



Haas Factory Outlet

A Division of Productivity Inc

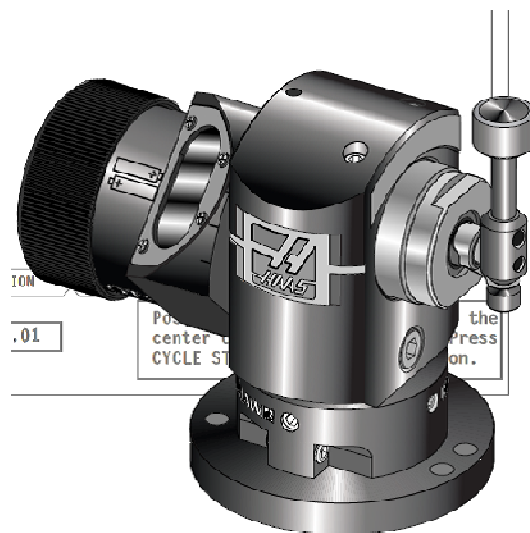
HFOMN/Haas CNC Series

NG2 Next Generation Control



VPS (Visual Programming System)

Probe Training Manual



This Manual is the Property of Productivity Inc

The document may not be reproduced without the express written permission of Productivity Inc.

The content must not be altered, nor may the Productivity Inc name be removed from the materials.

This material is to be used as a guide to operation of the machine tool. The Operator is responsible for following Safety Procedures as outlined by their instructor or manufacturer's specifications.

NOTE: Downloading and/or other use of this manual does not certify completion of the Training Course. This manual is for reference only.

To obtain permission, please contact trainingmn@productivity.com.


VISUAL PROGRAMMING SYSTEM – NG2 Control

Probe Training Manual Table of Contents

INTRODUCTION TO VPS PROBE TRAINING.....	3
ELEMENTS OF THE MACHINE PROBE SYSTEM	4
OMI – OPTICAL MACHINE INTERFACE	4
OMP – OPTICAL MACHINE PROBE	5
OTS – OPTICAL TOOL SETTER	6
BATTERIES.....	8
PROBE TEST	10
PROBE PROGRAMS.....	11
MACRO PROGRAM LIST.....	12
PROBE ALIGNMENT.....	14
VPS – VISUAL PROGRAMMING SYSTEM USE WITH PROBES.....	15
PROBE CALIBRATION	15
SETTING OF TOOL LENGTH OFFSETS USING VPS	22
SETTING WORK OFFSETS USING VPS	26
PROBING VISE CORNER.....	28
PROBING TOOL AND WORK OFFSETS THRU THE OFFSETS PAGE	30



For more information on Additional Training Opportunities
or our Classroom Schedule
Contact the Productivity Inc Applications Department in Minneapolis:

 763.476.8600

Visit us on the Web: www.productivity.com

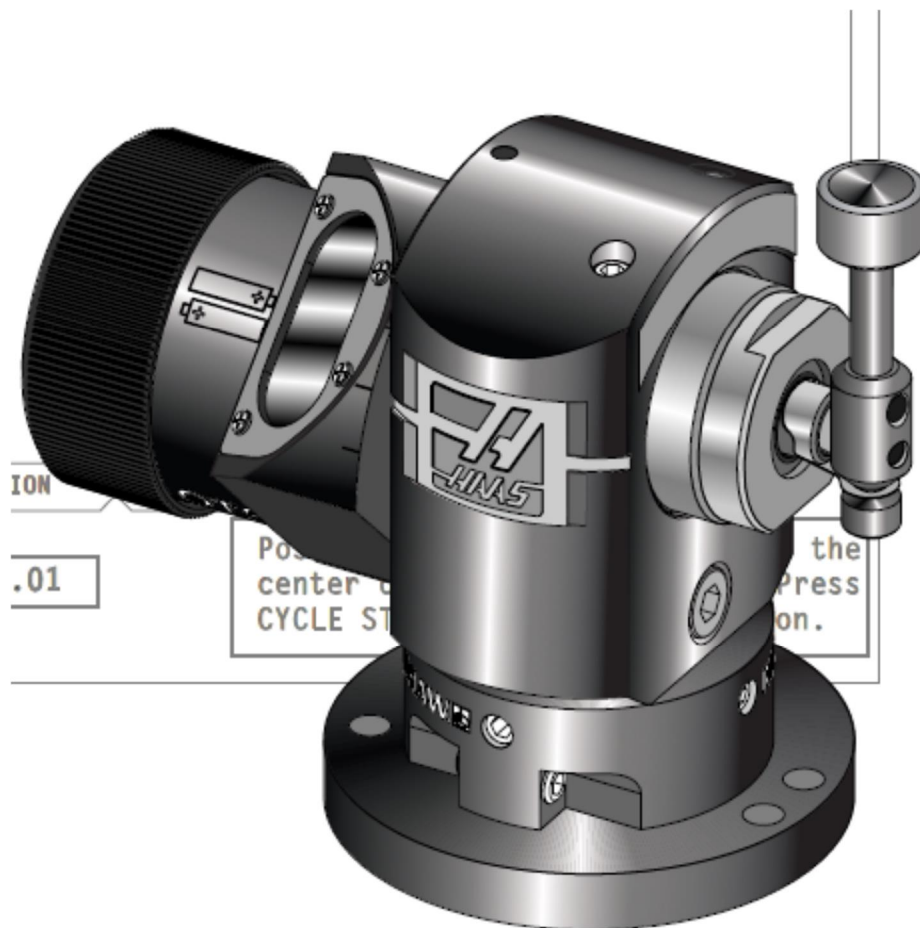
Click on the **Training Tab**

 trainingmn@productivity.com

Introduction to VPS Probe Training

This class is designed to introduce new users of machine tool probes how to set Tool Length Offsets and Work Offsets using **VPS (Visual Programming System)** software. The course does not cover how probe systems may be used in cycle gauging, inspection or automatic offset updates. This course is designed just to teach how VPS may be used to simplify and speed set up of the Haas vertical machining center.

This is achieved by a predictive process control of tool setting and part setting. With probe systems the length of a tool from spindle gauge-line may be easily determined. Accurate diameter of spinning tools may also be set by rotating the tool. Various spindle probe routines determine different types of datum used to establish work co-ordinate systems. Probe routines may be also run to detect broken tools.

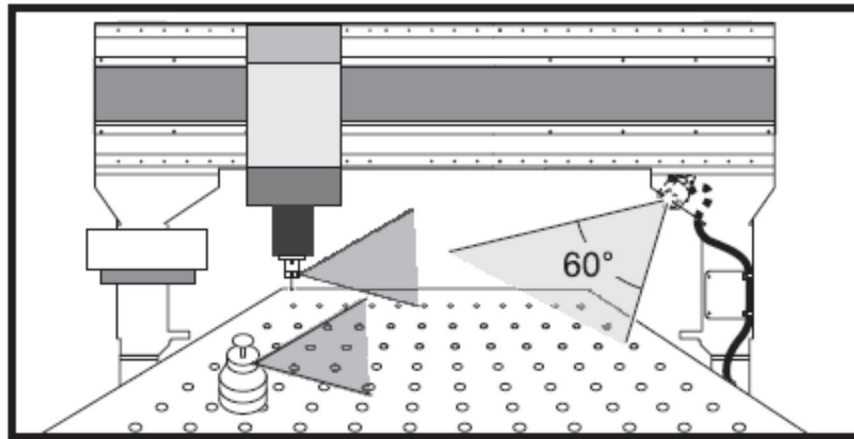


Elements of the Machine Probe System

The machine probe system consist of three units: the OMI (Optical Machine Interface), the OTS (Optical Tool Setting) tool probe and the OMP (Optical Machine Probe) work probe.

OMI – Optical Machine Interface

The **OMI (optical machine interface)** is tied in directly to the machine CNC control. The OMI or machine interface communicates with the other two probes by modulated infrared light. It acts as an optical receiver which sends and receives signals from the other two probes directly to the CNC machine control. Below shows the optical Machine Interface on the right which can communicate with either the table or the spindle probe.



The table probe, spindle probe and the machine interface use LEDs and photo diodes to communicate back and forth with each other. In each of the devices there are specific LEDs which indicate probe status, battery status and any error conditions. The machine interface power supply comes from the machine control while the table and spindle probes require batteries.

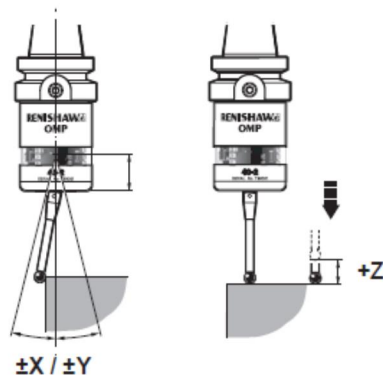
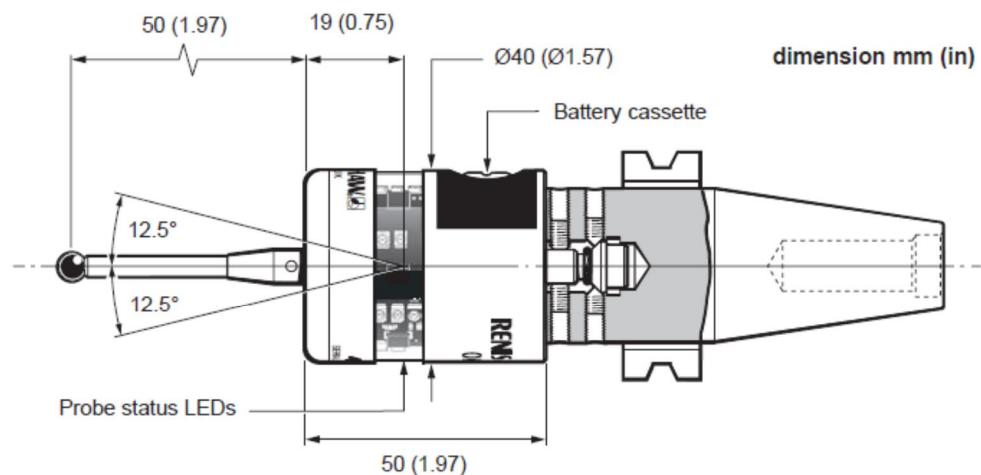
OMP – Optical Machine Probe

The **OMP (optical machine probe) or work probe** is a 3D touch triggered inspection probe loaded in a tool holder. It is used to set up work offsets and inspection of parts in a Haas machining center. It will sense direction hits in 5 different directions +/-X, +/-Y, and +Z. Because of the sensitive electronics it is characterized as a **heavy tool** in the **Tool Pot Position page** so that it will only index at 25% rapid. Normally it is designated as the last tool in the side mount carousel or larger. With a 24 tool holder machine it may be designated as T25.

The OMP operates most of the time in the standby mode. It is activated by a **switch on** signal from the optical machine interface to the operating mode. After it is done with a probing routine it is switched off by a signal from the optical machine interface or pressing the RESET key.

The Renishaw probe comes with a standard 50 mm long ceramic stylus. At the end is a 6mm ruby ball. As the ceramic will easily break a (break away pin) is not required with this type of stylus. Other length probes with different diameter balls and lengths are available and may be used.

Renishaw Probe OMP40-2 Dimensions
(From Renishaw *Data Sheet H-5514-8200-01-A*)



Stylus overtravel limits		
Stylus length	$\pm X / \pm Y$	$+Z$
50 (1.97)	12 (0.47)	6 (0.24)
100 (3.94)	22 (0.87)	6 (0.24)

OTS – Optical Tool Setter

The **OTS (Optical Tool Setter) or tool probe** is used to set tool length offsets, measure the diameters of tools or detect broken tools. The table probe gives precise measurements of tool length and diameter. It is similar to the spindle probe as it is run by batteries and is turned on and off by the OMI or machine interface. With the Renishaw OTS a weak link break pin P/N A-5003-5171 is between the stylus and the probe body. This protects the probe body from over travel or a collision.

The Tool Setting probe is normally mounted on the table of the vertical machine center. Haas recommends placing the probe on the right upper hand corner of the table. In this configuration the probe window would face away from any flying chips and opposite the tool changer side. The Probe base is held down by a socket head cap screw and T-nut. The Renishaw probe is mounted on the base using a locating pin for proper alignment. The probe is fastened down by two set screws, see B below.

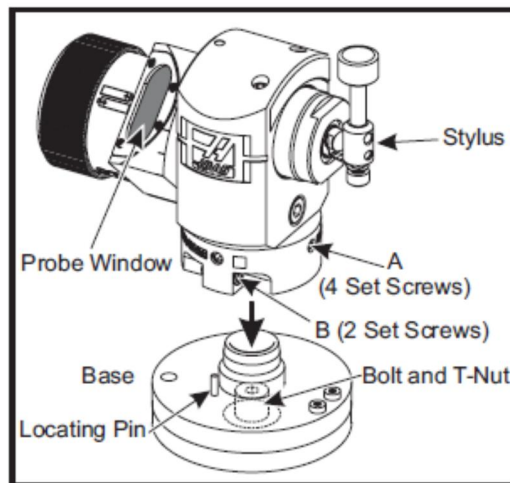
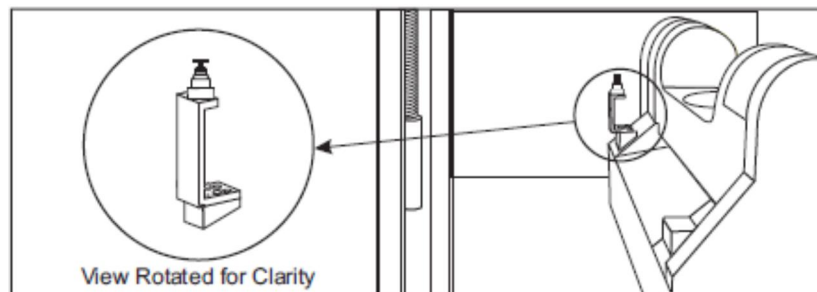


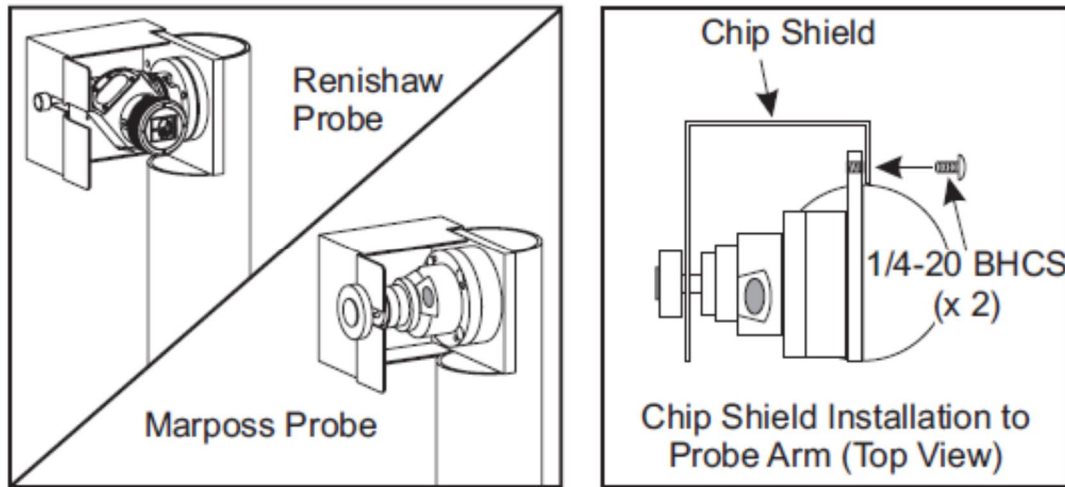
Table probes may also be mounted on plates. Two dowel pin holes are located on the base for mounting the probe on a plate where removal of the probe is required. This will relocate the probe in the same location.

Tool probes may also be mounted on risers for particular applications. Special brackets are needed with trunion mills, see below:



VF-6TR Probe Mounting

On horizontal mills the probe is mounted horizontally. On horizontal machines special brackets or arms are needed. See below:



See Haas ES0583 Document on WIPS Quick Setup for details.

Renishaw and Marposh Replacement Part Numbers

Marposh

- 60-0041 - Ceramic Stylus (Spindle Probe)
- 60-0042 - Disk Stylus (Table Probe)
- 60-0043 - Break Away Pin (Table Probe)
- 60-0075 - Angle Stylus (Table Probe)

Renishaw

- 60-0026 - Ceramic Stylus (Spindle Probe)
- 60-0028 - Disk Stylus (Table Probe)
- 60-0029 - Stylus Holder (Table Probe)
- 60-0030 - Link Break Protect (Table Probe)
- 60-0034 - Extension (Table Probe)

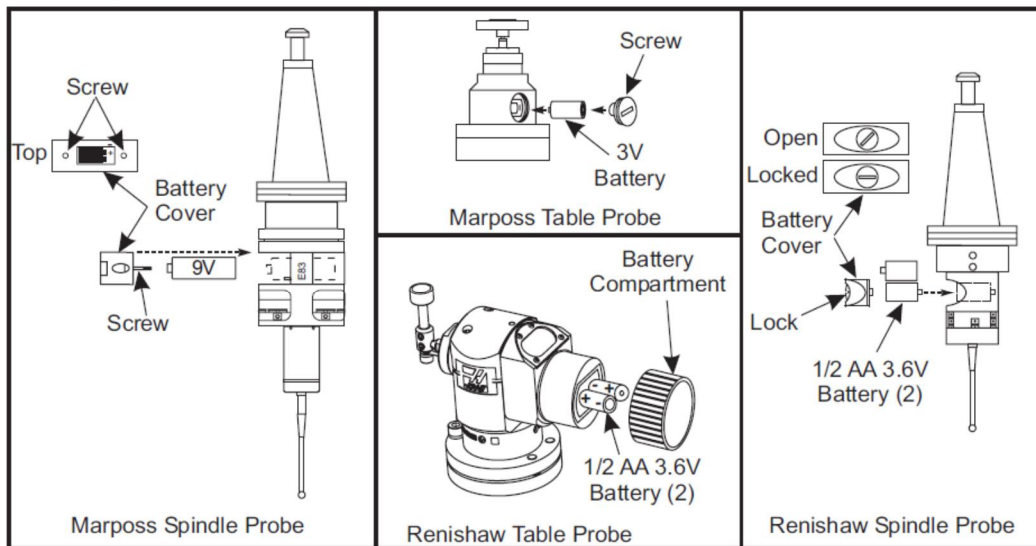
Batteries

WIPS table and spindle probes require batteries. The OMI or optical machine interface is supplied with power directly thru the machine controls. Below gives the information on the different batteries required for Marposs and Renishaw table and spindle. It also gives replacement information.

Batteries for Probing Options

			Haas Part #
Marposs	E83 Spindle Probe	9.0V Transistor	99-4354
Marposs	OTS30 Table Probe	3.0V CR123A Lithium	99-43554
Renishaw	OMP40 Spindle Probe OTS Table Probe	3.6V Lithium 1/2 AA	99-43553

Probe Battery Replacement



Marposs Spindle Probe – The Marposs Spindle Probe contains (1) 9-Volt battery. Use a 3mm Allen wrench to unscrew and remove the battery cover located on the side of the probe. Remove the spent 9V battery and insert a new one. Screw the battery cover back down on the 9V battery.

Marposs Table Probe – The Marposs Table Probe contains (1) 3-Volt battery. Use a coin to unscrew and remove the battery cover located on the side of the probe. Remove the spent 3V battery and insert a new one. Screw the batter cover back down on the 3V battery.

Renishaw Spindle Probe – The Renishaw Spindle Probe contains (2) 1/2 AA 3.6V batteries. Use a coin to unscrew and remove the battery cover located on the side of the probe. Remove both the spent 1/2 AA 3.6V batteries and insert new ones. Screw the batter cover back down on the 1/2 AA 3.6V batteries.

Renishaw Table Probe – The Renishaw Table Probe contains (2) 1/2 AA 3.6V batteries. Unscrew the battery cover/holder from the battery compartment located on the side of the probe. Remove both the spent 3.6V batteries and insert new ones into the battery holder. Place the battery cover/holder back onto the battery compartment.

(From Haas Wireless Intuitive Probe System, Quick Start Set-up and Operation. ES0583 rev C)

Battery life depends on how much the probe is used. The probe is only turned on when it is being used. After it is done with its routine it is shut off to conserve battery life. Renishaw gives the **standard life** of the LTC ½ AA 3.6 volt in the **OTS** as 300 hours continuous use, **5% usage** (72 minutes a day) at **140 days**. For the **OMP**, an average life of 170 hours continuous use, **5% usage** (72 minutes a day) at **115 days**. Most problems with probes stem from low batteries. Because the battery life of the probe on average is only about three months it is wise to keep spare batteries on hand. This is especially important if there is a night shift.

Trouble Shooting Battery Problems:

Most communication problems with IPS are caused by either low or dead batteries. Another problem may be accumulation of chips on the table probe. Looking at the LED lights from the probes will give an indication of weak or dead batteries.

LED Probe Status

Probe is in Operating Mode	Flashing Green light
Probe triggered in Operating Mode	Flashing Red light
Low battery in Operation Mode	Flashing Green and Blue
Probe triggered in Operating Mode low battery	Flashing Red and Blue
Battery Dead	Constant Red
Unsuitable Battery	Flashing Red Flashing Red and Green Flashing Red, Green and Blue

Probe Test

The following code verifies if the table probe (OTS), the spindle probe (OMP) and the machine interface (OMI) are functioning correctly.

When the vertical machine center is powered up the OMI **3 LED's** (light emitting diodes) are all **red**. The spindle and table probe are both off, no LEDs are on and black.

Enter the following code in MDI. Running the short program in MDI will **turn the table probe (OTS) on**.

M59 P2 (TOOL PROBE)

G04 P1.

M59 P3

If the machine interface and the table probe are working properly the **tool probe LED starts blinking green**. The machine interface or **OMI 3 LEDs will turn solid green**. Tapping the probe with your finger (simulating a hit) will temporarily turn the **tool probe LED flashing red** and **one** of the **LED's** of the machine interface **OMI** simultaneously will temporarily **turn red**. Also a **beep** will be heard. Press the **RESET** button to turn off the table probe.

The following code will turn on the spindle probe.

M69 P2 (WORK PROBE)

M59 P3

When the spindle probe (**OMP**) is turned on a LED in the spindle probe will start **blinking green**. At same time the machine interface or **OMI 3 LEDs will turn solid green**. The second line of code commands the machine to make an incremental move in the -X direction. Tapping the probe with your finger (simulating a hit) will stop the machine movement in X. This illustrates the G31 skip function. Along with stopping the machine movement the tapping will temporarily turn the **work probe LED flashing red** and **one** of the **LED's** of the machine interface **OMI** simultaneously will temporarily **turn red**. Also a **beep** will be heard. After running the above probe tests press the **RESET** button to turn off the different probes.

Probe Programs

Macro Programs: A **Macro** is a sub-program which includes commands that are not G code. Macros include decision statements such as IF, GOTO, various math functions and the use of variables. It provides more options for the use of CNC. Sometimes they are used to perform repetitive functions such as G83 peck drilling. G83 is a Macro program which is not visible on the Haas CNC control.

Probing macros direct the machine to perform complex probe movements. From these motions, probe contacts or hits are used to calculate offset locations, calculate distances, tool lengths with the use of variables. Most macro programs are long and difficult to follow simply by the fact of the use of so many different variables.

Variables: In computers a variable is a storage location where a known or unknown quantity (value) resides. This location has an associated symbolic name. In Haas machines the location is designated with a number with the pound (#) symbol in front of it.

There are three types of variables: local, global and system. Local variables are #1 thru #33. With local variables a G65, M30 or reset will clear all values to zero. Global variables have three ranges 100-199, Local and Global variables may be seen and changed by pressing the Current Commands key and using the page up or down. The title of the page will be Macro Variables. To search for a particular variable enter the variable number and search using the arrow keys. The values on these pages may be changed by highlighting the number, enter a value and press the Enter key. All the variables may be zeroed by pressing the origin key.

System variables are used by the machine tool to store particular information which the programmer has access to and may alter. They may not be altered thru the Macro Variable pages.

For example variable number #2001 stores the value for tool offset length number 1. What information a particular system variable stores may be found in the Haas Mill Operator's Manual. Some system variables are read only and may not be altered by the programmer.

G65 Macro Program Call: Macro program command **G65 Pxxxx** will call up a particular macro program. Using the G65 command allows the use of variables in the macro program called.

Probe programs use the **G31: Feed Until Skip** command. In the G31 block a specific axis move is commanded with a feed rate. When the move is started it will continue until the position is reached or the probe makes a hit. If the probe makes a hit the feed will stop and the program will skip down to the next block. An audible beep will be heard when the probe makes a hit.

9000 Programs: Probing programs are all designated as 9xxx programs. The next page gives a typical list of Renishaw programs loaded in a Haas vertical machining center. To prevent these programs from being altered **Setting 23 9xxx PROGS EDIT LOCK should be set to ON**. With setting 23 is set to ON the probe 9000 programs shown below will not viewed or accessible when the LIST PROGRAM key is depressed.

Macro Program List

<u>Macro Program Name</u>	<u>Function</u>
009023	(REN EASYSET V2.8)
009301	(REN IPS CALIBRATION V2.8)
009351	(REN IPS BORE RECT V2.8)
009352	(REN IPS BOSS BLOCK V2.8)
009353	(REN IPS WEB X V2.8)
009354	(REN IPS POCKET X V2.8)
009355	(REN IPS WEBB Y V2.8)
009356	(REN IPS POCKET Y V2.8)
009357	(REN IPS INSIDE CORNER V2.8)
009358	(REN IPS OUTSIDE CORNER V2.8)
009359	(REN IPS SINGLSE SURFACE V2.8)
009376	(REN OTS CALIBRATION)
009377	(REN OTS TOOL SET)
009378	(REN OTS TOOL SET L D)
009721	(REN X DIAM)
009722	(REN Y DIAM)
009723	(REN ACT OFFSET)
009724	(SETTINGS)
009725	(SETTINGS CHECK)
009726	(REN BASIC MEASURE -2T)
009727	(REN VECTOR OD-ID)
009730	(REN PRINT)
009731	(REN #[#161+9]\ LOAD/ATAN)
009732	(REN OFFSET TYPE)
009799	(REN SETTING)
009801	(REN CAL PROBE LENGTH)
009802	(REN CAL XY OFFSET)
009803	(REN CAL STYLUS RAD)
009804	(REN CAL VEC STYLUS RAD)
009810	(REN PROTECTED POSN)
009811	(REN XYZ MEASURE)
009812	(REN WEB POCKET MEASURE)
009814	(REN BORE BOSS MEASURE)
009815	(REN INTERNAL CORNER)
009816	(REN EXTERNAL CORNER)
009817	(REN 4 TH AXIS X MEASURE)
009818	(REN 4 TH AXIS Y MEASURE)
009819	(REN PCD BORE BOSS)
009820	(REN STOCK ALLOWANCE)
009821	(REN ANG XY SINGLE SURFACE)
009822	(REN ANG WEB POCKET)
009823	(REN 3-POINT BORE BOSS)
009830	(REN MULTI-STYLUS STORE)
009831	(REN MULTI-STYLUS LOAD)

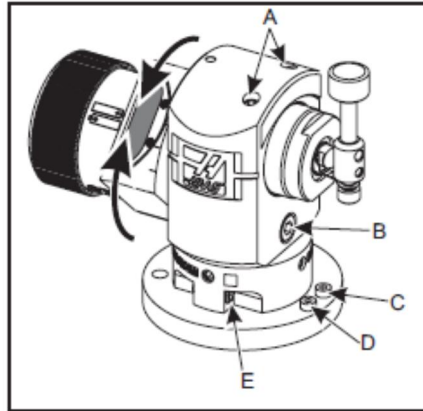
<u>Macro Program Name</u>	<u>Function</u>
O09832	(REN OMP 40 ON)
O09833	(REN OMP 40 OFF)
O09834	(REN FEATURE TO FEATURE)
O09835	(SPC OFFSET UPDATE)
O09843	(REN XY PLANE ANGLE)
O09850	(RENISHAW TOOL SELECT)
O09851	(RENISHAW LENGTH SET)
O09852	(RENISHAW DIA. SET)
O09853	(RENISHAW AUTO SET)
O09854	(RENISHAW DELAY)
O09855	(REN OTS ON)
O09856	(REN OTS OFF)
O09995	(WIPS FOR RENISHAW V2.8)
O09996	(HAAS VQC WIRELESS PROBE Eng Inc)

From the list above some 60 programs are used with different Renishaw probing applications. With **NGC WIPS** (Next Generation Control Wireless Intuitive Probe System) the appropriate programs are selected and variable information like diameter, work offset, or tool number are inserted into the different probe programs. Then work offset or tool length information is inserted into the Haas controls after the programs are run. This greatly simplifies using probes to get information like work offsets and tool length offsets. Probe selection is further simplified with visual screens showing the various options for getting tool lengths for different tools. Also the various features that may be used to determine work offsets are easy to pick out from the different screens available.

Probe Alignment

The OMI, OTS and the OMP must be aligned so that the infrared light will be able to be detected by the photodiodes from the machine interface (OMI) to both the spindle probe (OMP) and table OTS and vice versa. Assuming the probes are able to communicate with each other (see Haas ES0583 Haas Quick Start and Set Up) for recommended alignment locations.

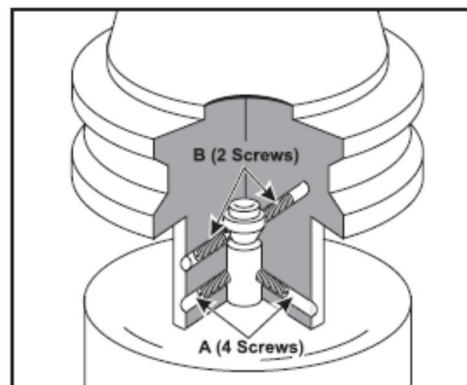
For accuracy the **table probe (OTS)** must be within .0001 inch parallel with the machine X and Y axis plane. Place a .0001 test indicator on the spindle with an Indicol or other method. Move indicator so it touches the top surface of the probe. The top surface of the probe cylindrical stylus should be within .001 inch when the table is moved in both the X and Y directions.



For **adjustments in the X axis** first loosen **screw C** above. **Then screw D** which acts like a jack screw can be adjusted so there is less than .001 inch change from one side to the other in the X axis. When properly adjusted tighten screw C down.

When moving the table in the **Y axis, adjustment** may be made by alternately adjusting the two **A screws**.

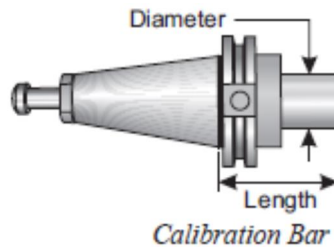
For the **spindle probe (OMP)** alignment place the indicator on the stylus ruby ball. Rotate the spindle. If run out of more than .0002 inch is detected, adjust the spindle probe by first loosening the two B screws. The four A screws may then loosened and tightened until there is run out below .0002". Then tighten down the B screws. See below:



VPS – Visual Programming System use with Probes

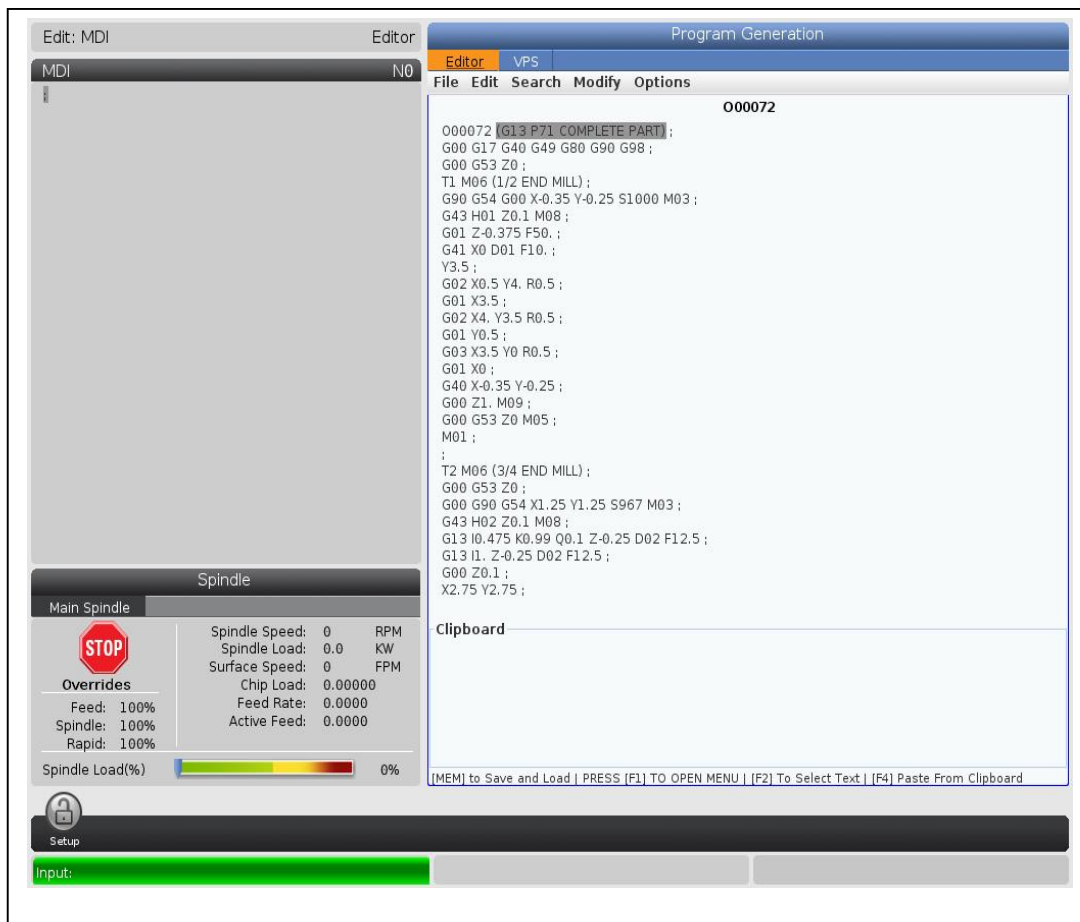
Probe Calibration

To calibrate the probe, **first** a standard must be purchased or created. The calibration bar as pictured below is simply a ground pin of known diameter placed in a tool holder. The other information which is needed is the length from the end of the tool to the gauge line of the tool. This gauge length may be determined using a Parlec Tool Setting machine.

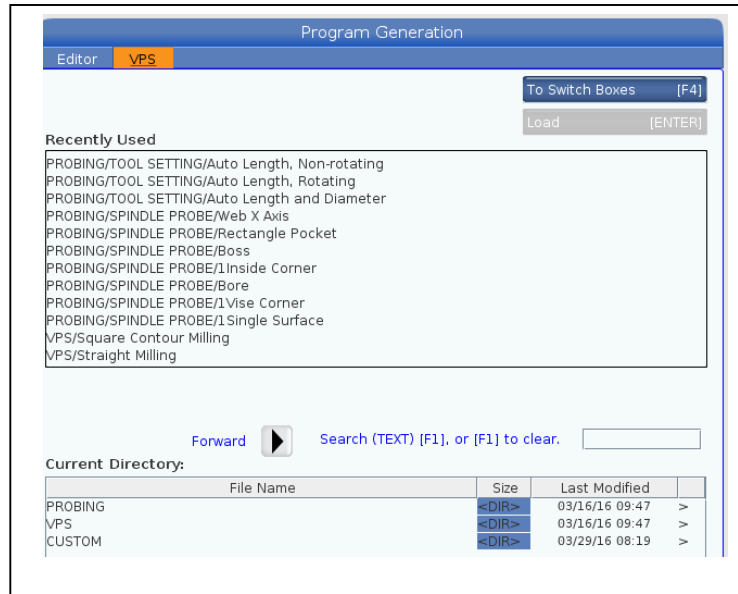


The second tool needed is a calibrated ring gage. This must be clamped to the table. After the spindle and table probes are aligned for run out and parallelism respectively Virtual Programming System (VPS) must be accessed thru the Haas control.

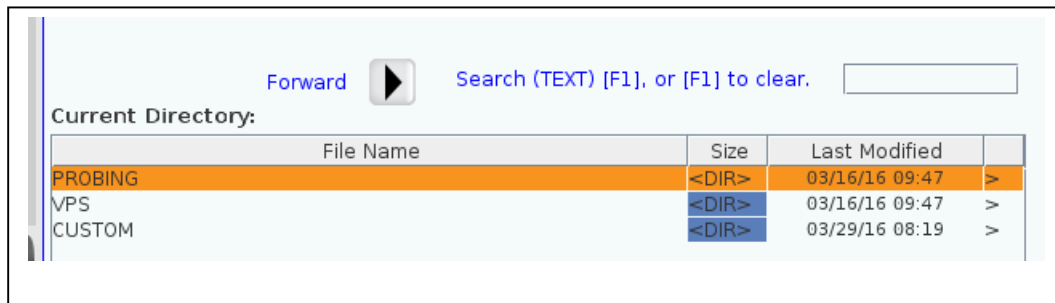
To access **VPS** press the **EDIT** key and then cursor using the **^** up key to highlight **Editor** in orange as shown below.



Cursor to the right using the > key to VPS which brings the following up. Note the library of recently used programs at top. They can be opened by depressing **F4** key, then cursing to the desired program down by using the down arrow v key. Otherwise the VPS directory can be opened by using the down v key.

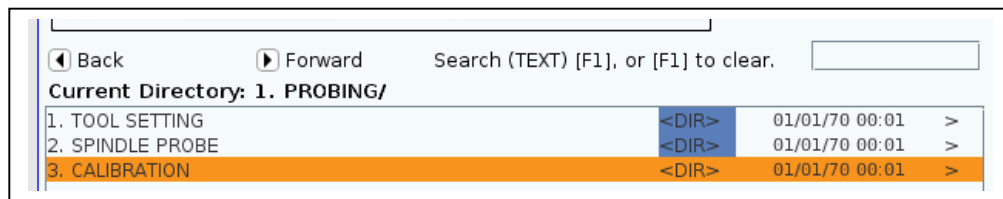


This will bring you to the VPS Directory. High light in orange 1. PROBING.

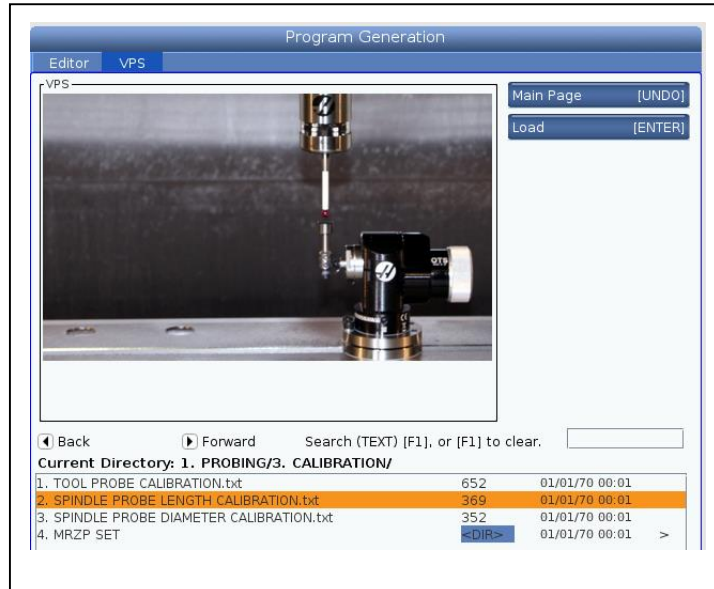


Cursor to the right using the > key will open the Probing Directory

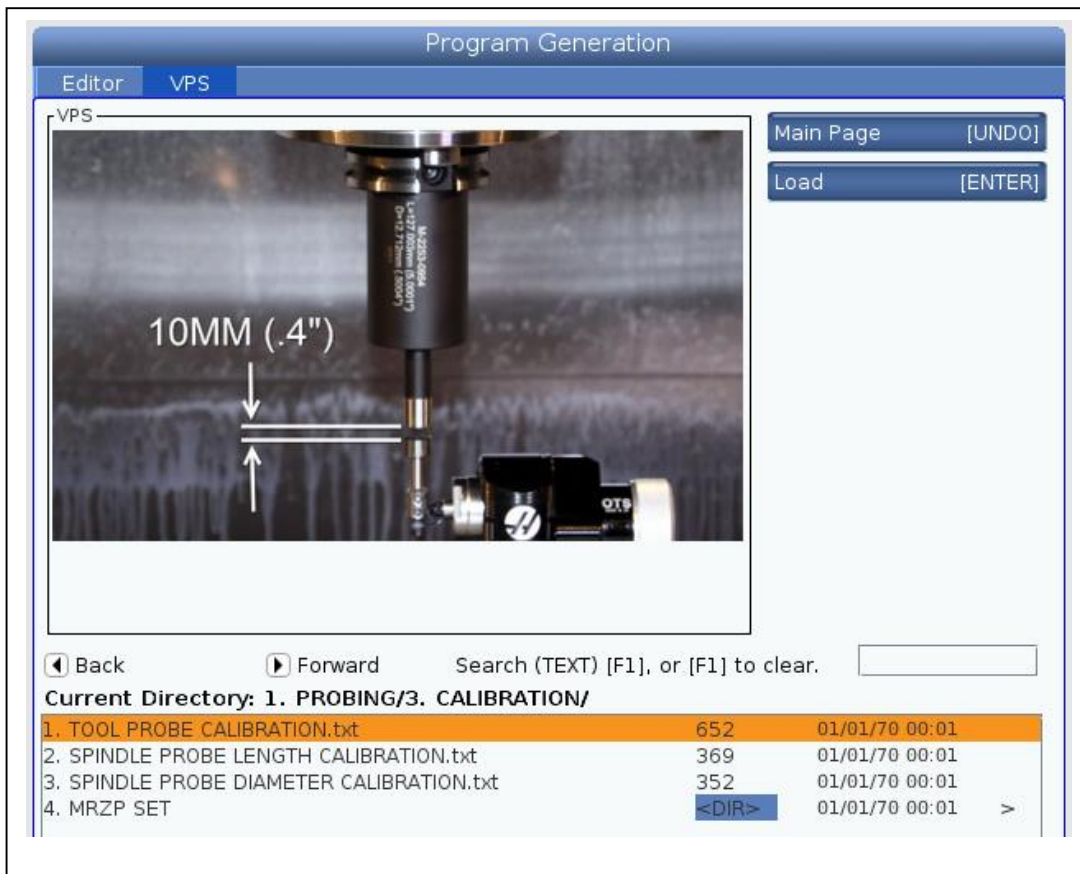
Next, Highlight in orange 3. CALIBRATION



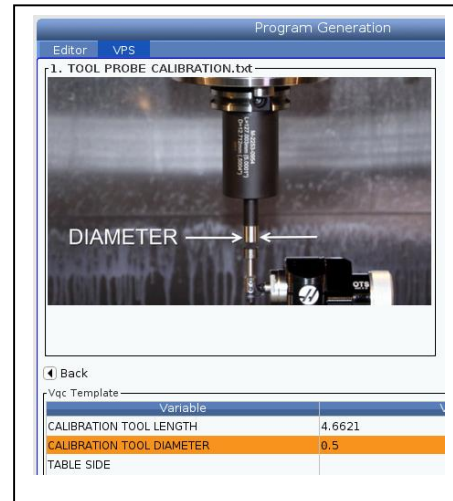
Cursor the right using the > key which opens options under calibration below.



Use the ^ or v key to highlight in orange **TOOL PROBE CALIBRATION.txt**



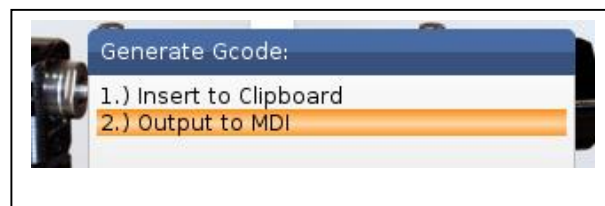
This screen directs the operator to move the tool standard to the center of the tool probe and .4” above it. Press [ENTER] key to open page to enter variables for tool probe calibration below. Key in the tool length and diameter of the standard.



Using the v down cursor key the orientation of the tool probe with respect to the sides of the machine is keyed in. Normally the tool probe is placed on the upper right hand side with orientation 2 as shown in the lower left (G=2) photograph.



Next the code is created by pressing the F4 key as shown above. A pop up appears as below.



Cursor v down to highlight [2.] Output to MDI] then press the [ENTER KEY]

This will put the calibration program in MDI shown below.

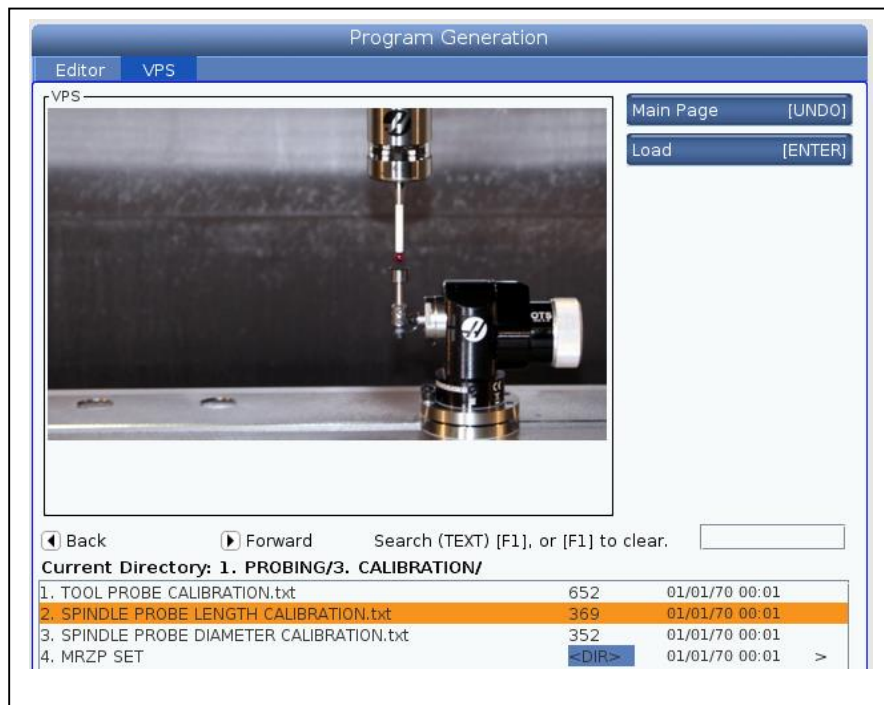
```
(CALIBRATION TOOL LENGTH = 4.6621)
(CALIBRATION TOOL DIAMETER = 0.5)
(TABLE SIDE = 2. LEFT = -2, RIGHT = 2)
G00 G90
G65 P9023 A20. K4.6621 S0.5 D2.
M30
```

Run this program in MDI. The Diameter and the X, Y, and Z location of the table probe disk will be stored in the machine controls to be used later. It uses this information to determine the Tool Length and Diameter when setting tools.

The next step is to calibrate the Z tool length of the work probe off the tool probe on the table. Index the machine to the tool where the spindle probe is located. Usually this is designated as the last tool.

In machines with the side mount option the spindle probe is always designated as an **H-Heavy Small Diameter tool in the Tool Pot Location** screen Because of the delicate nature of the electronics in the spindle probe. A rapid indexing may disturb the calibration of the spindle probe especially in the SS machines.

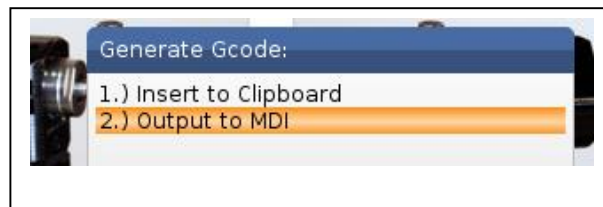
Next get back in the VPS Screen by pressing the **[HAND JOG]**, then the **[EDIT]** key, this will bring the calibration directory. Cursor down v to open calibration page then cursor down v to **2.SPINDLE PROBE LENGTH CALIBRATION.txt** as shown below.



Press the **ENTER** key and the following screen will appear. Move the spindle probe just .400" above the table probe disk. Key in the tool number of the probe and press the **ENTER** key.



Next press the **F4** key, cursor down **v** to 2.) Output to MDI.



This creates the following code in MDI.

Run the program in MDI. It puts the gauge length measured in the H value for the respective tool for the spindle probe. In this instance T24

(2. SPINDLE PROBE LENGTH CALIBRATION.txt)

(TOOL NUMBER OF PROBE: 24)

G00 G90

G65 P9023 A21. T24

M30

For the final step in calibrating the spindle probe, the diameter of the ruby ball and calibration in X and Y needs to be determined. Jog the work probe to the center of the calibration ring gage and just below as shown in the third and final calibration sequence. The MRZP SET is only used for 5 axis calibration in Haas UMC machines.



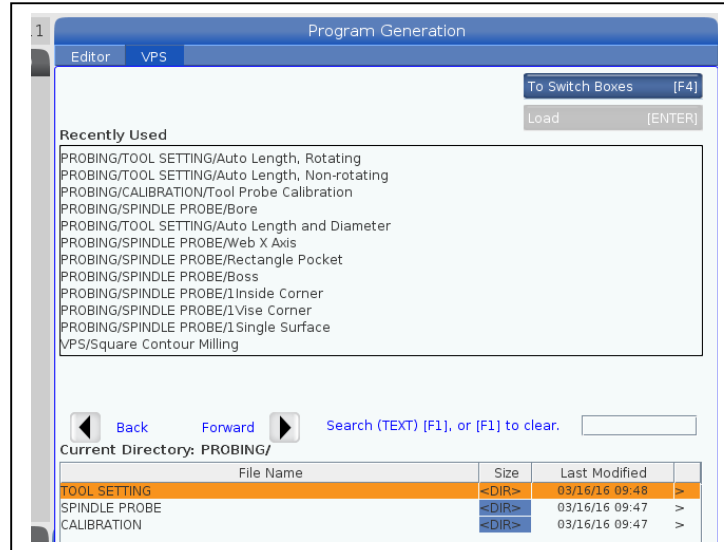
Press **ENTER**, key in the gage ring diameter, press F4, and output the code into MDI. This will put the following code into MDI for a 1 inch gage ring. Run the program in MDI and the probe calibration is complete.

(3 - SPINDLE PROBE DIAMETER CALIBRATION.txt)
(GAGE RING DIAMETER: 1.)

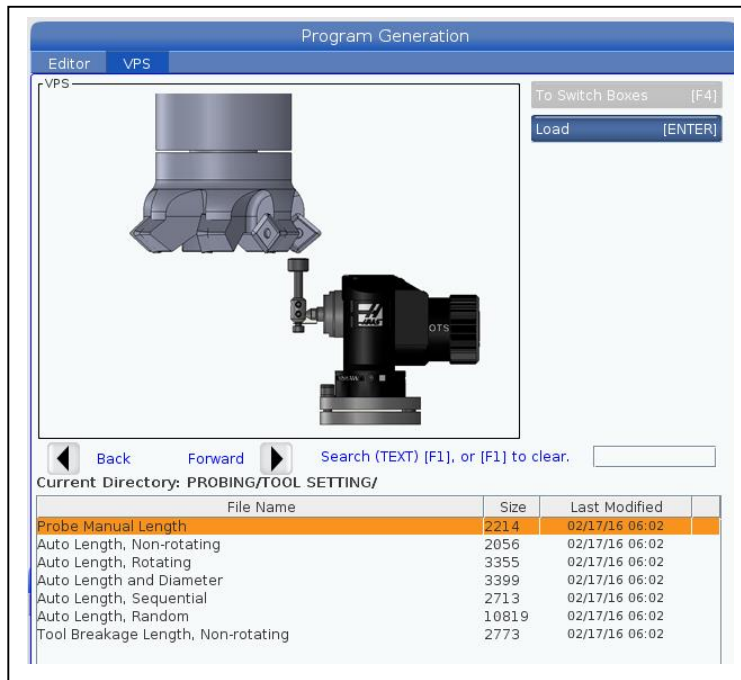
```
G00 G90  
G65 P9023 A10. D1.  
M30
```

Setting of Tool Length Offsets using VPS

Open VPS and get to the following page and highlight 1. TOOL SETTING as shown below.



Right cursor > opens up the following page.

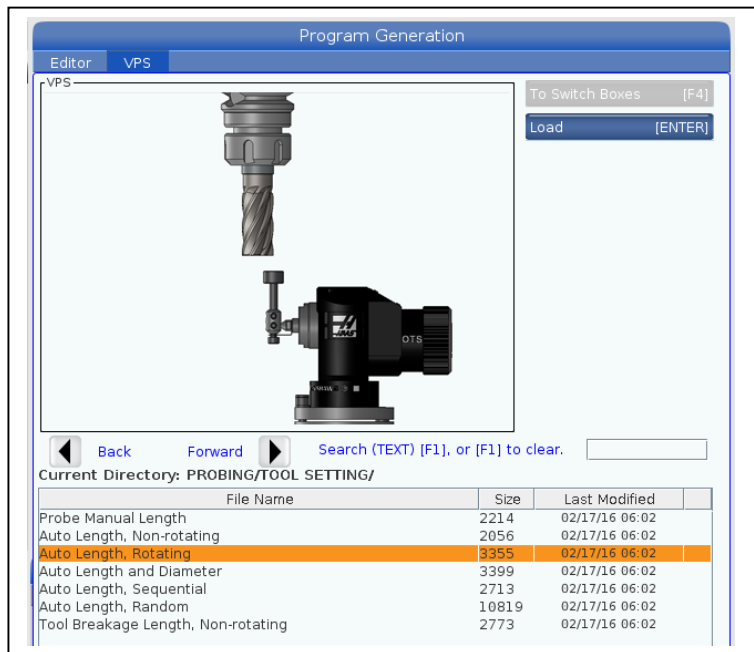


This page shows all the different ways tools may be touched of using the tool probe.

1. **Probe Manual Length** is the only one where the tool needs to be manually moved to a particular location. In this instance .40" centered above the tool probe. The other options just need the correct variables entered and the machine will move the tool to correct location.
2. **Auto Length, Non-rotating** is used for drill type tools. This includes reamers and taps.
3. **Auto Length, Rotating** is used for multiple tooth tools to determine the length of the longest flute.
4. **Auto Length and Diameter** is used when not only the tool gauge length is required but also an accurate value for the tool diameter is needed. This would be an end mill or cutter with a 90° angle for cutting shoulders.
5. **Auto Length, Sequential** is used for getting gauge lengths for drill type drills that are all in sequence.
6. **Auto Length, Random** is used for touching off all the drill type tools on set up with random tool numbers.
7. **Tool Breakage Length Non-rotating** is for checking a tool length while running a program to see if a tool is broken and stopping the program from running further.

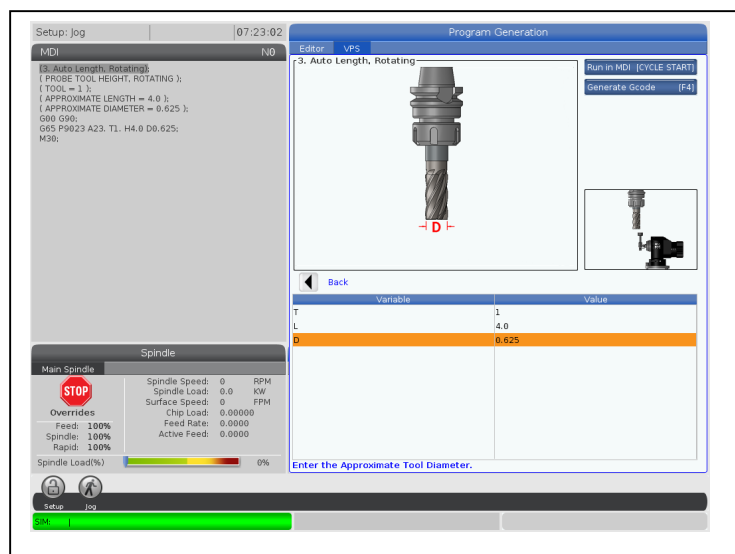
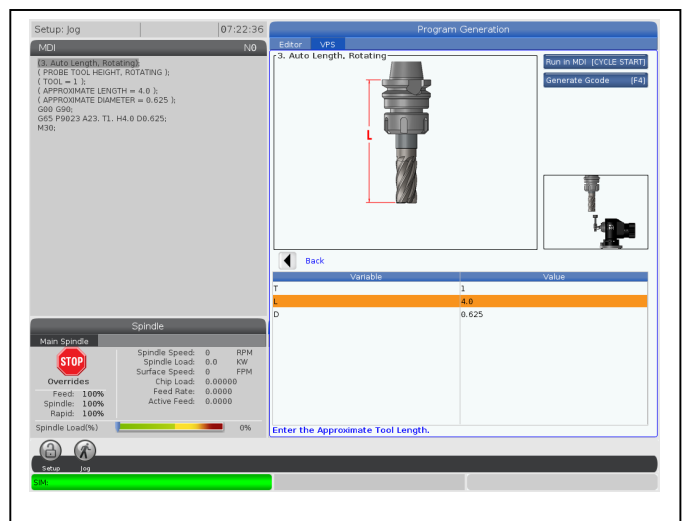
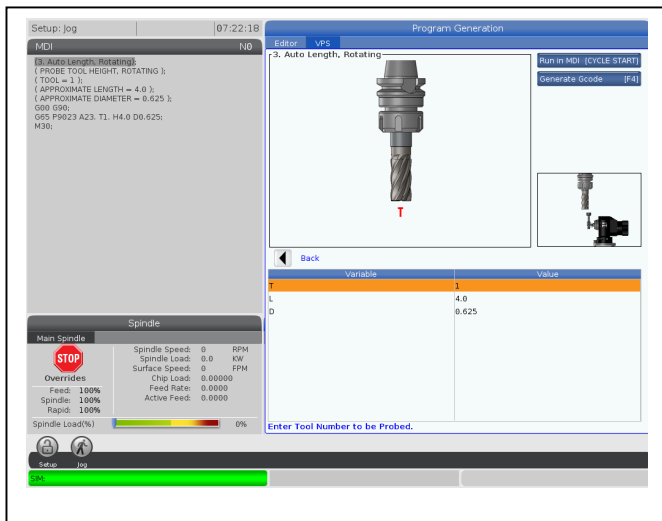
All the options above are run similarly, the option is selected and information for variables are provided by the operator.

Below the option Auto Length rotating is highlighted.



Note it visually indicates the information it needs for the variables for the run the probe program.

Note the < Back and > Forward key used for navigation.

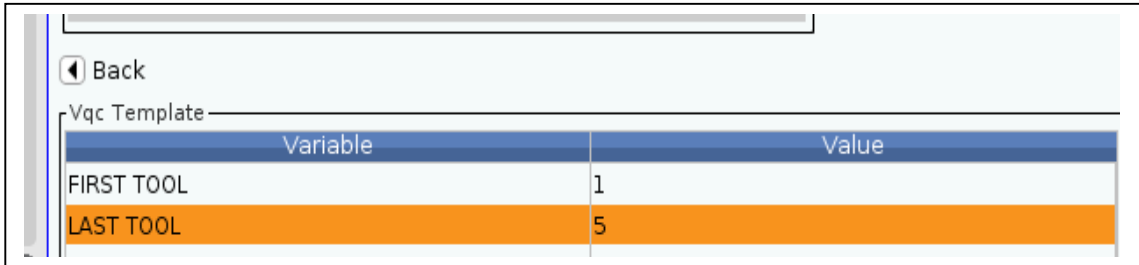


Pressing the > forward key brings the screen below where the variables are entered. Below Tool #2 is entered, the approximate gauge length measured with a scale 4.5", the diameter .5". F4 key is pressed to generate the code and output to MDI selected. The program then is run in MDI mode. This is the code that was generated below:

```
(3 - PROBE AUTO LENGTH, rotating.txt)
G00 G90
G65 P9023 A23. T2 H4.5 D.5
M30
```

The code for rest of the options is created in a similar fashion.

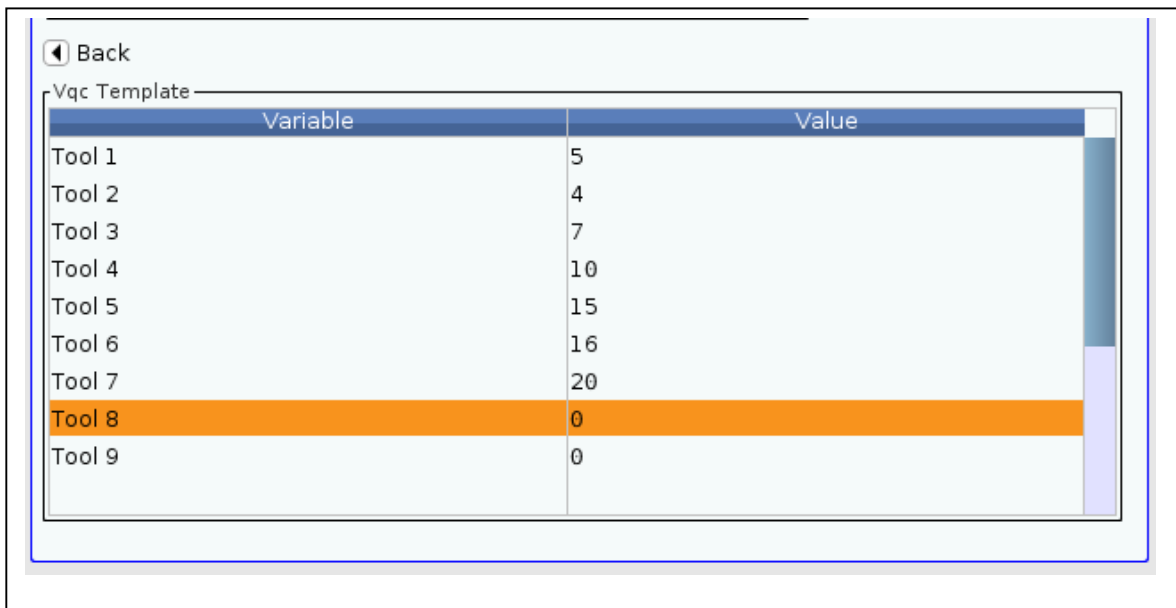
For sequential tools auto length the template looks like below. The first drill type tool to be probed is T1. The last is T5. All the tools in between will be touched off sequentially.



A screenshot of a software interface showing a 'Vqc Template' dialog. At the top left is a 'Back' button with a left-pointing arrow. Below it is the title 'Vqc Template'. The main area contains a table with two columns: 'Variable' and 'Value'. The first row has 'FIRST TOOL' in the 'Variable' column and '1' in the 'Value' column. The second row has 'LAST TOOL' in the 'Variable' column and '5' in the 'Value' column. The second row is highlighted in orange.

Variable	Value
FIRST TOOL	1
LAST TOOL	5

For auto random drill type tools the template is:



A screenshot of a software interface showing a 'Vqc Template' dialog. At the top left is a 'Back' button with a left-pointing arrow. Below it is the title 'Vqc Template'. The main area contains a table with two columns: 'Variable' and 'Value'. The rows are labeled 'Tool 1' through 'Tool 9'. The 'Value' column contains the following numbers: 5, 4, 7, 10, 15, 16, 20, 0, 0. The row for 'Tool 8' is highlighted in orange.

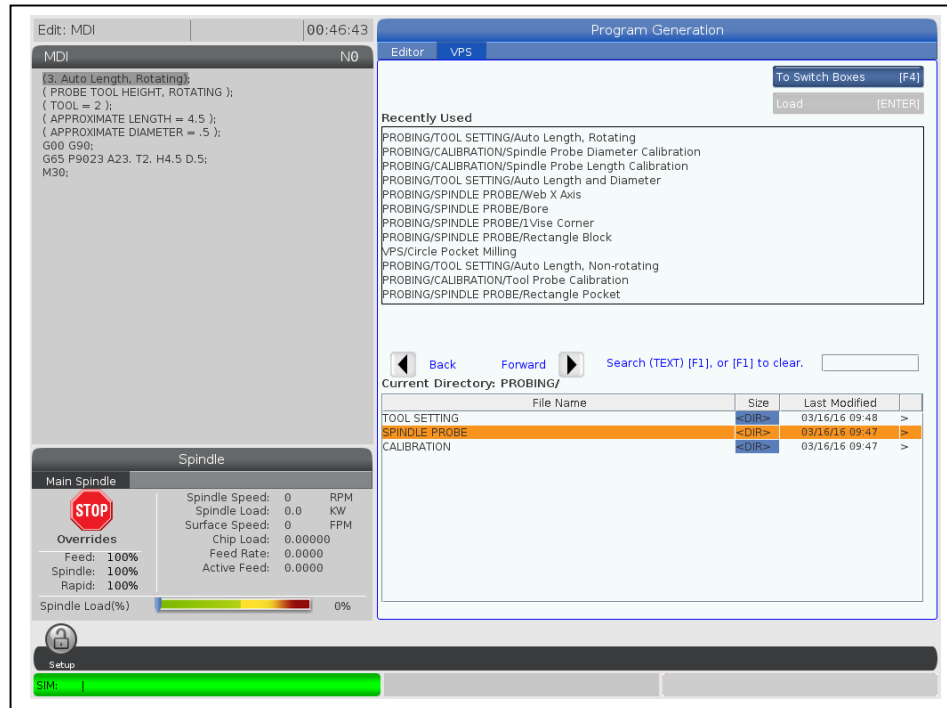
Variable	Value
Tool 1	5
Tool 2	4
Tool 3	7
Tool 4	10
Tool 5	15
Tool 6	16
Tool 7	20
Tool 8	0
Tool 9	0

In this instance only 7 tools are probed. On the left is the order that they are probed. On the right is which tool is probed. As only 7 tools need to be probed Tool 8 and Tool 9 are left as 0.

For the Tool Breakage option the code is not generated in MDI but inserted into the clipboard. It is then copied inside the program where it is desired. This is usually after the tool is done with its operation and before the next tool is indexed.

Setting Work Offsets using VPS

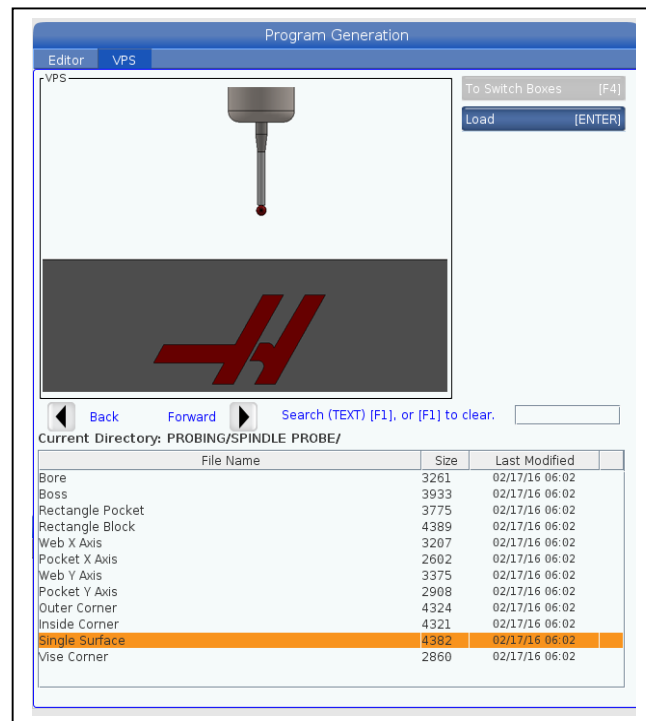
First go to the VPS probe directory and highlight spindle probe as shown below.



Next right cursor >.

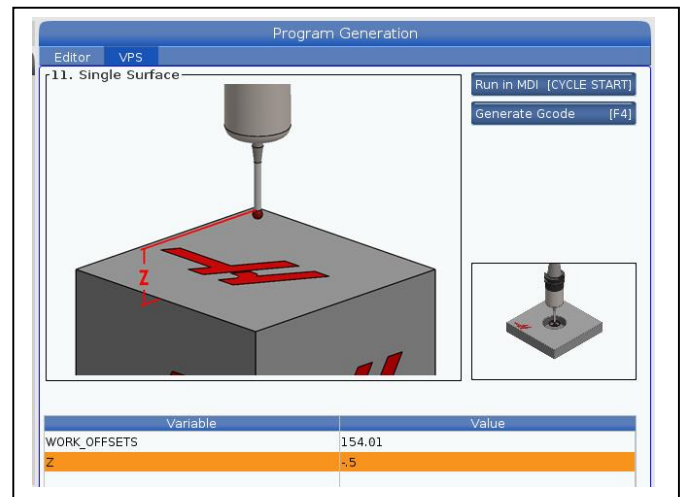
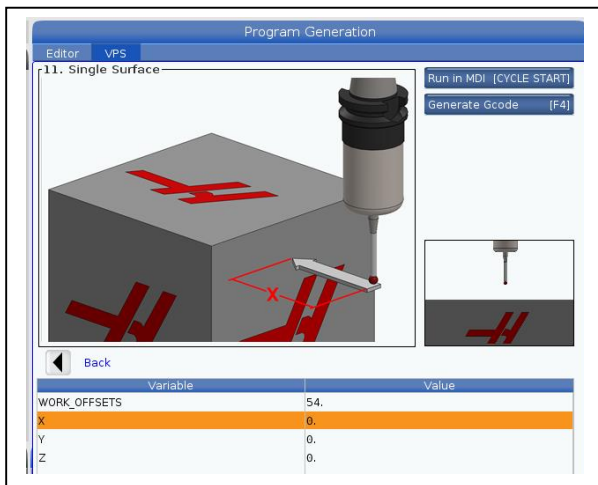
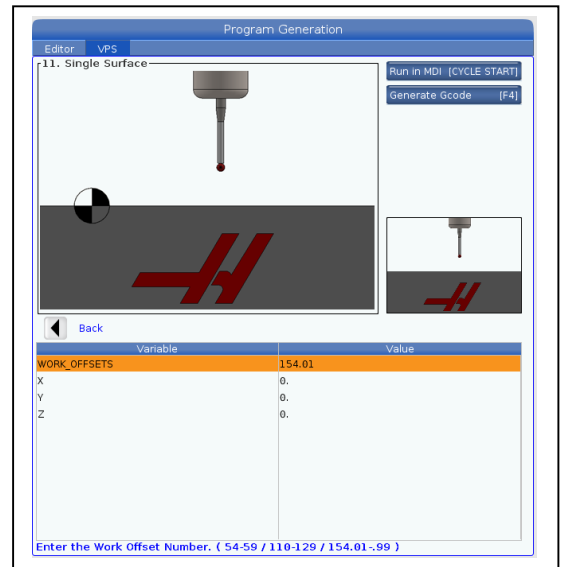
This shows all the options for setting work offsets with the probe. Highlight the option you want in orange. In this example **11. SINGLE SURFACE.txt** has been highlighted.

As the note instructs position the probe near the surface you want probed.

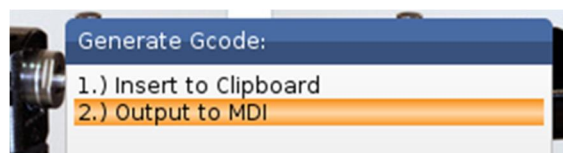


Next right cursor >. This indicates which direction the probe is to move. In the example to the right in the Work Offset line 154.01 is entered. This represents selection of one of the extra offsets that are offered by Haas: G154 P1 For G55 just enter 55.

Next the axis and direction in + or – are entered. In this example the movement of the probe will be in the negative Z direction by negative .500". Only one X, Y, or Z can be selected.



Next press the F4 key to generate create the code and place in MDI.

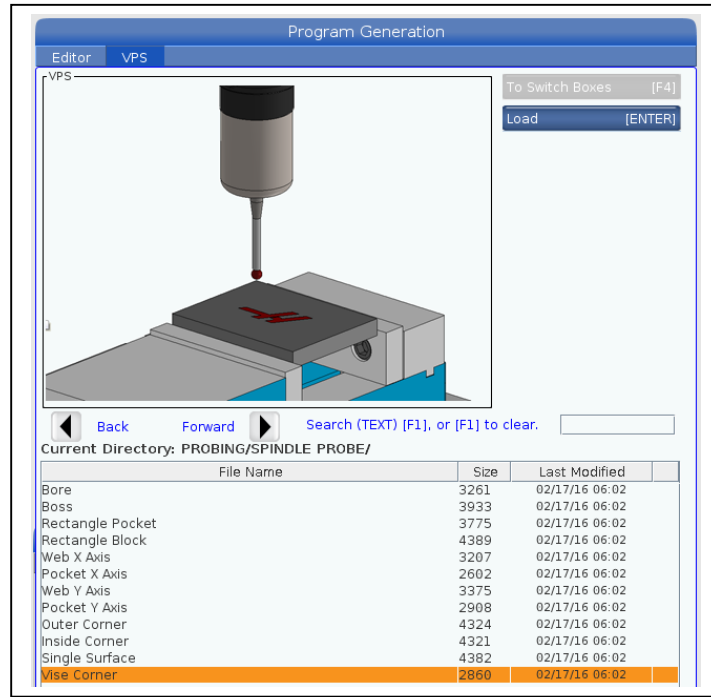


The following code is put in MDI and may be run to determine the Z value for G154 P1.

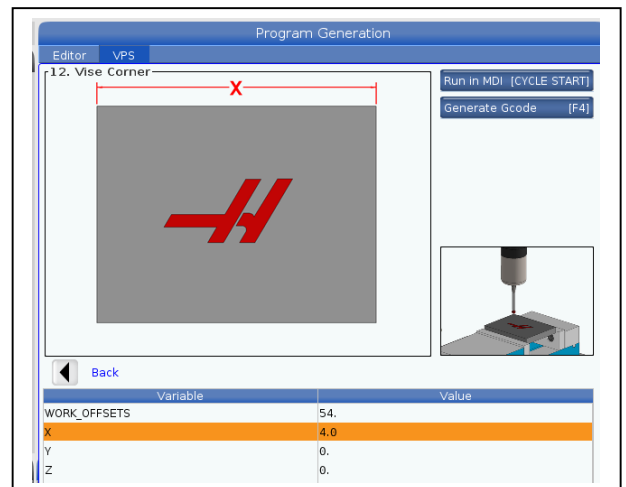
```
(11. Single Surface)
(SINGLE SURFACE PROBE )
(WORK OFFSET G154.01 )
G65 P9995 W154.01 A20. H-.5
```

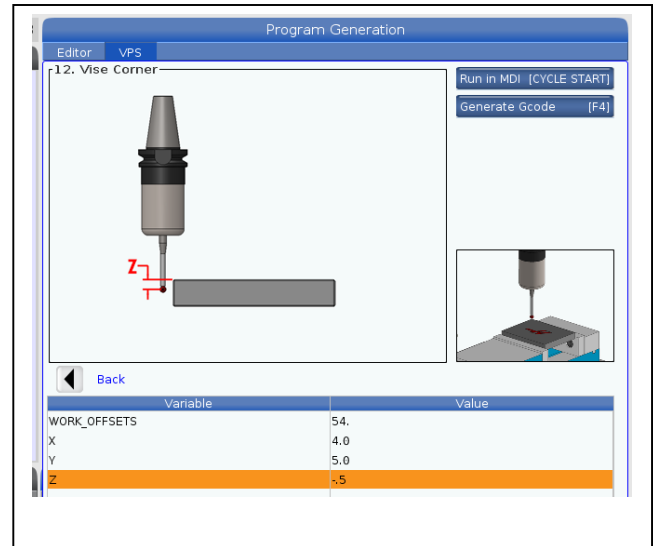
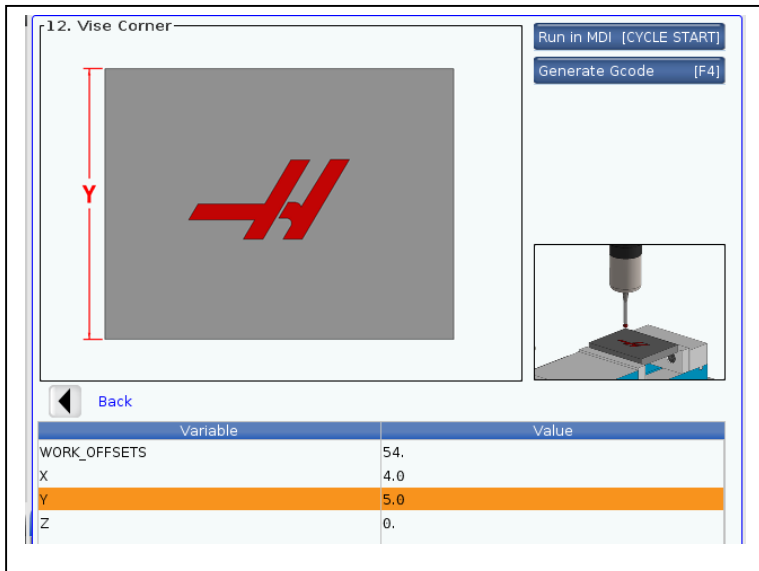
Probing Vise Corner

Go back to the spindle probe directory and highlight **Vise Corner**. This will probe the upper left corner of your part. Position the spindle probe directly above the upper left hand corner of the part.

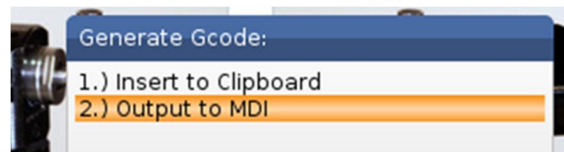


After hitting enter it gives you a series of pictures to show the information it needs for each variable by cursing to the variable.





Next press the F4 key to generate create the code and place in MDI.



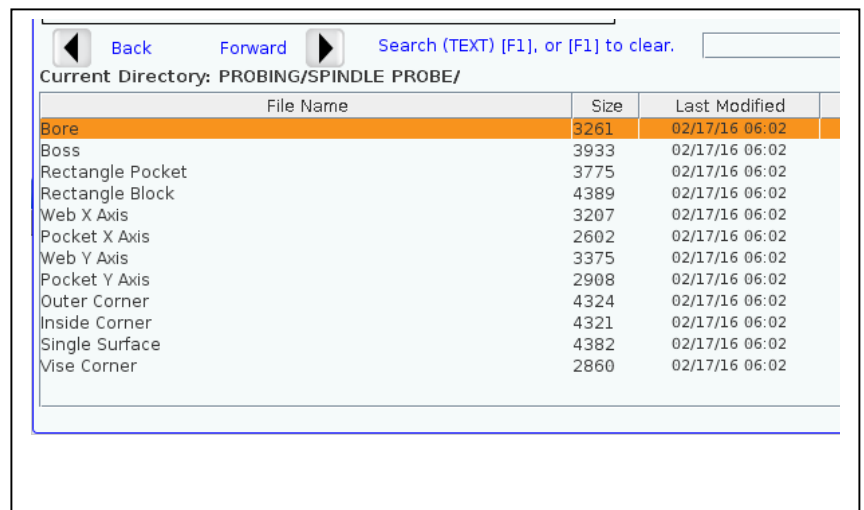
This gives the following code in MDI

(12. Vise Corner)
 (VISE CORNER PROBING)
 (WORK OFFSET G54.)
 G65 P9023 A17. S54. X4.0 Y5.0 Z-.5

Haas provides 12 different ways to set work offsets as shown to the right.

All provide visual aids so the right values are for the various variables.

Some have fields which will adjust the work offset in X and Y. These are used in special circumstances. Normally not used in most shops.



Probing Tool and Work Offsets thru the Offsets Page

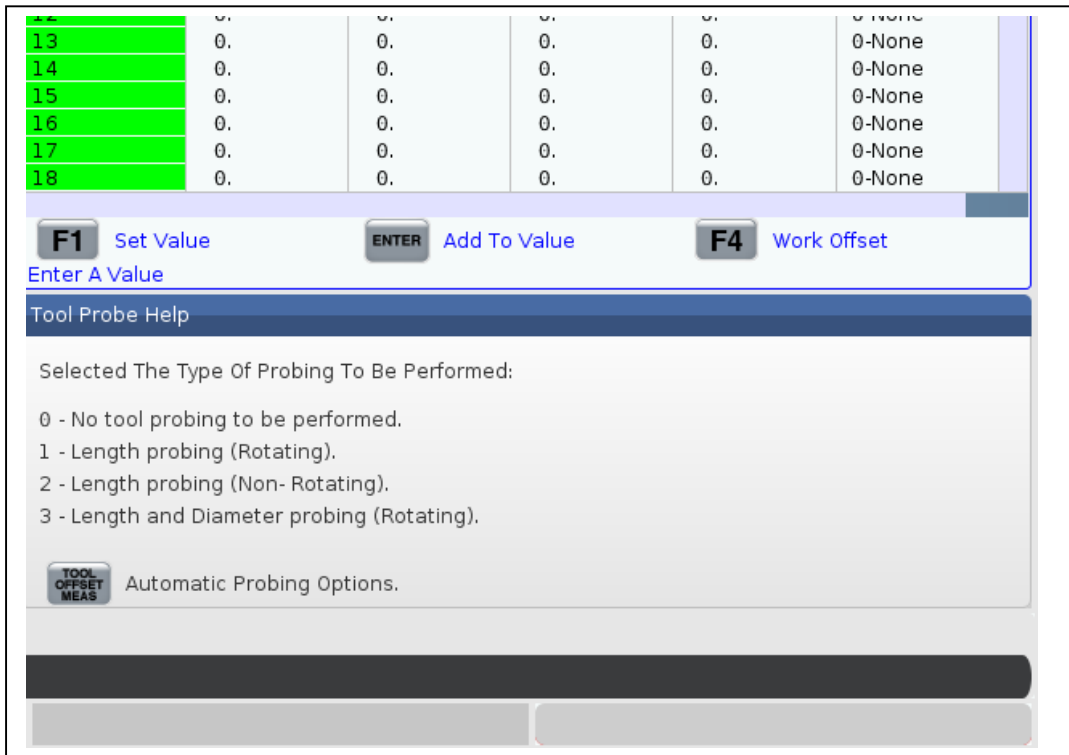
Tool and Work probing is simplified by going thru the Tool and Work Offset Page

First go to the offset pages by depressing **OFFSET** key. If say the work offset page comes up and the tool offset page is desired the **F4** key will **toggle** between the two different pages. Below shows the Tool Offsets page where Tool 4 is highlighted in orange.

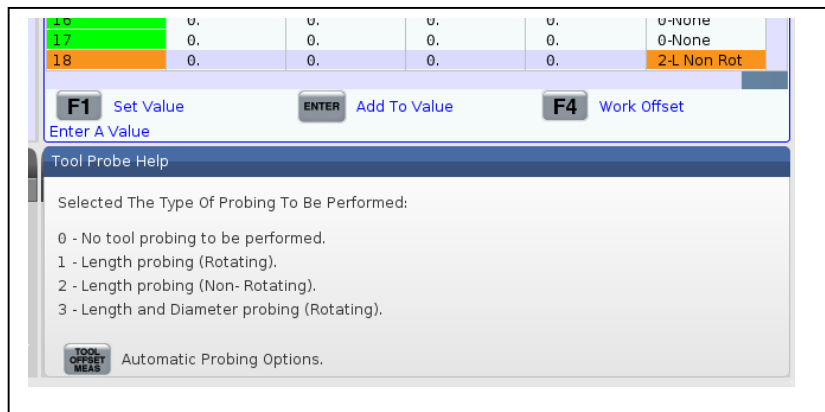
Offsets						
Tool		Work				
Tool Offset						
▶ Tool Info						
Tool	1	Coolant	H(Length)		D(Diameter)	
Offset		Position	Geometry	Wear	Geometry	Wear
1	Spindle		4.0000	0.	0.6250	0.
2			0.	0.	0.	0.
3			0.	0.	0.	0.
4			0.	0.	0.	0.
5			0.	0.	0.	0.
6			0.	0.	0.	0.
7			0.	0.	0.	0.
8			0.	0.	0.	0.
9			0.	0.	0.	0.
10			0.	0.	0.	0.
11			0.	0.	0.	0.
12			0.	0.	0.	0.
13			0.	0.	0.	0.
14			0.	0.	0.	0.
15			0.	0.	0.	0.
16			0.	0.	0.	0.
17			0.	0.	0.	0.
18			0.	0.	0.	0.

F1 Set Value **ENTER** Add To Value **F4** Work Offset
Enter A Value

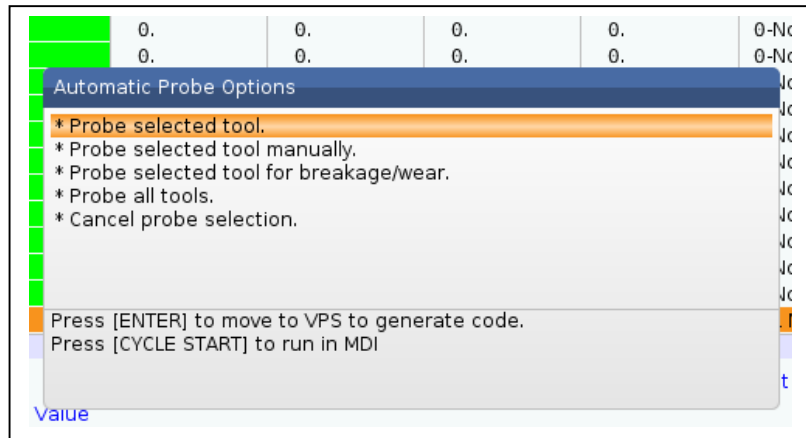
To get to the probing column right cursor > all the way to the end as shown below. At the top of the table column is labelled Probe Type. On the next page the 4 different options are shown.



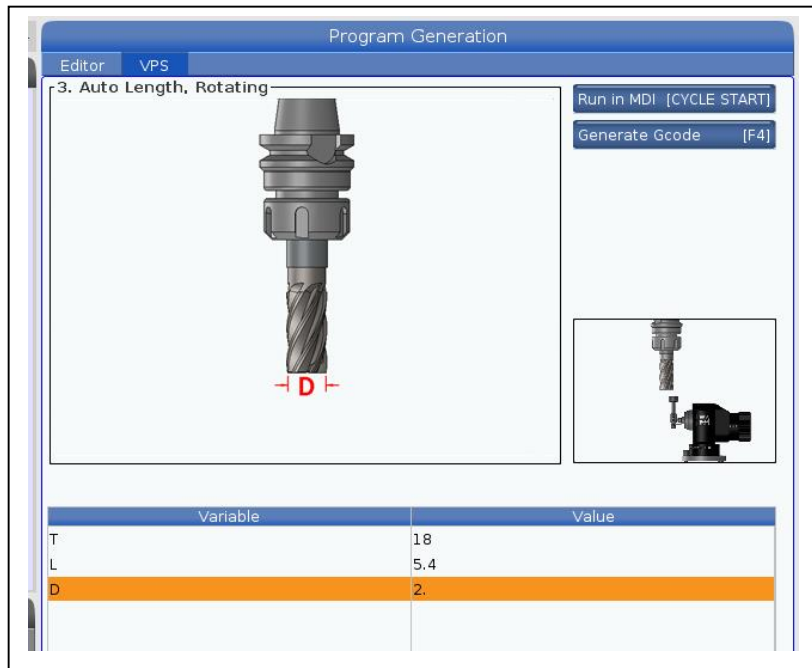
Option 1 would be for like a face mill where the only information that is needed is the tool length. Key in 2 and press enter. In the following picture for tool 18 option 3 is selected.



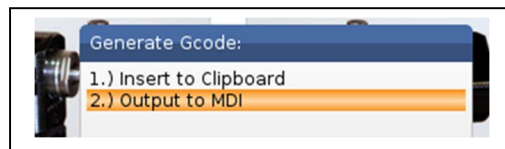
Next press **TOOL OFFSET MEAS KEY**. A pop up appears as below. Highlight [Probe selected tool] and press the enter key.



This gives the VPS page for entering the correct variables. In this tool the tool number is 18, the measured gauge length 5.4 and the diameter of the cutter 2.



After entering the variables then press the (F4) key to create the code.
Highlight **2.0 Output to MDI**. Then press enter.



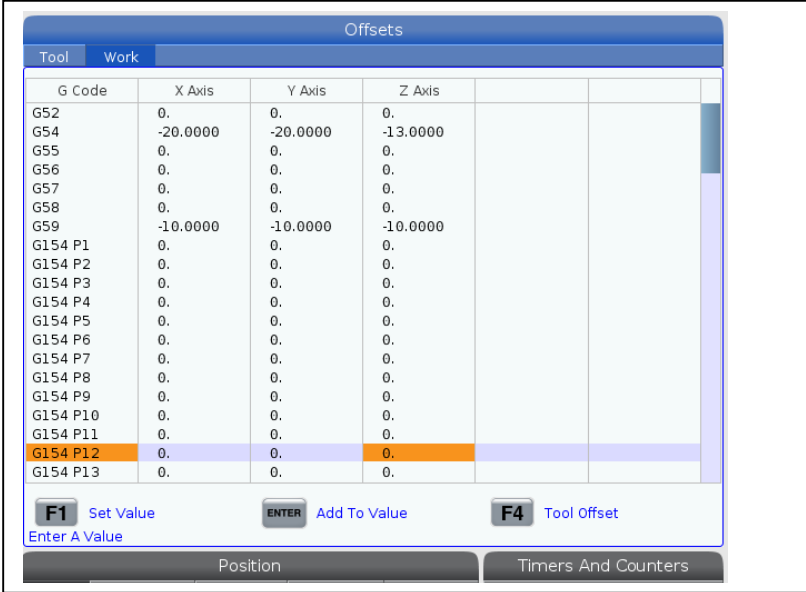
The following code is put into MDI

In the MDI mode press cycle start to run the probe program to set the tool length (H) for tool 18.

```
(3. Auto Length, Rotating);  
(PROBE TOOL HEIGHT, ROTATING);  
(TOOL = 18 );  
(APPROXIMATE LENGTH = 5.4)  
(APPROXIMATE DIAMETER = 2.);  
G00 G90;  
G65 P9023 A23. T18. H5.4 D2.;  
M30;
```

Setting work offsets using VPS thru the Work offset page.

First go to the work offset page by pressing the OFFSET key. If the tool offset page comes just hit the F4 key to toggle to the work offset page. Then highlight the work offset that is desired. Below G154 P12 is highlighted.

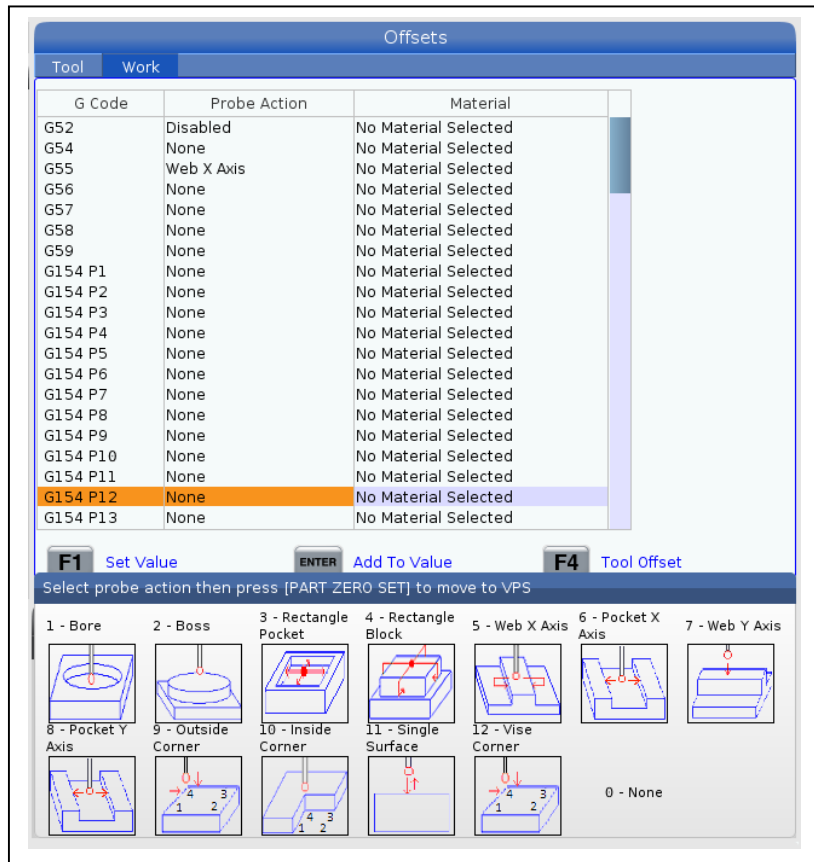


G Code	X Axis	Y Axis	Z Axis
G52	0.	0.	0.
G54	-20.0000	-20.0000	-13.0000
G55	0.	0.	0.
G56	0.	0.	0.
G57	0.	0.	0.
G58	0.	0.	0.
G59	-10.0000	-10.0000	-10.0000
G154 P1	0.	0.	0.
G154 P2	0.	0.	0.
G154 P3	0.	0.	0.
G154 P4	0.	0.	0.
G154 P5	0.	0.	0.
G154 P6	0.	0.	0.
G154 P7	0.	0.	0.
G154 P8	0.	0.	0.
G154 P9	0.	0.	0.
G154 P10	0.	0.	0.
G154 P11	0.	0.	0.
G154 P12	0.	0.	0.
G154 P13	0.	0.	0.

F1 Set Value **ENTER** Add To Value **F4** Tool Offset
Enter A Value

Position Timers And Counters

Next use the right cursor key > to highlight Probe Action column as shown below.



Note there are 12 different options. Pick the option that is desired and key the number in and press **ENTER** key. Then press the **(PART ZERO SET)** key which will bring up the VPS. If 12 Vise Corner is selected the VPS screen for setting the variables will come as below.

Enter the appropriate values by using the **v** down cursor. Then press F4 to generate the code in MDI and run it.

Program Generation

Editor VPS

12. Vise Corner

54 = G54
154.01 = G154P1

Run in MDI [CYCLE START]

Generate Gcode [F4]

Back

Variable	Value
WORK_OFFSETS	154.12
X	0.
Y	0.
Z	0.

Enter the Work Offset Number. (54-59 / 110-129 / 154.01-.99)