Advanced Lathe Training

Training course

www.hfoallendale.com

Table Of Contents

Machine Safety

Personal Protection Equipment

- Be aware of your surroundings while working on a CNC machine not paying attention is the easiest way to get injured
- Don't wear lose clothing and keep hair ties up
  - They can easily get caught in a rotating spindle or moving parts
  - The machine will NOT stop if your hand, arm or leg get stuck inside the machine and can lead to SERIOUS injury
- Appropriate eye and ear protection should be worn while working on the machine
  - Some companies will require you to wear them if you are in the shop
- Wear steel-toed shoes to protect your toes from material being dropped on them

E-Stop Button and Stopping the Machine

- The "Emergency Stop" or "E-stop" button will stop all machine motion
  - This is used in an emergency such as if you see a crash about to happen or a crash has just occurred
  - This will stop all axis motion, the spindle and any other motion in the machine
- The "RESET" will stop a program and return it to the first line
  - The machine will not stop until it finishes specific commands such as:
    - Changing a tool
    - Running a tapping cycle
- The "Feed Hold" button will stop the machine axis motion but not the spindle
  - This is good in a non-emergency situation and would like to check where the tool is
    - (Note: This can leave marks where the tool is stopped)
  - This can help avoid breaking a tool that is mid-cut

Warning Labels and Decals

Symbol	Description
	**Danger:** this symbol means that there is a condition or situation that ***will cause death or severe injury*** if you do not follow the given instructions
	**Warning:** This symbol means that there is a condition or situation that ***will cause moderate injury*** if you do not follow the instructions given
	**Caution:** This symbol means that ***minor injury or machine damage could occur*** if you do not follow the instructions given. You may also have to start a produce over if you do not follow the instructions in a caution statement
	**Note:** This symbol means that the text gives ***additional information, clarification or helpful hints***

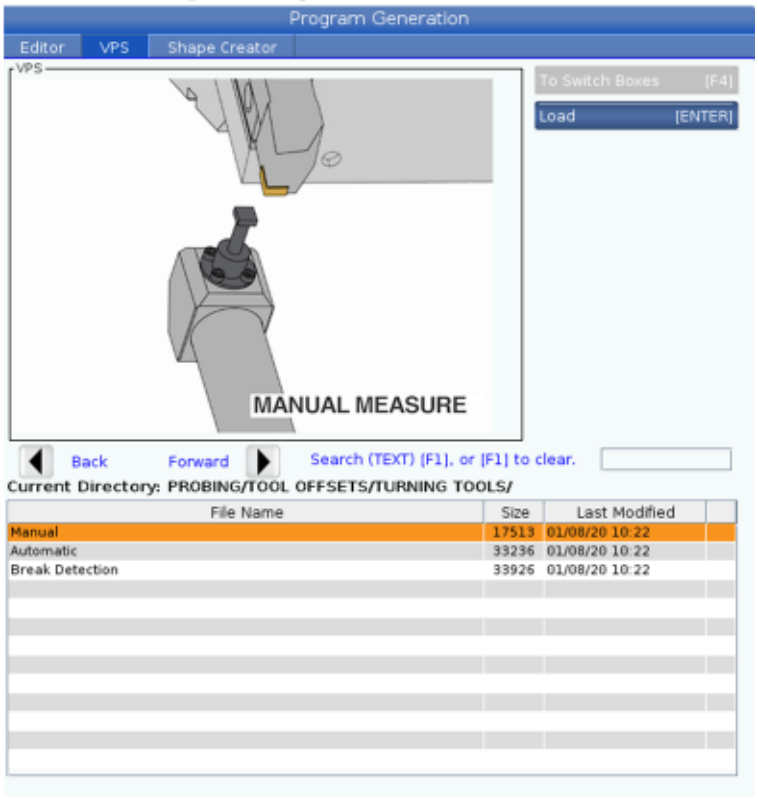
Safe House Keeping

- Be sure to clean coolant spills as soon as they are spotted
- Be sure to keep hands clear of the Auger shaft and the Chip Conveyor both inside and outside the machine
- Be very careful when cleaning chips, they are generally very sharp and will cut your hands
- Additionally, be sure to stay clear of the spindle, tool cutting edges are very sharp
- Never use your fingers to check if a tool is sharp and never handle a tool on the cutting edge
- There should be a safety manual in your shop on how your machine shop deals with specific material

Basic Tool Pre-setter Functions

Getting to the Probing Page

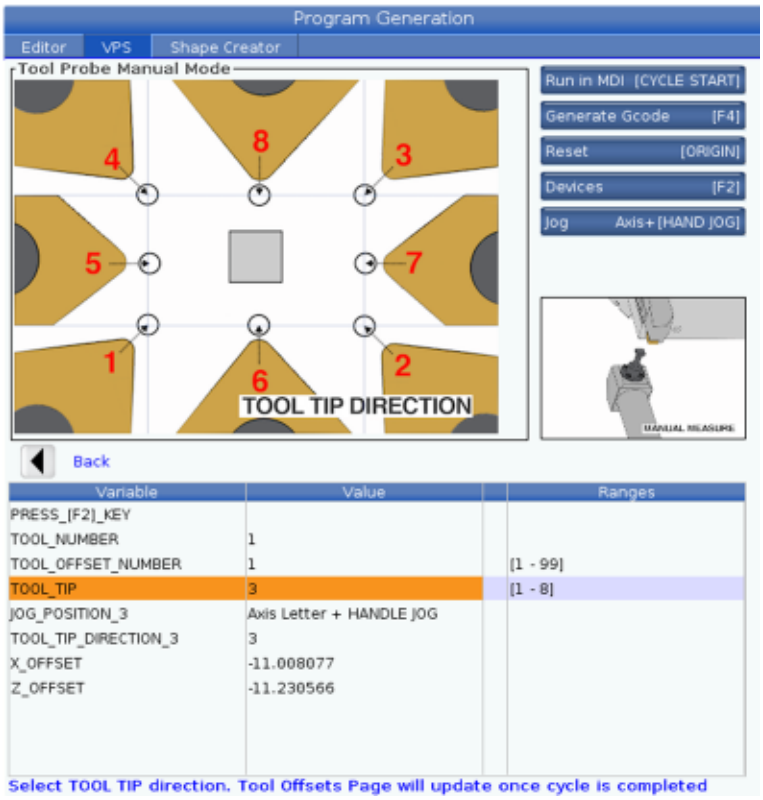
1. Go to EDIT
2. Cursor up to VPS
3. Cursor to the right on PROBING
4. Cursor right on TOOL SETTING
5. Cursor right on TURNING TOOLS



Probing Turning Tools

By selecting MANUAL you will be brought to a screen to touch of you tool

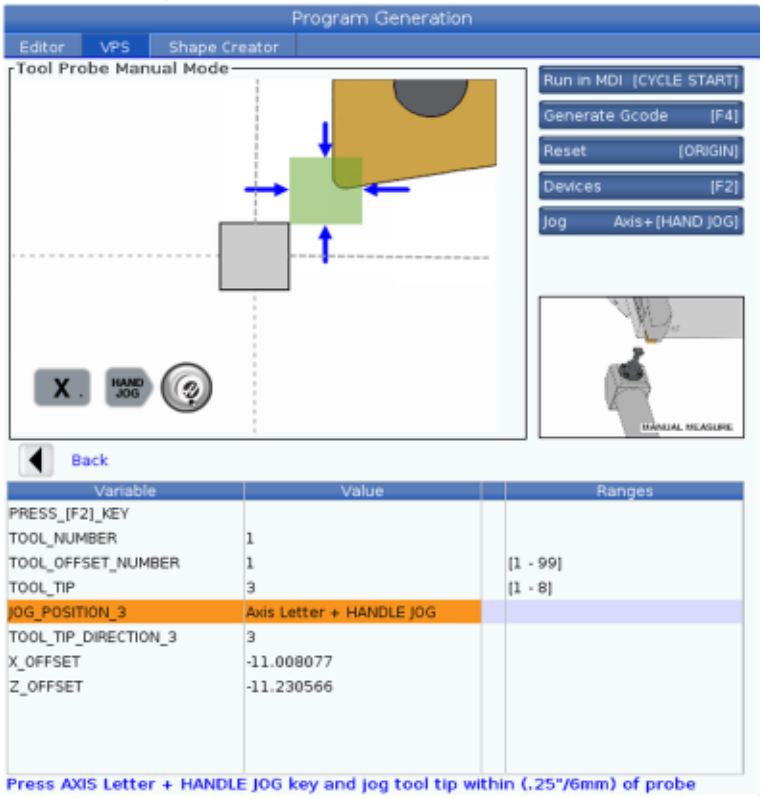
1. PRESS_[F2]_KEY: This prompt is used to lower the tool probe arm by pressing “F2”. Be sure that nothing is in the probe arm path
2. TOOL_NUMBER: This will automatically be generated base on the tool in the active cutting position
3. TOOL_OFFSET_NUMBER: This will automatically be generated based on the tool in the active cutting position. It can be change to another offset as required
4. TOOL_TIP: This is the direction of you tool (3 is the direction of a standard tool)



Probing Turning Tools

By selecting MANUAL you will be brought to a screen to touch of you tool

5. JOG_POSITION_#: This is the position you will have to jog your tool to in order to probe it
6. TOOL_TIP_DIRECTION_#: Will let you know the order in which the tool will be touched off
7. X_OFFSET: This is the previous offset that is in this Tool Offset page
8. Z_OFFSET: This is the previous offset that is in the Tool Offset page

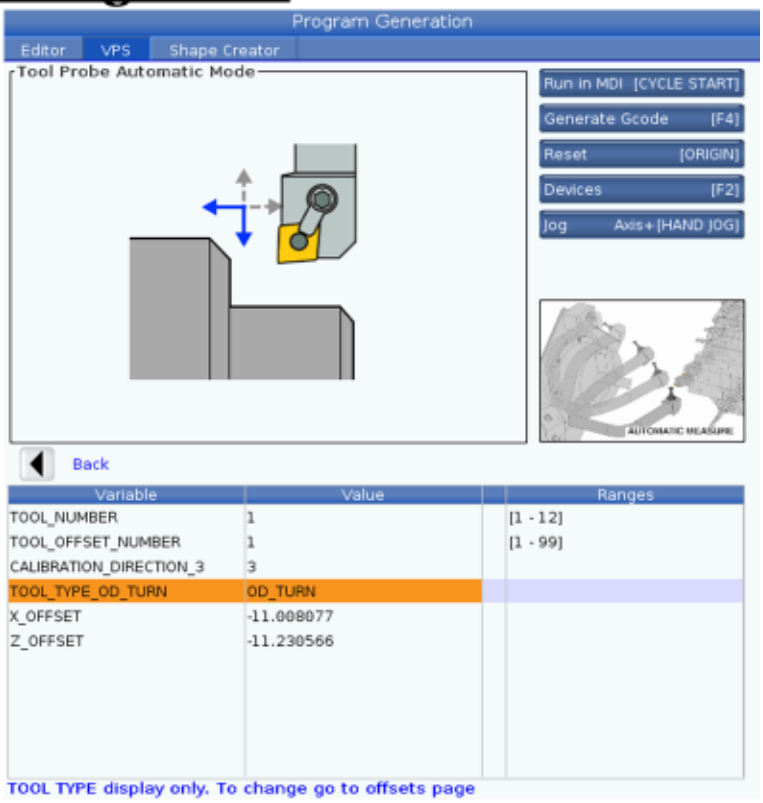


Probing Turning Tools

By selecting AUTOMATIC you will be brought to a screen to touch of you tool

- This should only be used if you flipping in insert

1. TOOL_NUMBER: This will automatically be generated base on the tool in the active cutting position
2. TOOL_OFFSET_NUMBER: This will automatically be generated based on the tool in the active cutting position. It can be change to another offset as required
3. CALIBRATION_DIRECTION_#: Will let you know the order in which the tool will be touched off
4. TOOL_TYPE: This is the type of tool being probed and is set in the OFFSETs page

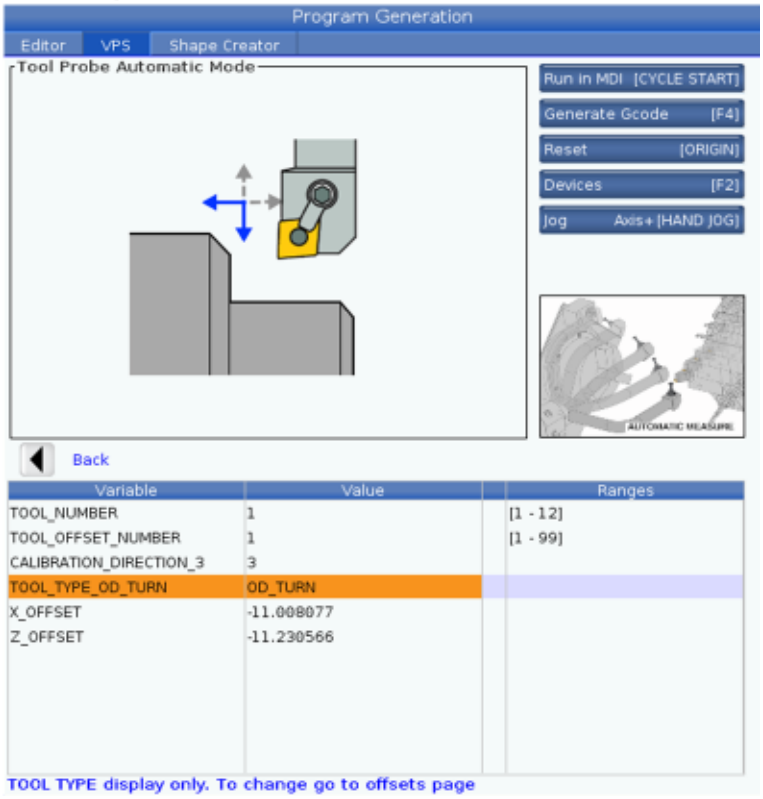


Probing Turning Tools

By selecting AUTOMATIC you will be brought to a screen to touch of you tool

- This should only be used if you flipping in insert

5. X_OFFSET: This is the previous offset that is in this Tool Offset page
6. Z_OFFSET: This is the previous offset that is in the Tool Offset page

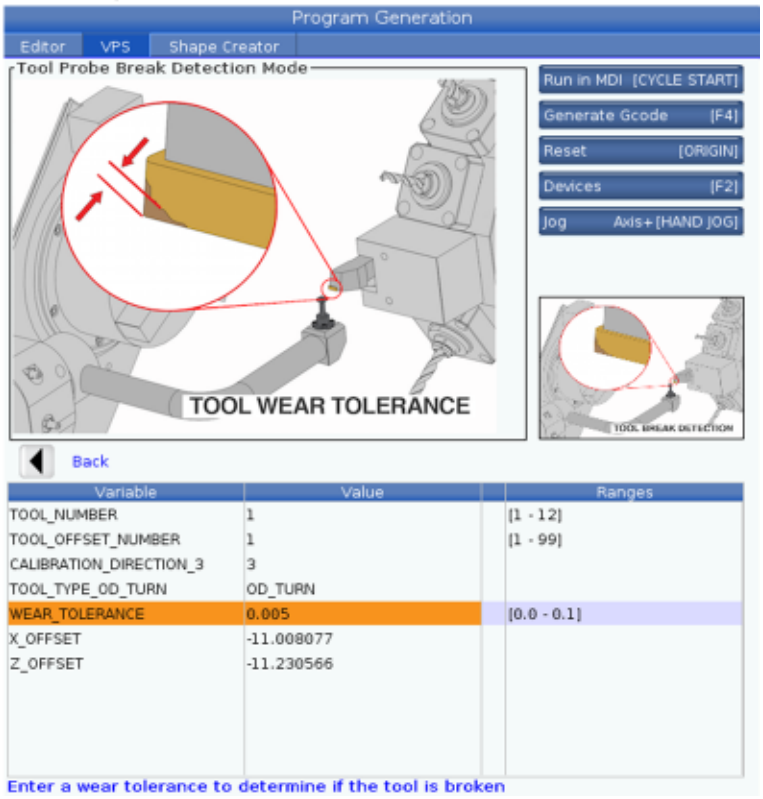


Probing Turning Tools

By selecting AUTOMATIC you will be brought to a screen to touch of you tool

- This should only be used if you flipping in insert

1. TOOL_NUMBER: This will automatically be generated base on the tool in the active cutting position
2. TOOL_OFFSET_NUMBER: This will automatically be generated based on the tool in the active cutting position. It can be change to another offset as required
3. CALIBRATION_DIRECTION_#: Will let you know the order in which the tool will be touched off
4. TOOL_TYPE: This is the type of tool being probed and is set in the OFFSETs page

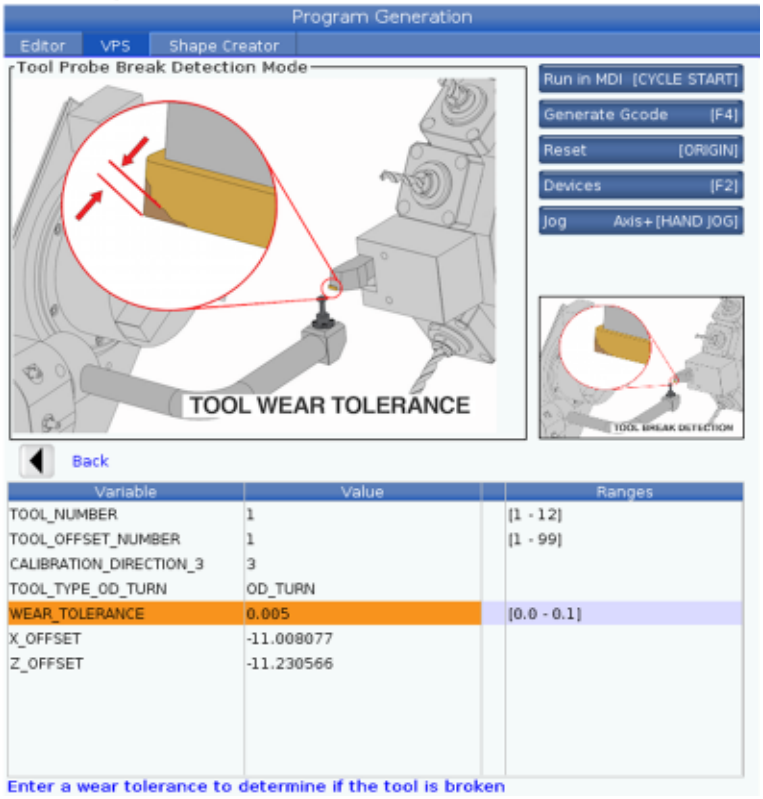


Probing Turning Tools

By selecting BREAK DETECTION you will be brought to a screen to touch of you tool

- This is used to check if a tool is broken or chipped

5. WEAR_TOLERANCE: This is the allowable variation in measurement a tool can have. This measurement is in reference to the number in the tool offset page
6. X_OFFSET: This is the previous offset that is in this Tool Offset page
7. Z_OFFSET: This is the previous offset that is in the Tool Offset page



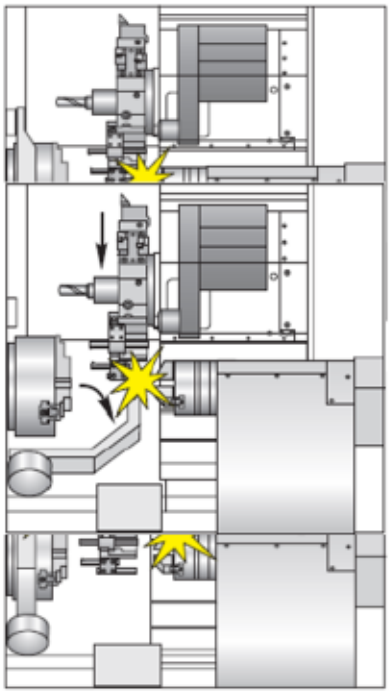

Dual Spindle
Operations

Dual Spindle (DS) or Secondary Spindles

- A DS lathe is a lathe with a second spindle mounted where the tail stock would normally be mounted
- The secondary or sub-spindle is the B axis with the positive direction is the same as the Z-axis
- Clearance becomes very important when you introduce a second spindle

Secondary Spindle Clearance

- When setting up a machine with a secondary spindle it is important to:
  - Check the clearance of the main and sub-spindle
  - Check for clearance on long sub-spindle tools
  - Check live tools for clearance
  - If there is a parts catcher, be sure to check for clearance on both spindles
  - Be sure to check for clearance when lowering Tool Presetter



Live Tooling

Common M/G Codes for Secondary Spindles

M/G Code	Code Description
M110	Secondary Spindle (Chuck Close)
M111	Secondary Spindle (Chuck Open)
M114	Secondary Spindle (Brake On)
M115	Secondary Spindle (Brake Off)
M143 (P)	Secondary Spindle Forward (P is the spindle speed)
M144 (P)	Secondary Spindle Reverse (P is the spindle speed)
M145	Secondary Spindle Stop
M119 R0.	Secondary Spindle Orientation (R is the orientation in degrees)
G04 P1.	Dwell (P is time of dwell)
G14	Secondary Spindle Swap
G15	Secondary Spindle Swap Cancel
G199	Engage Sync Spindle Control
G198	Disengage Sync Spindle Control

Spindle Sync Sample (G199)

G97 M03 S500 Start Main Spindle
G199 Sync Spindles

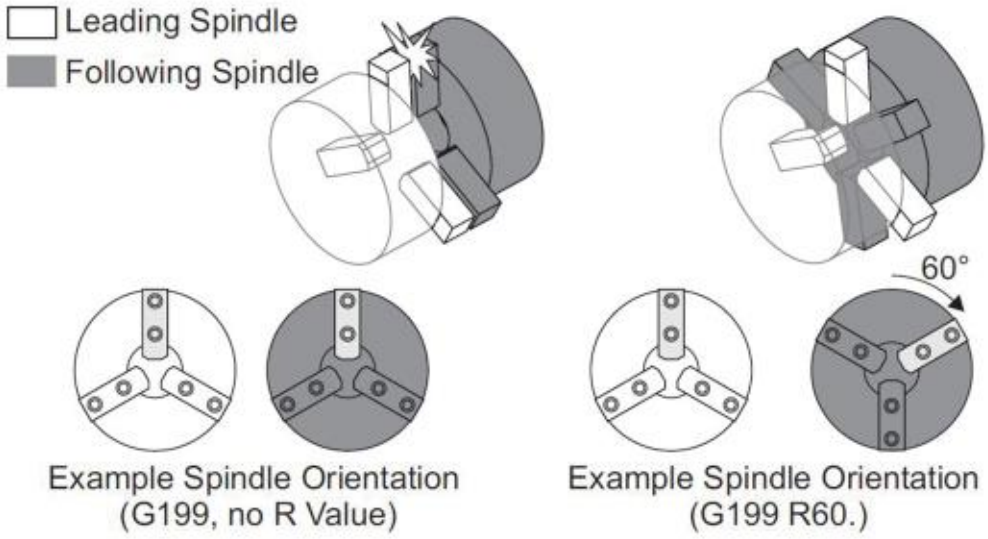
- This is often used to cut off parts or to hand off parts from one spindle to another
  - To end spindle sync you must use G198
  - RESET will NOT un-sync the spindles
- When transferring parts
  - Use a slow RPM (ex. S50) [Spindles can be stopped but it is suggested that they are slowly spinning]
  - Bring the B axis close to the end of part
  - Use spindle air or coolant to ensure there are no chips that will be gripped
    - Give time for spindles to sync
  - Once the B axis it in position you may clamp the secondary spindle (M111)
  - Once spindles are sunk and clamped a higher spindle speed may be used to for cutting off part

Spindle Sync Sample (G199)

```
O0005
G198          Desync – Part should not be clamped on both ends
G97 M03 S50   Start main spindle
G97 M144 S50  Start secondary spindle
G04 P1.       Dwell for 1s
G199          Sync Spindles
G04 P1.       Dwell for 1s
              Run program
G198          Desync – Part should not be clamped on both ends
M05           Stop main spindle
M145          Stop secondary spindle
M30
```

Spindle Sync Sample (G199 R0.)

- Running with R command in the G199 will offset the spindle being synchronized



Example Spindle Orientation (G199, no R Value)

Example Spindle Orientation (G199 R60.)

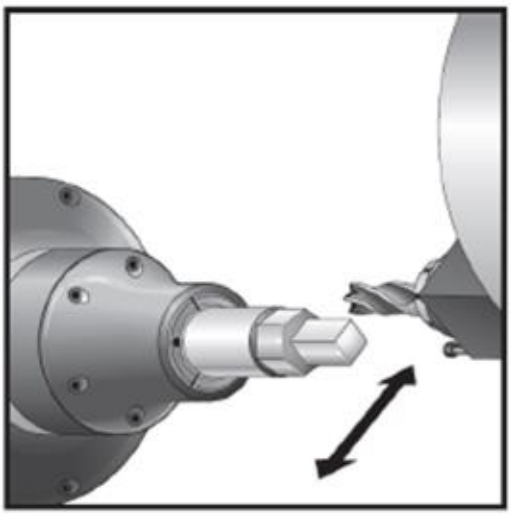
Machining on Sub - Spindle Sample (G14)

```
                G14   Secondary spindle swap
G97 M03 S1000
                G99   Feed per rev.
G01 Z-1. F0.02
                G15   Cancel secondary spindle swap
```

- The G14 command allows you to cut on the secondary spindle while programming as if you were to cut on the main spindle
- All operations should start with a G14 and end with a G15
  - Alarm 186 Spindle Not Turning is likely because a G14 was forgotten

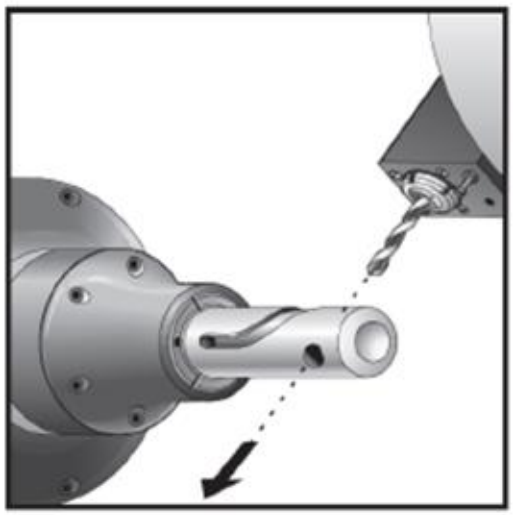
Live Tool Concepts

Axial Tools vs. Radial Tools



Axial Tool

AXIAL (Face-Working) Tool

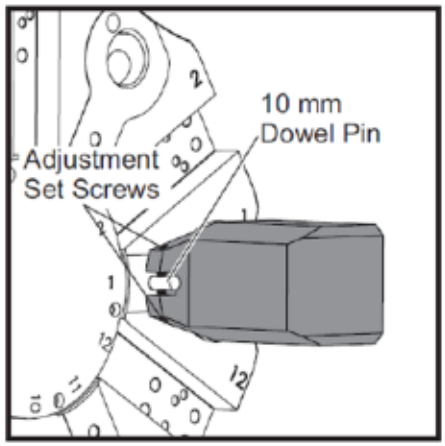


Radial Tool

RADIAL (Cross-Working) Tool

Setting Up Radial Tool Holders

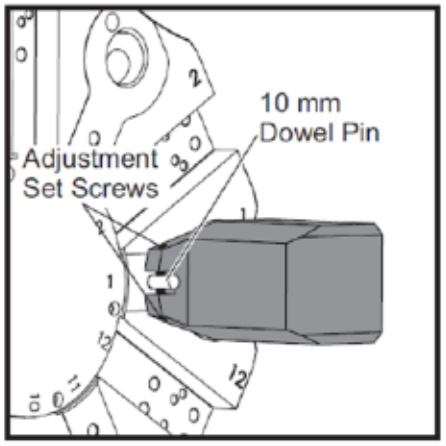
1. Install 10mm Ground Pin on the turret
2. Before mounting live tool remove and Thoroughly clean VDI bold. Also clean inside of the VDI holder on the turret
3. Apply thin film of way lube on the internal contact surfaces of the Radial Tool
4. Mount the Radial Tool and snug the adjustment set screws against the dowel pin at a visually-even and centered position
5. Snug VDI Allen bolt to allow for some movement and adjustment of the tool. Ensure that the back face of the tool holder is flush with the face of the turret
6. Position the Y-axis at zero



Adjust Alignment with Set Screws

Setting Up Radial Tool Holders

7. Install a dowel or gauge pin on the holder like you would install the cutting tool
8. Make sure the tool sticks out at least 1.25". This will be used to run the indicator across it to insure parallelism to X-axis
9. Set indicator with a magnetic base on a rigid surface
10. Position the indicator tip on the end point of the pin and zero the indicator dial
11. Sweep the indicator along the pin to measure the parallelism between the pin and X-axis
12. Adjust the set screws until the tool is aligned properly and parallel to the X-axis
13. Tighten VDI allen bolt to recommended to torque (35-45ft-lbs)



Adjust Alignment with Set Screws

Setting Up Axial Tool Holders

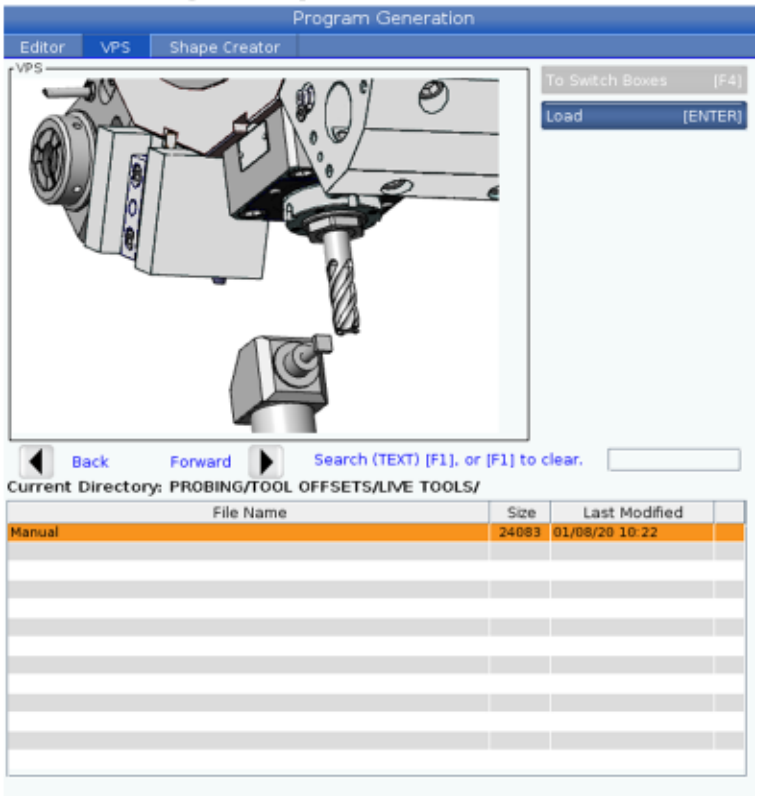
1. Before mounting live tool remove and thoroughly clean VDI bolt. Clean the inside of the VDI cavity on the turret
2. Apply thin film of way lube on the internal contact surfaces of the Axial Tool
3. Mount Axial Live tool aligning on alignment pin. Snug the VDI bolt. Ensure the back face of the tool holder is flush to the face of the turret
4. Tighten the VDI bolt to recommended torque (35-45ft-lbs)

Setting up Tools in Live Tooling Holders

1. Insert the tool into an ER-32 collet.
2. Thread the ER-32 collet nut insert into the collet housing over the tool bit and ER-32 collet
3. Place spanner wrench over the pin of the live tool holder and lock it against the collet
4. Engage the teeth of the collet wrench and tighten

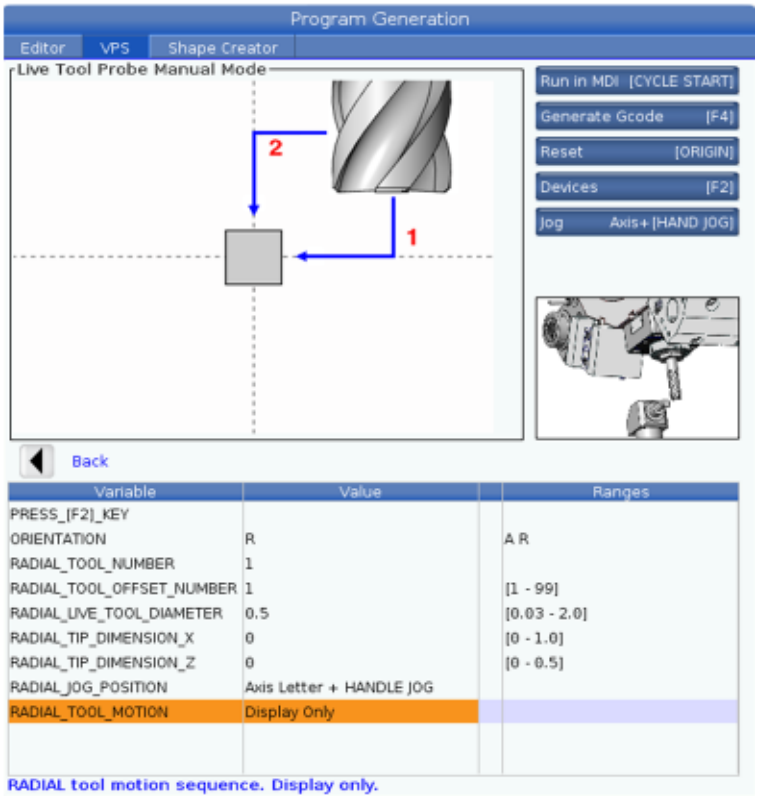
Getting to the Probing Page

1. Go to “EDIT”
2. Cursor up to “VPS”
3. Cursor to the right on PROBING
4. Cursor right on TOOL SETTING
5. Cursor right on LIVE TOOLS
6. Cursor right on MANUAL



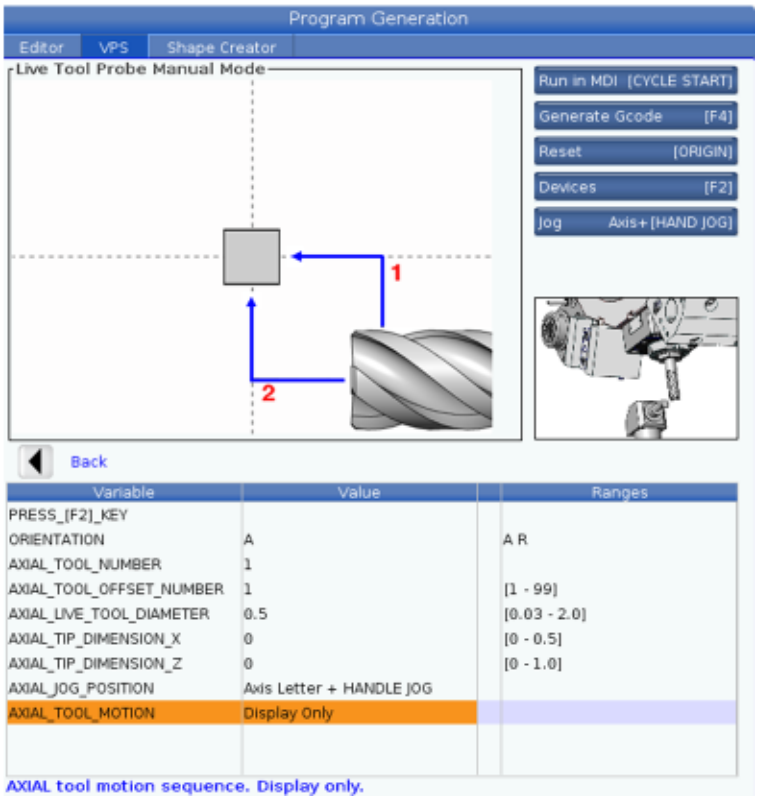
Radial Live Tool Setting

- Pressing “F2” the probe arm will go down
- “ORIENTATION” is the tool, in this case you will input “R” for a Radial tool
- RADIAL_TOOL_NUMBER is the tool number
- RADIAL_TOOL_OFFSET_NUMBER is the tool offset you’d like to set
- RADIAL _LIVE_TOOL_DIAMETE is the diameter of the tool you would like to probe
- RADIAL_TIP_DIMENSION_X is the tool radius
- RADIAL_TIP_DIMENSION_Z is the distance to hit the max diameter of the tool
- RADIAL_JOG_POSITION is where you need to place the tool to run
- RADIAL_TOOL_MOTION shows the order in which the tool will be probed



Axial Live Tool Setting

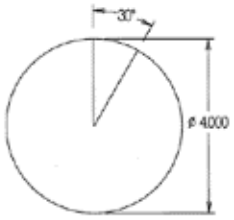
- Pressing “F2” the probe arm will go down
- “ORIENTATION” is the tool, in this case you will input “A” for a Axial tool
- AXIAL_TOOL_NUMBER is the tool number
- AXIAL_TOOL_OFFSET_NUMBER is the tool offset you’d like to set
- AXIAL _LIVE_TOOL_DIAMETE is the diameter of the tool you would like to probe
- AXIAL_TIP_DIMENSION_X is the tool radius
- AXIAL_TIP_DIMENSION_Z is the distance to hit the max diameter of the tool
- AXIAL_JOG_POSITION is where you need to place the tool to run
- AXIAL_TOOL_MOTION shows the order in which the tool will be probed



C-Axis Info

- When programming on the C-axis the feed rate is in inches per min
  - The feed rate is determined by the diameter entered in SETTING 102 (C-axis Diameter)
    - To change unit to degrees/min the lathe must be turned to metric and setting 102 set to 114.5
      - This is the same as $\frac{360}{\pi}$

Calculating Feed Rate in Degrees/min on C-axis



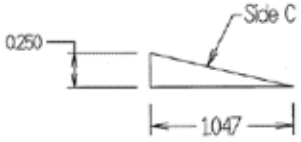
1. Calculate radial move in inches

Circumference of a circle = Diameter(D) * Π

= 4 *Π = 12.566

Distance travelled over 30° = $\frac{30}{360} * 12.566 = 1.047$

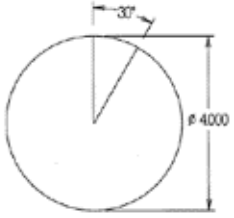
2. Movement in Z, if any



3. Calculate total movement (Side C)

C = $\sqrt{A^2 + B^2} = \sqrt{.25^2 + 1.047^2} = 1.0762$

Calculating Feed Rate in Degrees/min on C-axis



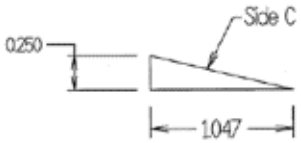
4. Additional Calculations

RPM = $\frac{3.82*(S[ft/min])}{Diameter}$

IPM = FPT * T * RPM

Minutes to move C = $\frac{C}{IPM}$

Feed Rate (degrees/min) = $\frac{Degrees\ traveled}{Time\ in\ minutes}$



Live Tooling G-Codes

G17	Axial or face working cycles (X-Y plane) *Only works when using a G112
G18	Axial or face working cycles (X-Z plane)
G19	All cross working or radial cycles
G98	Feed rates (in/min)
G99	Feed rates (in/rev) Used for tapping cycles
G112	Converts Cartesian coordinate system to Polar system. This allows a program in X-Y coordinates to be read as a X-C interpolation

Live Tooling M-Codes

M14	Clamp Main Spindle
M15	Unclamp Main Spindle
M19	Orient Spindle (Optional) [Ex. M19 P90]
M119	Orient Sub-Spindle
M133	Live Tool Drive Forward
M134	Live Tool Drive Reverse
M135	Live Tool Drive Stop
M154	C-Axis Engage: Turns the C-axis motor on
M155	C-Axis Disengage: Turns C-axis motor off

Axial Live Tool Canned Cycles

Canned Cycle	Cycle Description	Plane Selection	Feed Rate
G81	Drill	G18	G98
G82	Drill with Dwell	G18	G98
G83	Drill with Peck	G18	G98
G95	Rigid Tap	G18	G99
G186	Rigid Tap Left	G18	G99
X-C Axis Milling	Manual Slots	G18	G98
X-Y Axis Milling	Manual Radius	G17	G98
G112	Cartesian to Polar	G17	G98

Axial Live Tool Canned Cycle Example

Drilling with G81

```
O0051
T101
G18          Calling work plane
G54 G00 X3.0 Z0.1
G98          Feed in in/min
M154         C-Axis Engage
C0.0         Rotate C-axis to 0 degrees
M133 P2000   Turn live tool drive forward at 2000RPM
G81 Z-0.8 F40.  Axial Drill Cycle
C120.        Rotate C-axis to 120. degrees
C240.        Rotate C-axis to 240. degrees
G00 G80 Z0.1
M155         C-axis Disengage
M135         Live Tool drive stop
G53 X0.
G53 Z0.
M30
```

Axial Live Tool Canned Cycle G112 Example

M154	C- Axis Engage
G28 H0	Unwind C-axis
M133 P2650	Live Tool Drive on at 2650 RPM
G59 G00 Z0.15 C0.	
G59 G00 X0. Y0.	
G98	Feeding in In/min
G17	Use X-Y Plane
G112	X-Y to X-C interpolation
G01 Z-0.0075 F20.	
G41 X.5 F11.5	
Y0.25	
G03 X0.25 Y0.5 R0.25	
G01 X-0.25	
G03 X-0.5 Y0.25 R0.25	
G01 Y0.25	
G03 X0.5 Y-0.25 R0.25	
G01 X0.25	

Axial Live Tool Canned Cycle G112 Example

G03 X-0.5 Y0.25 R0.25	
G01 Y0.25	
G03 X0.5 Y-0.25 R0.25	
G01 X0.25	
G03 X0.5 Y-0.25	
G01 Y0.	
G40 X0.	
G00 Z0.25 M00	
G113	Cancel G112
G18	X-Z Plane
G99 M135	
G53 G00 X0.	
G53 G00 Z0.	
M30	

Radial Live Tool Canned Cycles

Canned Cycle	Cycle Description	Plane Selection	Feed Rate
G75	Drill with Peck	G19	G98
G241	Drill	G19	G98
G242	Drill with Dwell	G19	G98
G243	Drill with Peck	G19	G98
G195	Radial Rigid Tap	G19	G99
G196	Radial Rigid Tap Left	G19	G99

Radial Live Tool Canned Cycle Example

Drilling with G81

```
G54
G00 G53 Y0.   Home Y-axis
G00 G53 X0. Z-7.
T303
M154   Engage C-axis
M133 P2500   Live Tool Drive at 2500 RPM
G19
G98   Feed in In/Min
G00 X5. Z-0.75 Y0.
G243 X2.1 Y0.125 Z-1.3 C35. R4. Q0.25 F20.
X1.85 Y-0.255 Z-0.865 C-75. Q0.25
G00 G80 Z1.
M135   Stop Live Tool Drive
G00 G53 X0. Y0.
G00 G53 Z00
M00
```

Resources

Haas Automation, Inc YouTube Channel

HAASCNC.com