC tab and press WRITE/ENTER
6. Select “TOOL SETTING ” and press WRITE/ENTER

You will see the V QC tool setting page.
Auto Length Only measures the LENGTH of one tool. It is only suitable for tools with cutting edges at the center, like drills, taps, center-cutting end mills, etc.

1. **On the screen with all the sub-routines for tool setting**, select Auto Length Only and press **WRITE/ENTER**

2. Type in the tool number as prompted and press **WRITE/ENTER**

3. Output the code to MDI, either by selecting the option from the menu or pressing 3

4. Go into MDI

5. Press **CYCLE START** to execute the measurement
Manual Length Only

Manual Length Only measures the length of a manually positioned tool.

The operator manually jogs the tool so its tip is slightly above the top of the stylus, and the cutting edges on the tool needs to be aligned with the stylus on the tool setting probe.

1. HANDLE JOG the tool towards the tool setting probe. Position the cutting edge to be probed 0.4” above the stylus

2. On the screen with all the sub-routines for tool setting, select Manual Length Only and press WRITE/ENTER

3. Enter in the tool number as prompted

4. Use cursors to select MDI, press WRITE/ENTER and press CYCLE START to execute the program
Automatic Length Rotating

Automatic Length Rotating measures the LENGTH of a non-center-cutting tool, such as a facemill, boring bar, or large endmill. It offsets the tool location from the probe and spins the tool backwards to ensure that touches the cutting edge of the tool.

1. Roughly measure the tool diameter and length from the spindle

**Remember that you want the diameter of the tool's lowest cutting point**

2. **On the screen with all the sub-routines for tool setting**, select Auto Length Rotating and press WRITE/ENTER

3. Type in the tool number, length and diameter as prompted, and press WRITE/ENTER

4. Output the code to MDI, either by selecting the option from the menu or pressing 3

5. Go into MDI

6. Press **CYCLE START** to execute the measurement

*Set Tool Offsets*
Automatic Length and Diameter

Automatic Length and Diameter sets the length offset and diameter offset of a tool. This is only necessary when your program uses radial cutter compensation

1. Roughly measure the tool diameter and length from the base of the spindle

   Remember that you want the diameter of the outside cutting edge of the tool

2. On the screen with all the sub-routines for tool setting, select Auto Length and Diameter and press WRITE/ENTER

3. Type in the tool number, length and diameter as prompted, and press WRITE/ENTER

4. Output the code to MDI, either by selecting the option from the menu or pressing 3

5. Go into MDI

6. Press CYCLE START to execute the measurement

Set Tool Offsets
Auto Length Only SequentialTools

Auto Length Only Sequential measures the LENGTH of a sequential set of tools in order.

1. **On the screen with all the sub-routines for tool setting**, select Auto Length Only Sequential Tools and press WRITE/ENTER

2. Type in the tools number of the first tool and last tool to be measured as screen prompts and press WRITE/ENTER

**e.g. tools 1-5**

3. Output the code to MDI, either by selecting the option from the menu or pressing 3

4. Go into MDI

5. Press **CYCLE START** to execute the measurement
Auto Length Only Random Tools

Auto Length Only Random measures the LENGTH of a set of non-sequential tools, up to twelve at a time.

1. **On the screen with all the sub-routines for tool setting**, select the Auto Length Only Random Tools option and press **WRITE/ENTER**

2. Type in the first tool number that needs to be probed and press **WRITE/ENTER**

3. Repeat this for the rest of your tools

4. If you are probing less than twelve tools, press **WRITE/ENTER** without an entry for the rest of them and it will skip them

*For example: to probe 4, 6, and 10, enter 4, 6, 10, pressing WRITE/ENTER between each, and then WRITE/ENTER with no input for the rest of the spots*

5. Output the code to MDI, either by selecting the option from the menu or pressing **3**

6. Go into MDI

7. Press **CYCLE START** to execute the measurement
Tool Breakage

Tool Breakage probes the tool length and checks how much it’s changed to see if it’s broken. If the change in length exceeds a set tolerance, it throws an alarm.

1. **On the screen with all the sub-routines for tool setting**, select Tool Breakage and press **WRITE/ENTER**

2. Type in the tool number as prompted and press **WRITE/ENTER**

3. Type in the allowed length tolerance and press **WRITE/ENTER**

4. Output the code to MDI, either by selecting the option from the menu or pressing 3

5. Go into MDI

6. Press **CYCLE START** to execute the measurement
Setting Work Offsets

Setting Work Offsets

On a mill, all parameters in the Work Zero Offset need to be defined for the machining operations to perform correctly. On a 3 axis mill, X, Y, and Z offsets all need to be set.

Contents

Setting XY Work Offset at the Current Location 42
Setting Z Offset 43
Coaxial Indicator 44
Position the Coaxial Indicator 45
Find the Center with a Coaxial Indicator 46
Assembling a Dial Indicator 47
Find the Center with the Dial Indicator 48
Set X and Y with a Mechanical Edge Finder 49
Compensating for the Edge Finder’s Radius 50
Set Z Work Offset on a Mill with Gauge or Indicator 51
Set Work Offset on a Mill with a Gauge 52
Set Work Offset on a Mill with an Indicator 53
Set Work Offset on a Mill with the Spindle Probe 54
Accessing VQC for Work Offsets 55
Setting XY Work Offset at the Current Location

1. Press the OFFSET button until the WORK ZERO OFFSET screen is displayed
2. Highlight the offset you need to set
3. Press PART ZERO SET to set the offset in the current location. Repeat for any other offsets

This will set the selected offset at the current location of the machine. If you used an edge finder to set your offset, remember to compensate for half the diameter of the edge finder.
Setting Z Offset
The Z work offset is the distance between the surface where the tools were set and the Z origin of the part. Once you’ve measured it, enter it by:

1. Press **OFFSET** until the Work Offset page is shown

2. Highlight the Z offset you want to set

3. Enter the measurement, but don’t press **WRITE/ENTER**

4. Hit **F1** to set the offset

**WRITE/ENTER** would add the entered number to whatever is in the offset. **F1** will overwrite it to the correct value
Coaxial Indicator

Coaxial indicators can be used to find the center of either a bore or a round boss, similar to a dial indicator. However, they are generally easier to use, as the readout is always facing the user even as the feeler spins. Follow these steps to assemble the indicator:

1. Install the stop arm to the indicator
2. Pick the appropriate feeler. Bores should use the straight feelers, bosses should use the bent ones
3. Hand-tighten the feeler into the indicator arm
4. Assemble the co-axial indicator to a tool holder and install it to the machine tool

*The shank goes into a proper size collet (3/8” dia.)*

5. Attach a magnetic base to the table to hold the stop arm so the indicator can remain stationary and face the operator
Position the Coaxial Indicator

1. **Handle Jog** the feeler near the feature to be probed, try to roughly align the center of the spindle with that of the bore/boss

2. Gently push or pull the feeler while jogging down so that the feeler will touch the probing surface upon release

3. Touch the magnetic base with the stop arm so the indicator remains stationary while the spindle turns

4. Gently turn the spindle **BY HAND** for a whole revolution to make sure the feeler moves freely
Find the Center with a Coaxial Indicator

1. Start the spindle at 10-20 RPM
2. Observe the positions of the indicator dial at the four points
   - Two blue points – two utmost positions along the X axis
   - Two red points – two utmost positions along the Y axis
3. Handle Jog one axis to minimize the indicator movement
   - Use 0.001” increment
   - Jog only in the XY directions, not Z
4. Jog the other axis so the indicator movement is minimized
5. Switch to 0.0001” increment and minimize the indicator movement again
6. Without jogging any more, set the X and Y work offsets
Assembling a Dial Indicator

A dial indicator held in the spindle can be used to find the center of a round part, much like a coaxial indicator.

The dial indicator can be held several ways:

- A ¼” diameter post can be attached and held in a tool holder
- A magnetic base can be attached to the base of the spindle or a tool holder. This is generally more convenient

To install these:

1. Loosen the dovetail clamping bolt on the post or base
2. Slide it over the mating dovetail on the indicator. For this purpose you want the dovetail on the top
3. Tighten the dovetail clamping bolt
4. If you are using the post, install it in a tool holder
5. Put the tool in the spindle or the magnetic base on the spindle
Find the Center with the Dial Indicator

1. Handle jog the indicator roughly over the center of the part. Rotate the spindle by hand to visually inspect positioning.

   You can switch to a drill to roughly center the tool over the part, then change to the indicator without moving the XY.

2. Lower the indicator down so that it can touch the side of the part.

3. Bend the arm joints and the needle so that the needle point is in contact with the part.

4. Rotate the spindle by hand to ensure the indicator maintains contact with the part the whole way around.

5. Move X-position so that indicator reads the same number on both sides.

6. Once X-values are matching, move the position in Y to match that value as well.

7. Without jogging any more, set the X and Y work offsets.

Setting Work Offsets
© Engineering Media Productions, LLC 2015-2016
Set X and Y with a Mechanical Edge Finder

A mechanical edge finder is a spring loaded rotating tool used to find X and Y offsets.

As the rotating edge finder touches the surface to be found, friction causes the lower part of the edge finder to “pop out” to one side. This provides an accurate, repeatable measurement to the edge.

To use an edge finder:

1. Put the edge finder in a tool holder and install it in the machine tool
2. Determine which faces you need to locate

Look at your setup sheet. Where’s the origin?

3. Start the spindle at 750 rpm
4. Handle jog the edge finder near the “edge”
5. Slowly jog the edge finder in 0.01” increment towards the part until the edge finder tip “pops” off as shown
6. This will locate the edge to within 0.01”. If you need more accuracy, back off and repeat with a lower increment
7. Stop the spindle, and set your offset for the axis you just touched off
8. Compensate for the radius of the edge finder
Compensating for the Edge Finder’s Radius

1. Determine which compensation value you need based on the diagrams below and the side you just located.

2. Type in the value shown below with the offset indicated highlighted. Make sure to keep the negative sign, if shown.

\[ \text{Offset by } \frac{0.2''}{2} = 0.1'' \]

**You must be in the work offset display for this step**

3. Press WRITE/ENTER to add the compensation value.
Set Z Work Offset on a Mill with Gauge or Indicator

The Z work offset is the relative distance between the surface where the tools were set and the top of the part. If the tool offsets have been set using a reference gauge, the work offset can be set with either one of those previously set tools + gauge block or a dial indicator.

If you set your tools off the part and your work origin is in the same location, then your work offset is zero.

This diagram shows the Z work offset if the tools were set off the back of the vise and the work origin was on top of the part.
Set Work Offset on a Mill with a Gauge

1. **Set your tool offsets with a gauge**

2. Keep the last tool in the position you found with the gauge

3. Press **HANDLE JOG** and select Z axis

4. Press the **POSIT** button once. If you can’t see the operator coordinates then press **PAGE DOWN** until the operator coordinates appear

5. Press the **ORIGIN** button so the values for Z for the Operator’s Coordinate is set to 0.0000"

6. Use the gauge to touch the tool off at your work origin

7. **Set your Z offset to the value of your Z operator position**
Set Work Offset on a Mill with an Indicator

1. Assemble and install the indicator
2. Handle Jog the indicator to the common reference surface (the worktable in this document) until the stylus moves to zero
3. Press HANDLE JOG and select the Z axis
4. Press the POSIT button once. If you can’t see the operator coordinates then press PAGE DOWN until the operator coordinates appear.
5. Press the ORIGIN button so the values for Z for the Operator’s Coordinate is set to 0.0000”
6. Jog the indicator above the work Z zero and touch off so that it goes to the same needle position
7. Set your Z offset to the value of your Z operator position
Set Work Offset on a Mill with the Spindle Probe

The spindle probe allows the mill to measure the workpiece and set offsets automatically. During setup, the operator jogs the probe to the part and runs probing macros. Part probing commands can be run through [Visual Quick Code](#), a conversational programming interface.

There are too many probing macros to fit on one page in VQC. A list is below.

<table>
<thead>
<tr>
<th>Spindle Probe 1-9</th>
<th>Spindle Probe 10-18</th>
</tr>
</thead>
<tbody>
<tr>
<td>Probe Bore</td>
<td>Probe Boss</td>
</tr>
<tr>
<td>56</td>
<td>61</td>
</tr>
<tr>
<td>Probe X/Y Surface</td>
<td>Probe Pocket</td>
</tr>
<tr>
<td>57</td>
<td>62</td>
</tr>
<tr>
<td>Probe Z Surface</td>
<td>Probe the Center of a Web</td>
</tr>
<tr>
<td>58</td>
<td>63</td>
</tr>
<tr>
<td>Probe Vise Corner</td>
<td>Probe Internal Corner</td>
</tr>
<tr>
<td>59</td>
<td>64</td>
</tr>
<tr>
<td>Probe Center of Block</td>
<td>Probe External Corner</td>
</tr>
<tr>
<td>60</td>
<td>65</td>
</tr>
</tbody>
</table>
Accessing VQC for Work Offsets

Older Haas Controls

On older controls, VQC is accessed through the EDIT display.

1. Press **EDIT**, then **F1**, and use the cursors to highlight the HELP tab
2. Highlight “VISUAL QUICK CODE” and press **WRITE/ENTER**
3. Select SPINDLE PROBE 1-9 or 10-18 and press **WRITE/ENTER**

You will see the VQC tool setting page.

New Model Controls

On newer generation controls, VQC has been moved to the conversational programming display. To access it:

4. Press **MDI**, then **PRGRM CONVRS**
5. Scroll over to the VQC tab and press **WRITE/ENTER**
6. Select SPINDLE PROBE 1-9 or 10-18 and press **WRITE/ENTER**

You will see the VQC tool setting page.

Setting Work Offsets
Probe Bore

Probe Bore finds the center of a round or square internal pocket. You will manually position the probe tip within the bore, then run the probing macro.

1. **Tool change** to the probe, and **handle jog** it into the bore

2. **In VQC**, enter **SPINDLE PROBE 1-9**, and select **PROBE BORE**

3. Type in the Work Offset number (eg 54) as prompted and press **WRITE/ENTER**

4. Type in the rough bore diameter as prompted and press **WRITE/ENTER**

5. Output the code to MDI, either by selecting the option from the menu or pressing 3

6. Go into MDI

7. Press **CYCLE START** to execute the measurement
Probe X/Y Surface

There are four edge finding probing macros. They will move the probe in the direction indicated, and set the offset at the surface they touch.

1. Tool change to the probe, and handle jog into the place indicated for what you want to do

2. In VQC, enter SPINDLE PROBE 1-9, and select the appropriate edge probing operation

   Look at the diagrams to see which direction the operation will probe

3. Type in the Work Offset number (eg 54) as prompted and press WRITE/ENTER

4. Output the code to MDI, either by selecting the option from the menu or pressing 3

5. Go into MDI

6. Press CYCLE START to execute the measurement
Probe Z Surface

Probe Z Surface probes down and sets the Z offset.

1. **Tool change** to the probe, and **handle jog** to 0.4” above the surface you want to probe

2. **In VQC**, enter SPINDLE PROBE 1-9, and select Probe Z Surface

3. Type in the Work Offset number (eg 54) as prompted and press **WRITE/ENTER**

4. Output the code to MDI, either by selecting the option from the menu or pressing **3**

5. Go into MDI

6. Press **CYCLE START** to execute the measurement
Probe Vise Corner

Probe Vise Corner will set XY offsets at the back left of the part, and Z on the top. Note that it can’t probe any other corners.

1. **Tool change** to the probe, and **handle jog** to 0.4” above the back left corner of the part
2. **In VQC**, enter SPINDLE PROBE 1-9, and select Probe Vise Corner
3. Type in the Work Offset number (eg 54) as prompted and press **WRITE/ENTER**
4. Enter rough estimates for the X and Y widths of the parts
5. Enter the depth that the probe should plunge when probing X and Y. Remember that going down is a negative Z move
6. Output the code to MDI, either by selecting the option from the menu or pressing 3
7. Go into MDI
8. Press **CYCLE START** to execute the measurement
Probe Center of Block

The two Probe Center of Block macros will set the XY offset at the center of a rectangular block and the top in Z. In Start From Center you roughly center the probe over the part, while in Start From Corner you position the probe over the back left corner of the part.

1. **Tool change** to the probe, and **handle jog** to 0.4” above the back left corner or center of the part

2. **In VQC**, enter SPINDLE PROBE 1-9, and select Probe Center of Block Start From Corner or Start From Center

3. Enter the Work Offset (eg 54) and press WRITE/ENTER

4. Enter rough estimates for the X and Y widths of the parts

5. Enter the depth that the probe should plunge when probing X and Y as a negative Z move

6. Output the code to MDI, either by selecting the option from the menu or pressing 3

7. Go into MDI

8. Press **CYCLE START** to execute the measurement

---

**Setting Work Offsets**

© Engineering Media Productions, LLC 2015-2016

**Table of Contents**

Page 60
**Probe Boss**

Probe Boss finds the center of a round or square external boss. You will manually position the probe tip within the bore, then run the probing macro.

1. **Tool change** to the probe, and **handle jog** it above the center of the boss.
2. **In VQC**, enter **SPINDLE PROBE 10-18**, and select **PROBE BOSS**
3. Type in the Work Offset number (eg 54) as prompted and press **WRITE/ENTER**
4. Type in the rough boss diameter as prompted and press **WRITE/ENTER**
5. Enter the depth that the probe should plunge when probing X and Y as a negative Z move.
6. Output the code to MDI, either by selecting the option from the menu or pressing **3**
7. Go into MDI
8. Press **CYCLE START** to execute the measurement

**Setting Work Offsets**

© Engineering Media Productions, LLC 2015-2016
**Probe Pocket**

The two Probe Center of Block macros will set the X or Y offset at the midpoint of a rectangular pocket. Probe X Pocket sets the X offset, and Probe Y Pocket sets the Y offset.

1. **Tool change** to the probe, and **handle jog** it to the center of the pocket you want to probe

2. **In VQC**, enter SPINDLE PROBE 10-18, and select Probe Pocket X or Y

3. Enter the Work Offset number (eg 54) and press **WRITE/ENTER**

4. Enter a rough estimate for the width of the pocket in the direction you’re probing

5. Output the code to MDI, either by selecting the option from the menu or pressing **3**

6. Go into MDI

7. Press **CYCLE START** to execute the measurement
Probe the Center of a Web

The two Probe Web macros will set the X or Y offset at the midpoint of a rectangular web. Probe X Pocket sets the X offset, and Probe Y Pocket sets the Y offset.

1. **Tool change** to the probe, and **handle jog** it above the center of the web you want to probe

2. **In VQC**, enter SPINDLE PROBE 10-18, and select Probe Web X or Y

3. Enter the Work Offset number (eg 54) and press **WRITE/ENTER**

4. Enter a rough estimate for the width of the web in the direction you’re probing

5. Enter the depth that the probe should plunge when probing X and Y as a negative Z move

6. Output the code to MDI either by selecting the option from the menu or pressing 3

7. Go into MDI

8. Press **CYCLE START** to execute the measurement
Probe Internal Corner

Probe Internal Corner will set the X and Y offsets at the corner inside a pocket. You need to position the probe inside the corner, and tell it which corner it’s probing.

1. **Tool change** to the probe, and **handle jog** within the corner as shown

2. **In VQC**, enter SPINDLE PROBE 1-9, and select Probe Internal Corner

3. Enter the Work Offset number (eg 54) and press **WRITE/ENTER**

4. Enter rough estimates for the X and Y widths you wish to probe

5. Select which corner number you want to probe, referring to the diagram

6. Output the code to MDI, either by selecting the option from the menu or pressing **3**

7. Go into MDI

8. Press **CYCLE START** to execute the measurement

---

**Setting Work Offsets**

© Engineering Media Productions, LLC 2015-2016

**Table of Contents**

Page 64
Probe External Corner

Probe External Corner will set the X and Y offsets at the outside corner of a part. You need to position the probe outside the corner, and tell it which corner it’s probing.

1. **Tool change** to the probe, and **handle jog** outside the corner as shown.

2. **In VQC**, enter SPINDLE PROBE 1-9, and select Probe External Corner

3. Enter the Work Offset number (eg 54) and press **WRITE/ENTER**

4. Enter rough estimates for the X and Y widths you wish to probe

5. Select which corner number you want to probe, referring to the diagram

6. Output the code to MDI, either by selecting the option from the menu or pressing 3

7. Go into MDI

8. Press **CYCLE START** to execute the measurement

---

**Setting Work Offsets**

© Engineering Media Productions, LLC 2015-2016  

Table of Contents
Program Operation

When you are running the program, there are some basic steps you can take to ensure that your program won’t fail. This section describes how to perform some of the steps on the operator checklist.

Contents

Simulate in CAM .......................................................... 67
Simulate on the Controller ............................................. 68
Reduce the Rapid Rate .................................................. 69
Check the Distance to Go ............................................ 70
Simulate in CAM

The instructions given here are for ESPRIT. However, the process will be similar for any CAM software

1. Open your CAM file.
2. Enter the simulation mode
3. Enter Simulation Parameters and enable collision detection
4. Run the simulation and watch for crashes
Simulate on the Controller

1. Ensure the desired program is selected. Press the MEM button
2. Press SETNG GRAPH twice above the arrow keys to enter the graphics display
3. Press CYCLE START to start the simulation
4. Look at the display to check if there are alarm messages or whether the tool path is as expected
5. If the tool path isn’t on the screen, press F2. Use the arrow keys to move the window around. PAGE UP and PAGE DOWN will zoom the display in and out. WRITE/ENTER will confirm the movement, then press CYCLE START to run the simulation again

Pay attention to the small preview display below the main display. You can see where the window will end up
Reduce the Rapid Rate

1. Determine what rapid rate you’re comfortable with
2. Hit the appropriate rapid rate override in the override section
3. Check the display to make sure that the rapid rate has actually been reduced
Check the Distance to Go

1. In the CURRENT COMMANDS display, pull up the DISTANCE TO GO coordinate display:
   a. On older controllers, press the DOWN ARROW until DIST TO GO appears in the top right of the screen
   b. On newer controllers, press PAGE DOWN until the coordinates display appears

2. Run your program. As each tool approaches the part, press feedhold about 0.5-1” away from the part

3. Check which Z position it’s going to by reading the program

4. Compare the Z distance in the DIST TO GO coordinate to how far the machine physically has to go

   This is just an eyeball measurement. In the picture image with a 3/8” tool, anywhere between 0.375” and 0.625” is probably okay.

5. If the DIST TO GO matches up with the physical distance, continue the program. Otherwise, stop and fix your offsets

   Also take this opportunity to realign the coolant nozzles to hit the tool, if necessary.
Clean Work Area

It is important to clean the work area every time you use the machine.

1. Remove your part safely and brush chips off the fixture
2. Use the coolant spray-down hose to wash chips down to the bed of the machine tool
3. Use a chip rake to pull the chips into the chip bin

*Give the coolant time to drain from the chips*

4. Use a broom to sweep the area around the machine tool
5. Wipe down the outside of the machine tool and the controller as necessary