




Welcome to Trident Machine Tools

Live Tool Lathe Training





# Haas live tool lathe training

- This 1/2 day course is designed to familiarize the user with the theory, set-up and basic programming of live tooling on a Haas CNC lathe.

# Schedule

- Introductions
- Live tooling types
- Turret configuration
- Live tooling offsets
- Break
- Live tooling specific G and M codes
- Radial live tooling canned cycles
- Axial live tooling canned cycles

# CNC Slant Bed Lathes

- Most CNC slant bed lathes consist of two axis.
  - X – controls diameter
  - Z – controls depth
- These machines can perform simple turning, drilling, threading, boring, etc.
- All work must be done on the centerline of the spindle.



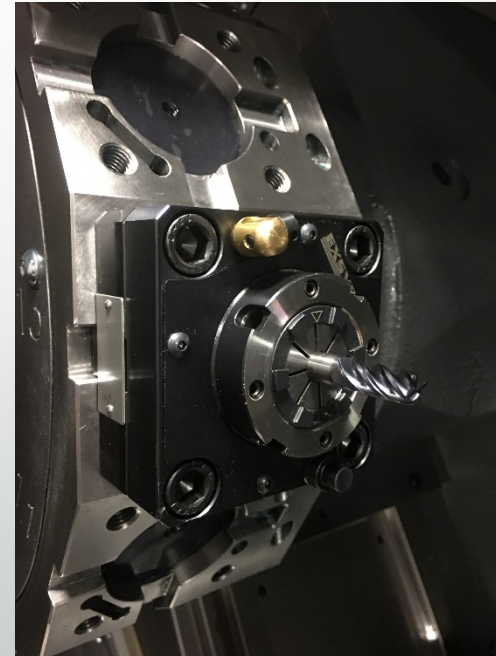
# CNC Lathes with Live Tooling

- Live tooling is the ability to add milling operations to a lathe.
  - Done by adding driven tools that can be bolted on the turret of the CNC lathe.
- Tools will not be as powerful as a CNC mill, but reduce the number of set-ups needed to complete a part.
- Tools will enable turning and milling in one set-up.



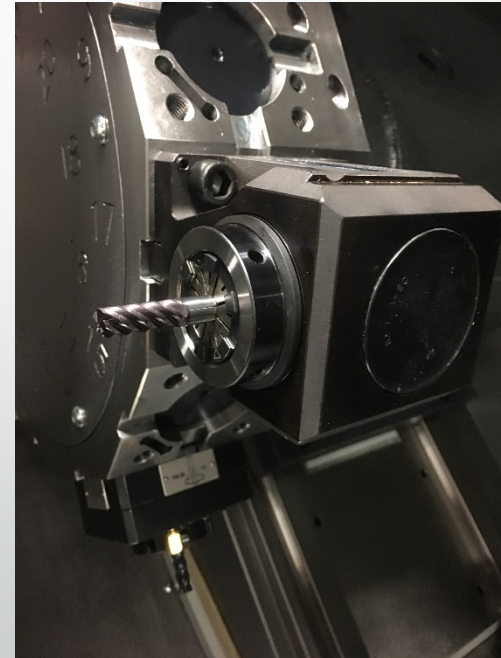
# Types of Live Tooling

- **Radial tools**
  - Hold the cutting tool parallel to the face of the turret.
  - Designed to work on the diameter of the part performing operations:
    - Milling
    - Drilling
    - Tapping



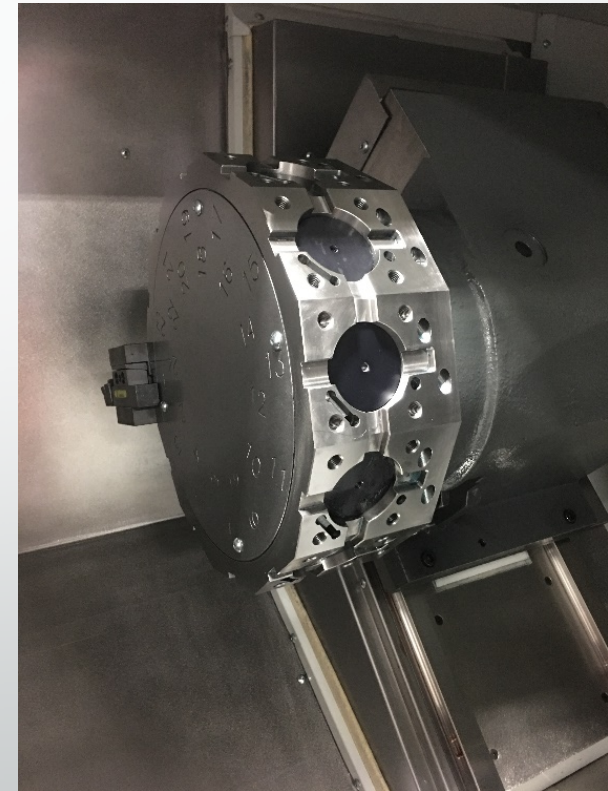
# Types of Live Tooling

- **Axial tools**
  - Hold the cutting tool perpendicular to the face of the turret
  - Designed to work on the face of the part performing operations
    - Milling
    - Drilling
    - Tapping



# Base Mount Turret (BMT)

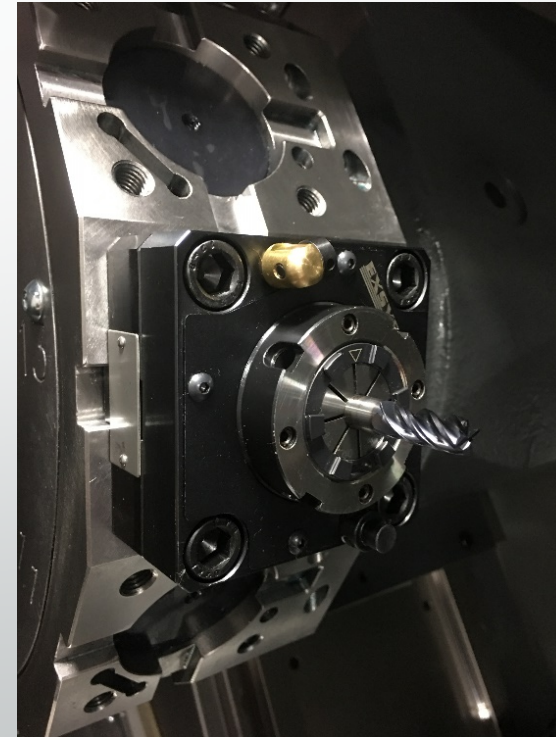
- The base mount turret provides extra-rigid mounting for turning and boring tools to improve cutting performance, and offers additional tool clearance when working with a tailstock.
- All tools (static or driven) need a holder installed to be attached to the turret.
- When loading live tooling, the tools should be installed in the active tool position.





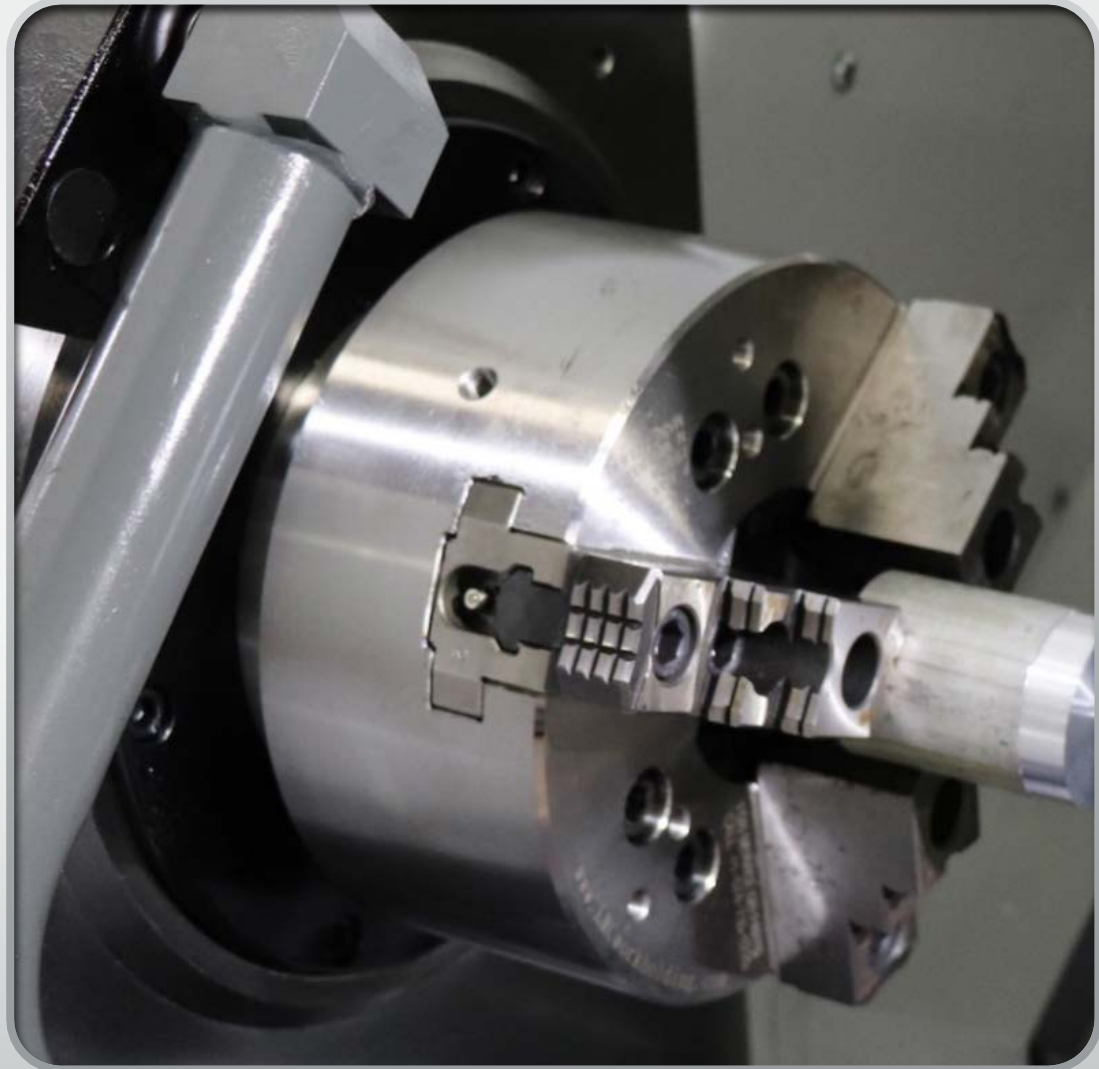
# Types of Live Tooling

- Options:
  - Typical live tooling holders have a 1:1 gear ratio.
  - The following holders can be purchased:
    - High Torque- 1.5: 1 ratio
      - Reduced RPM, increased torque
    - High Speed - 1:3 ratio
      - Increased RPM, Decreased Torque
    - Compact Size
    - Adjustable Angle



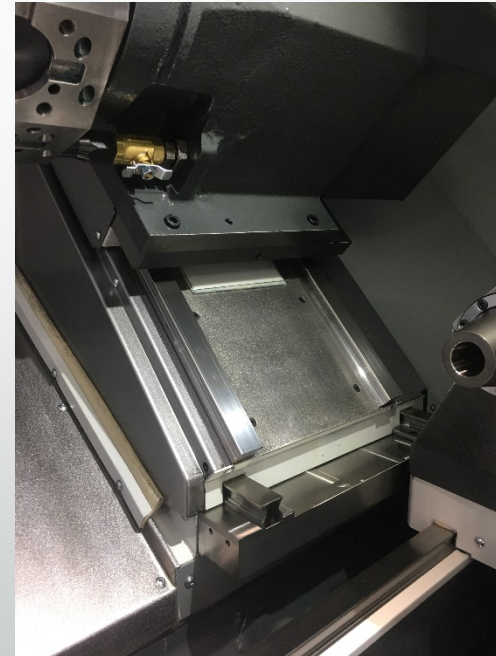
# C Axis

- The lathe spindle becomes a rotary to be utilized by the live tooling.
  - This is a true rotary which can perform simultaneous cutting.
- C axis enables positioning for live tooling .



# Y axis

- With live tooling and a C axis rotation, machinists can make cuts on the centerline of the part.
- ***What if we want to make cuts off the centerline of the part?***
  - The Y axis can be added to the lathe to move the turret perpendicular to the x axis.
    - The Haas lathes have the ability to travel +/-2.000" from spindle centerline.
- This allows features like the following to be machined:
  - Flats
  - Bolt circles on the diameter
  - Holes of centerline



# Axial Tools

- The driven tools on the Haas lathes are mounted in the VDI tool holders



**Tool Change Without  
Removing Adapter**

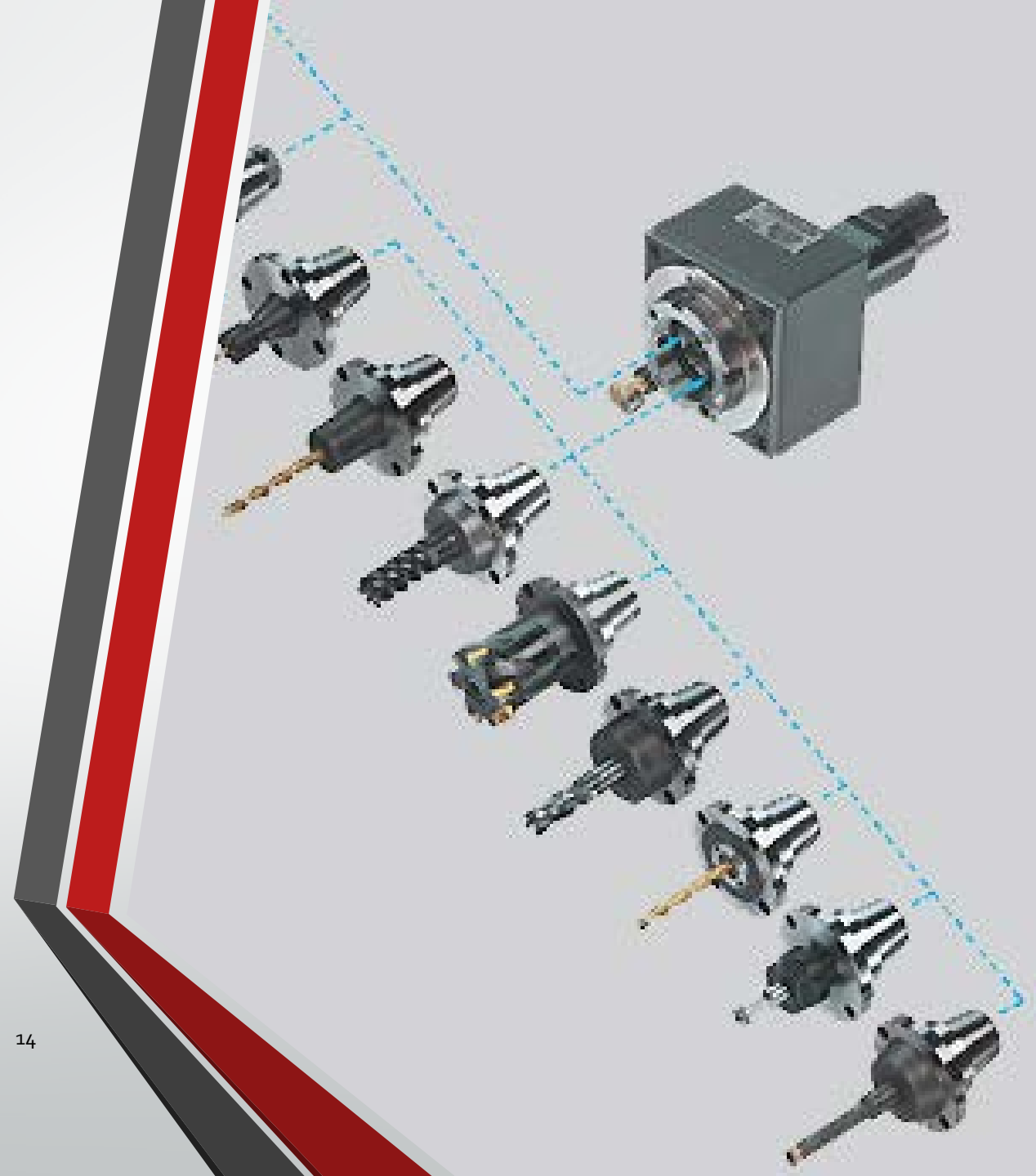


# Axial Tool Offsets

- **Axial tools** are offset similar to turning tools:
  - The Z offset can be set by touching off the face of the part.
  - The X offset can be set by touching the tool off on the outside of the part, then adding the diameter of the tool and part to the offset.

# Radial Tools

- Like axial tools, **radial tools** are set-up in the turret.
- These live tooling holders have many types of holders that can be used:
  - ER collets
  - Endmill holders
  - Shellmill holders
  - Tap holders



# Radial Tools

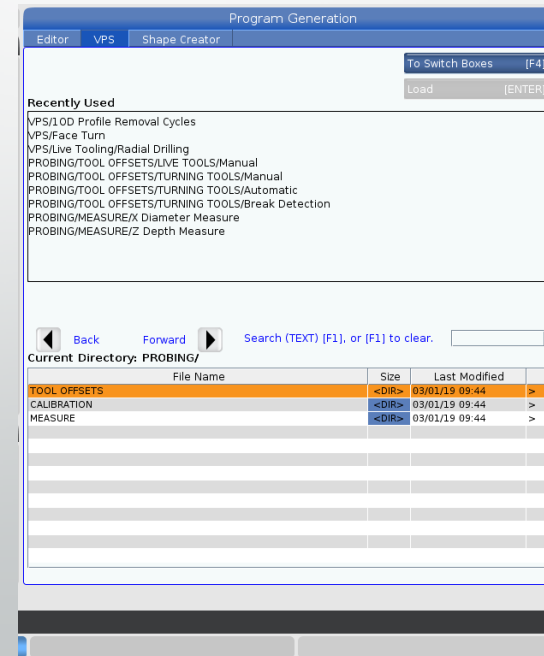
- Radial tools are offset opposite the axial tools:
  - **To set Z offset:** touch the diameter of the cutting tool off the face of the part.
  - **To set X offset:** set end of the endmill off the outer diameter of the part, then subtract the diameter from the offset.
  - The coolant line of the tool should also be set while doing the tool offsets.



Coolant should be adjusted here when setting the tool up.

# Live Tooling Geometry Offsets

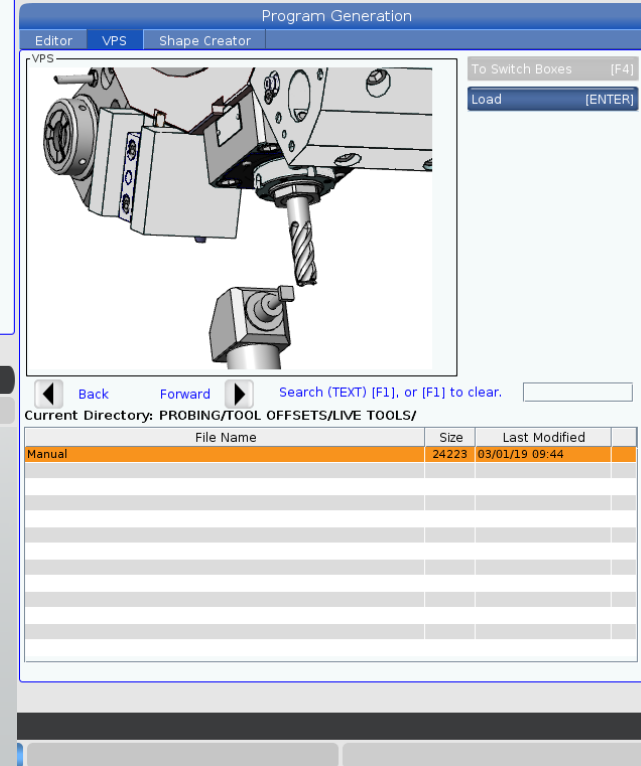
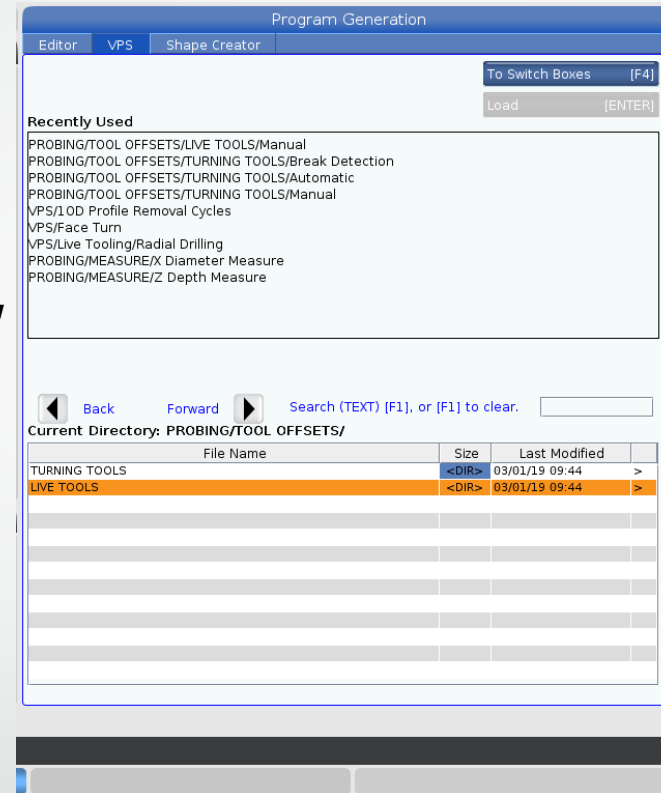
- Both radial and axial live geometry offsets can be set with the tool setter.
- Like regular turning tools, VPS can be used to set the tool offsets.





# Live Tooling Geometry Offsets

- After selecting VPS, enter the live tools option, then enter manual.
- Manual tool setting is the only option for setting live tool offsets.



# Live Tooling Geometry Offsets

- After entering the manual offset page, the orientation of the tool needs to be established.
  - A- axial tool offset
  - R- radial tool offset

MDI N451

```
(OD Profile Removal Cycles);
( SAFETY LINE BELOW );
G00 G54 G18 G40 G80 G97 G99;
( TOOL = 5 / OFFSET = 5 );
( MAXIMUM SPINDLE RPM = 1800 );
( SURFACE SPEED SPEED = 500 );
( DEPTH PER PASS = 0.05 );
( WORK OFFSET = 54);
(OD STOCK REMOVAL CYCLE);
T505;
G54;
G50 S1800;
G96 S500 M03;
G00 Z0.1;
M08;
G00 X2.6 Z0.1;
G71 P144 Q145 D0.05 U0.01 W0.003 F0.01;
N144 G42 X1. Z0.1;
G01 X1. Z0.1;
G01 X1. Z-1.;
G01 X1.25 Z-1.;
G01 X1.25 Z-1.5;
G02 X1.75 Z-1.75 R0.25;
G01 X2.5 Z-1.75;
N145 G01 G40 X2.6 Z-1.75;
G00 X2.6 Z0.1;
M09;
G00 G53 X0.
```

Main Spindle

STOP

Overrides

Feed: 100%  
Spindle: 100%  
Rapid: 100%

Spindle Speed: 0 RPM  
Spindle Power: 0.0 KW  
Surface Speed: 0 FPM  
Chip Load: 0.00000 IPT  
Feed Rate: 0.0000 IPR  
Active Feed: 0.0000 IPR

Spindle Load(%) 0%

Program Generation

Live Tool Probe Manual Mode

Axial = A

Radial = R

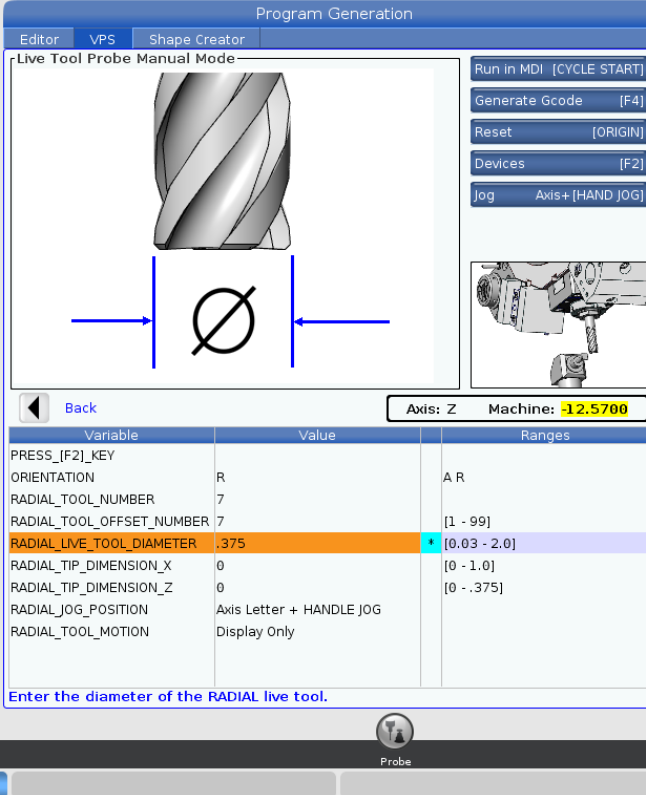
Variable	Value	Ranges
PRESS_[F2]_KEY		
ORIENTATION	?	A R

SELECT Live Tool Orientation. AXIAL = A RADIAL = R

Input:

# Live Tooling Geometry Offsets

- In this case, radial has been chosen. This brings up the prompt fields specific to the radial tool.
  - The tools diameter and tip information will need to be entered.
  - If these are improperly entered, the tools offsets could be set wrong.



Program Generation

Editor VPS Shape Creator

Live Tool Probe Manual Mode

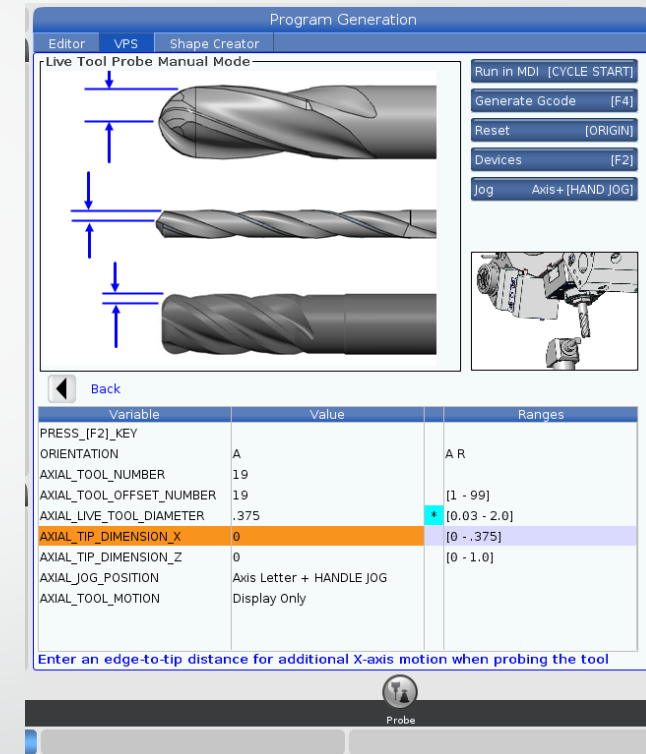
Run in MDI [CYCLE START]  
Generate Gcode [F4]  
Reset [ORIGIN]  
Devices [F2]  
Jog Axis+[HAND JOG]

Axis: Z Machine: 12.5700

Variable	Value	Ranges
PRESS_[F2]_KEY		
ORIENTATION	R	A R
RADIAL_TOOL_NUMBER	7	
RADIAL_TOOL_OFFSET_NUMBER	7	[1 - 99]
RADIAL_LIVE_TOOL_DIAMETER	.375	[0.03 - 2.0]
RADIAL_TIP_DIMENSION_X	0	[0 - 1.0]
RADIAL_TIP_DIMENSION_Z	0	[0 - .375]
RADIAL_JOG_POSITION	Axis Letter + HANDLE JOG	
RADIAL_TOOL_MOTION	Display Only	

Enter the diameter of the RADIAL live tool.

Probe



Program Generation

Editor VPS Shape Creator

Live Tool Probe Manual Mode

Run in MDI [CYCLE START]  
Generate Gcode [F4]  
Reset [ORIGIN]  
Devices [F2]  
Jog Axis+[HAND JOG]

Back

Variable	Value	Ranges
PRESS_[F2]_KEY		
ORIENTATION	A	A R
AXIAL_TOOL_NUMBER	19	
AXIAL_TOOL_OFFSET_NUMBER	19	[1 - 99]
AXIAL_LIVE_TOOL_DIAMETER	.375	[0.03 - 2.0]
AXIAL_TIP_DIMENSION_X	0	[0 - .375]
AXIAL_TIP_DIMENSION_Z	0	[0 - 1.0]
AXIAL_JOG_POSITION	Axis Letter + HANDLE JOG	
AXIAL_TOOL_MOTION	Display Only	

Enter an edge-to-tip distance for additional X-axis motion when probing the tool

Probe

# Live Tooling Geometry Offsets

Setup: Jog 09:03:59 MDI N451

Program Generation Editor VPS Shape Creator

Live Tool Probe Manual Mode

.25" (MAX.)

.25" (MAX.)

Run in MDI [CYCLE START]  
Generate Gcode [F4]  
Reset [ORIGIN]  
Devices [F2]  
Jog Axis+[HAND JOG]

Back Axis: Z Machine: 12.5700

Variable	Value	Ranges
PRESS_[F2]_KEY		
ORIENTATION	R	A R
RADIAL_TOOL_NUMBER	7	
RADIAL_TOOL_OFFSET_NUMBER	7	[1 - 99]
RADIAL_LIVE_TOOL_DIAMETER	.375	[0.03 - 2.0]
RADIAL_TIP_DIMENSION_X	0	[0 - 1.0]
RADIAL_TIP_DIMENSION_Z	0	[0 - .375]
RADIAL_JOG_POSITION	Axis Letter + HANDLE JOG	
RADIAL_TOOL_MOTION	Display Only	

Jog RADIAL tool within .25" of probe as shown. Press AXIS letter + HANDLE JOG key.

Main Spindle  
STOP  
Overrides  
Spindle Speed: 0 RPM  
Spindle Power: 0.0 KW  
Surface Speed: 0 FPM  
Chip Load: 0.00000 IPT  
Feed Rate: 0.0000 IPR  
Active Feed: 0.0000 IPR  
Feed: 100%  
Spindle: 100%  
Rapid: 100%

Spindle Load(%) 0%

Setup Jog Probe

Input:

- F2 can be used to lower the tool probe.
- Selecting an axis and then pressing hand jog allows the user to hand wheel the tool to the probe without leaving the screen.
- After the tool is in place, F4 can be used to output the code. In this case, it is output to MDI.

# Live Tooling Geometry Offsets

The screenshot displays a CNC control interface in MDI mode. The top left shows the MDI window with the following text:

```
MDI N451
Live Tool Probe Manual Mode
(**RADIAL LIVE TOOL MEASUREMENT - MANUAL MODE**);
(TOOL = 7 / OFFSET = 7);
(TOOL DIAMETER = -.375);
(RADIAL TOOL TIP X = 0);
(RADIAL TOOL TIP Z = 0);
(**USE CAUTION WHEN TOOL AND OFFSET NUMBERS DO NOT MATCH**);
:
M1.34 P800 (**VERIFY LIVE TOOL SPINDLE IS SPINNING OPPOSITE OF CUTTING DIRECTION**);
G211 T707 H03 Z0.1875 U0. W0.;
M30;
```

The top right shows the Active Codes window with the following list:

Code	Description
G01	Linear Feed Motion
G99	Feed Per Revolution
G40	Cancel Tool Nose Compensation
G80	Cycle Cancel
G54	Work Offset #54

The top right also shows the Active Tool window with the following data:

Parameter	Value
Tool	7
Offset	7
Type	None
Tool Group	-----
Max Load	0
Life	100%

The top right also shows the Coolant window with the following data:

Parameter	Value
Coolant	Off

The middle section shows the Spindles window with the following data:

Parameter	Main Spindle	Live Tool
Spindle Speed: (RPM)	0	0
Spindle Power: (KW)	0.0	0.0
Spindle Load: (%)	0%	0%
Surface Speed: (FPM)	0.0000	0.0000
Chip Load:	0.00000	0.00000
Spindle Override:	100%	100%
Direction:	Stop	Stop

The bottom left shows the Main Spindle window with the following data:

Parameter	Value
Spindle Speed	0 RPM
Spindle Power	0.0 KW
Surface Speed	0 FPM
Chip Load	0.00000 IPT
Feed Rate	0.0000 IPR
Active Feed	0.0000 IPR

The bottom left also shows the Overrides window with the following data:

Parameter	Value
Feed	100%
Spindle	100%
Rapid	100%

The bottom middle shows the Positions window for Program G54 T707 with the following data:

Axis	Position (IN)	Load
X	4.2235	40%
Y	0.0000	2%
Z	0.2570	3%
B	-0.2	0%
C	-----	0%

The bottom right shows the Timers And Counters window with the following data:

Parameter	Value
This Cycle	0:00:08
Last Cycle	0:00:08
Remaining	0:00:00
M30 Counter #1	93
M30 Counter #2	93
Loops Remaining	0

The bottom of the interface shows the Setup and Probe buttons, and the Input field.

- With the program in MDI, the probe down, and the tool in position, the program is ready to run.
- It is important that the live tool is spinning the opposite of the tools cutting direction.
  - If not, the tool could grab the tool probe with contact is made.

# Live Tooling Geometry Offsets

The screenshot displays a CNC control interface with the following sections:

- MDI (Manual Data Input):** Shows a program in manual mode with parameters like TOOL DIAMETER = -0.375 and various G-codes (G21, G54, M30).
- Offsets Table:** A table listing tool offsets for 18 tools. Tool 9 is highlighted in red, showing X Geometry of -21.9657 and Z Geometry of -8.6713.
- Main Spindle:** Displays spindle speed, power, and feed rate.
- Positions:** Shows X, Y, Z, B, and C coordinates with corresponding bar graphs.
- Timers And Counters:** Displays cycle time, remaining time, and counter values.

Tool Offset	Turret Location	X Geometry	Y Geometry	Z Geometry	Radius Geometry	Tip Direction
1	0	-17.9363	0.	-9.3449	0.	3: X- Z-
2	0	0.	0.	0.	0.	3: X- Z-
3	0	-17.9362	0.	-9.3449	0.	3: X- Z-
4	0	0.	0.	0.	0.	0: None
5	0	-10.0000	0.	-9.0000	0.	0: None
6	0	0.	0.	0.	0.	0: None
7	0	-20.1635	0.	-12.8270	0.	3: X- Z-
8	0	0.	0.	0.	0.	0: None
9 Spindle	0	-21.9657	0.	-8.6713	0.	2: X+ Z-
10	0	0.	0.	0.	0.	0: None
11	0	0.	0.	0.	0.	0: None
12	0	0.	0.	0.	0.	0: None
13	0	0.	0.	0.	0.	0: None
14	0	0.	0.	0.	0.	0: None
15	0	0.	0.	0.	0.	0: None
16	0	0.	0.	0.	0.	0: None
17	0	0.	0.	0.	0.	0: None
18	0	0.	0.	0.	0.	0: None

- The geometry offsets for the live tools are entered into the offset page automatically.

# Starting the Live Tooling

- **M133** – Live Tool Forward
  - **P** sets the speed
    - **M133 P3000** – would turn the tool on at 3000 RPM
- **M134** – Live Tool Reverse
- **M135** – Live Tool Stop

```
%  
O00000 (LIVE TOOL LATHE FORMAT)  
N1 (TOOL NOTES)  
G00 G53 X0  
G53 Z0  
T0101  
M05  
G18 G40 G54 G80 G98 Z0.1 M08  
M133 P3000 (LIVE TOOL ON)  
M154 (C-AXIS ON)  
X2.
```

# C Axis Engagement

- **New Codes:**

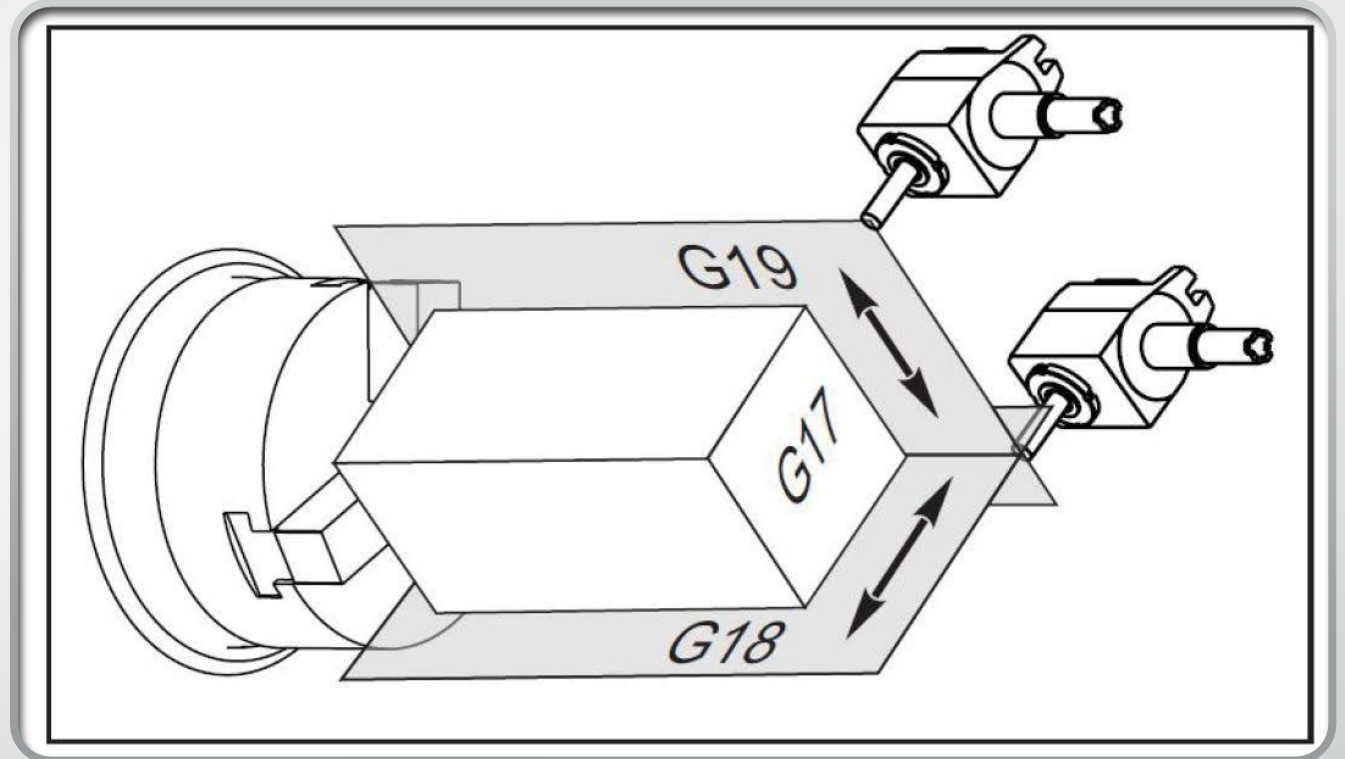
- **M154** – C axis engage
  - The C axis must be engaged to locate and position the spindle.
- **M155** – C axis disengage
  - The C axis must be disengaged before using the spindle at high RPM.

```
%  
O00000 (LIVE TOOL LATHE FORMAT)  
N1 (TOOL NOTES)  
G00 G53 X0  
G53 Z0  
T0101  
M05  
G18 G40 G54 G80 G98 Z0.1 M08  
M133 P3000 (LIVE TOOL ON)  
M154 (C-AXIS ON)  
X2.  
  
(MACHINE PART)  
(THERE ARE SEVERAL LIVE TOOL CANNED CYCLES)  
  
G00 G80 G99 Z0.1 M09  
M135 (LIVE TOOL OFF)  
M155 (TURN OFF C-AXIS IF NECESSARY)  
G53 X0  
G53 Z0  
M01
```



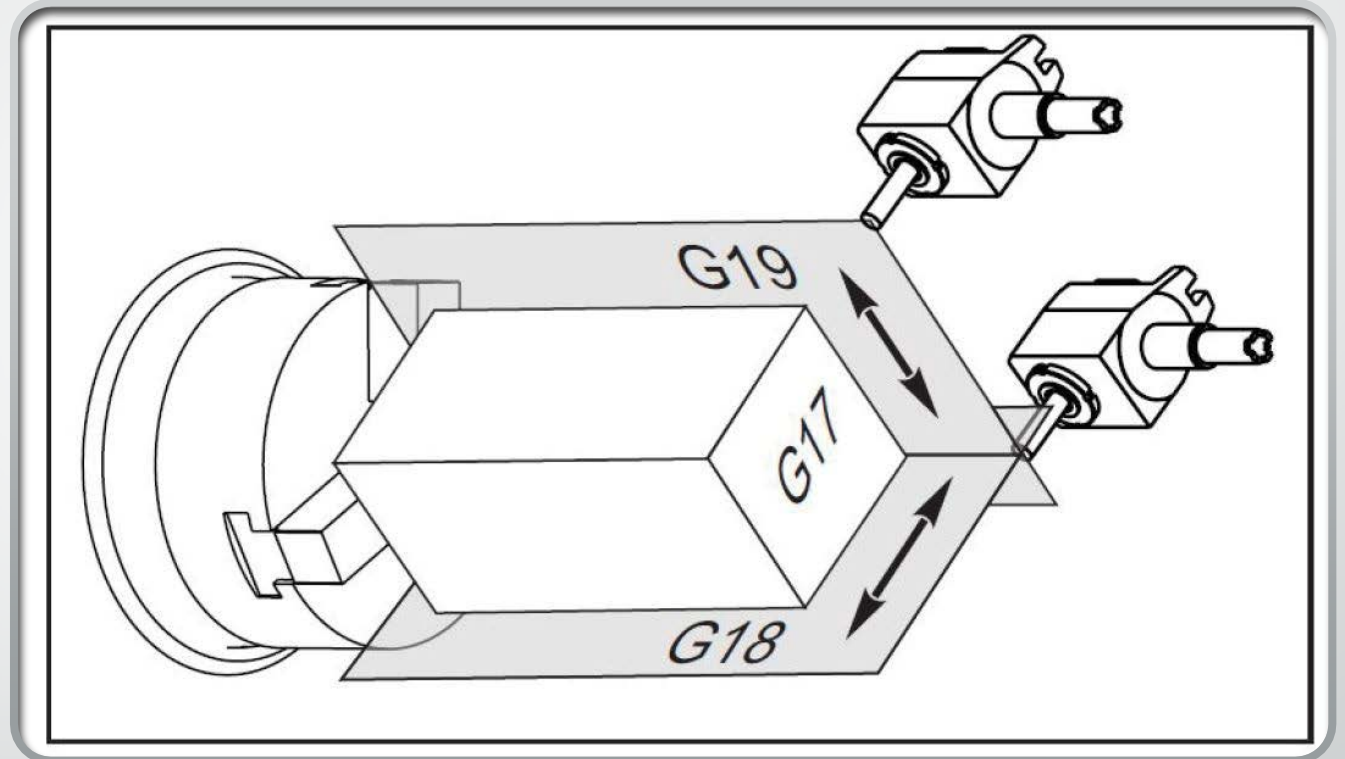
# Plane Selection

- Different planes can be used to make different canned cycles and milling tool paths work correctly on a live tooling lathe.
- **Drill or Mill** – refers to face work or axial work on a live tooling lathe.
- **Cross Mill or Cross Drill** – refers to work on the OD or radial work on a live tooling lathe.



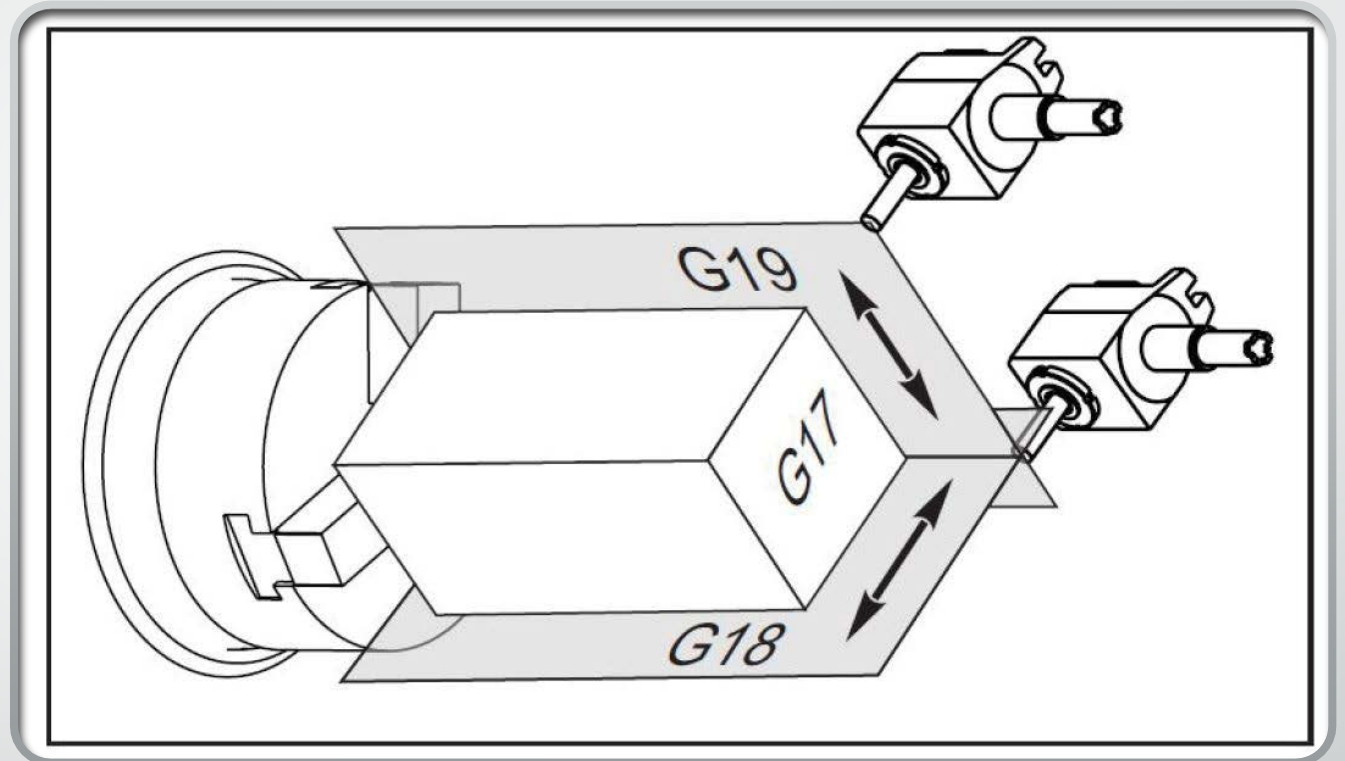
# Plane Selection

- **G17, G18, G19**
  - What are these codes?
    - **G17** – XY plane
    - **G18** – XZ plane
    - **G19** – YZ plane



# Plane Selection

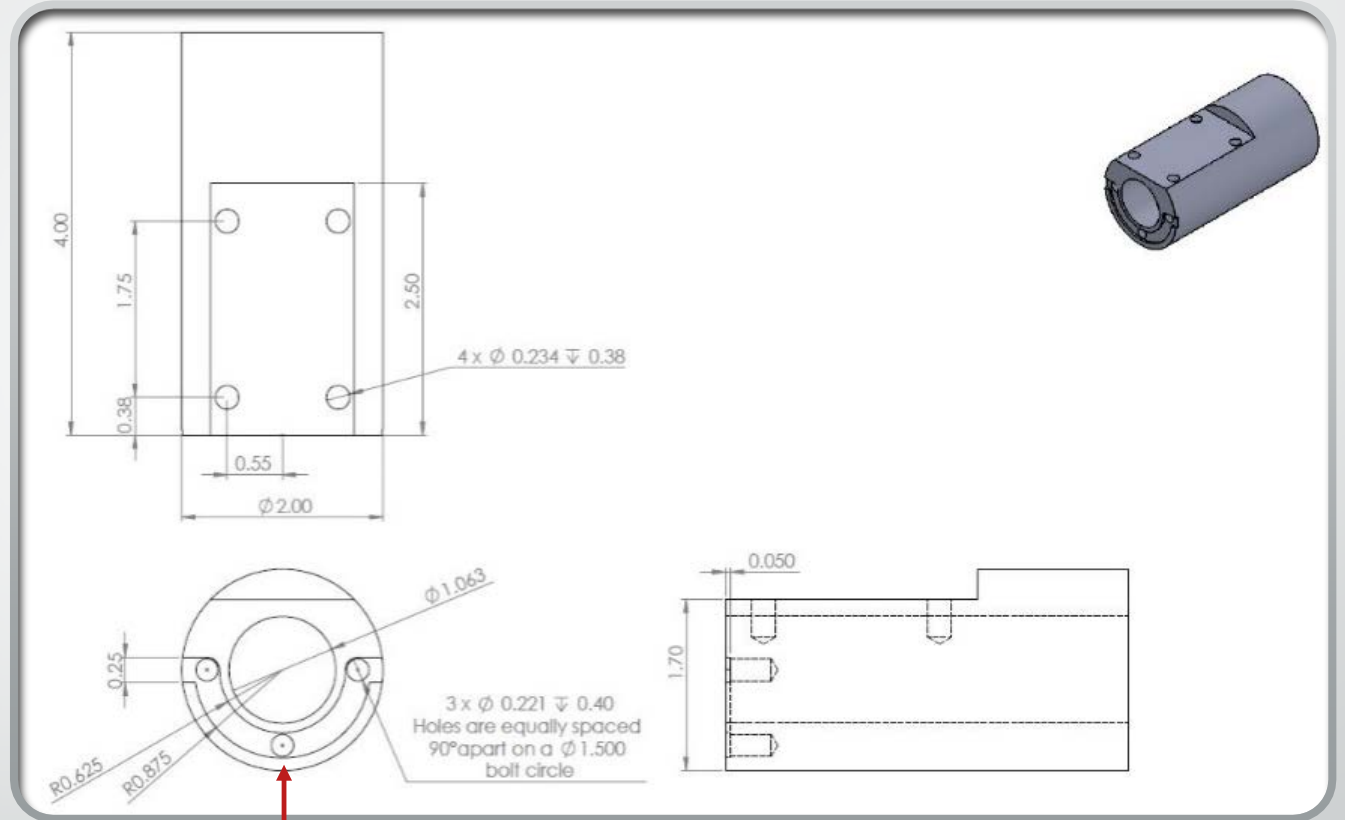
- What do these planes mean?
  - Plane selection (**G17**, **G18**, **G19**) determines the main plane of work.
  - Think of these planes as axis selection:
    - **G17** - the Z axis positions and the main cutting axis would be X and Y.
    - **G18** - the Y axis positions and the main cutting axis would be X and Z.
    - **G19** - the X axis positions and the main cutting axis would be Y and Z.



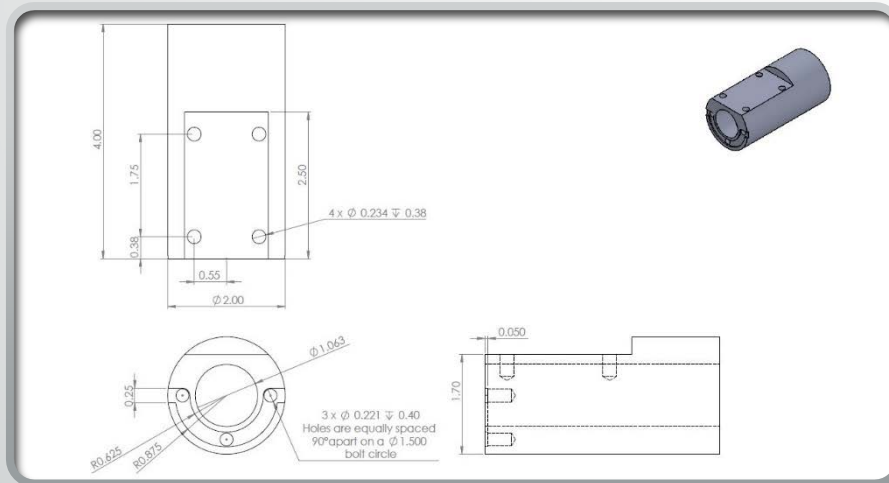
Productivity Inc. "Plane Selection." *Live Tool For has Lathes*. N.p.: Productivity Inc., 2012. 10. Print.

# G17 – Axial Milling

- **G17** is used when XY milling is taken place.
- **G17** is also used with G112
  - **G112** can convert XY moves to XC moves .
  - This means a milling toolpath could be converted to XC moves.
  - G112 is used when a live tool path needs to go beyond the X axis centerline but will not reach.



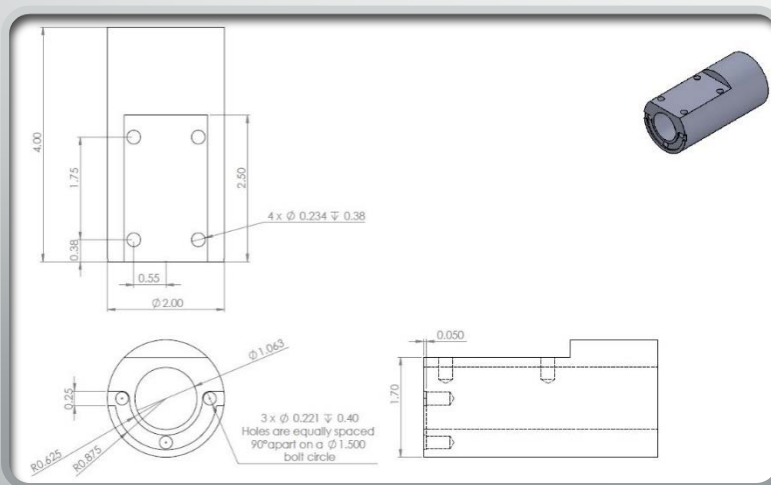
# G18 - The Default



- The turning, drilling and boring for the part would all happen with the **G18** plane.
- **G18** could also be used for the XC milling for the slot on the front face.
- **G18** is the default for slant bed lathes.
- All of the typical OD and ID turning will happen in the XZ plane.
  - These are the main axes of the machine.
- XC milling also happens with **G18** active.

# G19 – Radial and Cross Drilling

- **G19** would be used for milling the flat on the OD of the part.
- **G19** would also be used when drilling the four holes on the flat of the part.
- **G19** is used with all radial and cross drilling toolpaths.
- **G19** would also be used with any facing or milling happening on the OD of the part.



# G17 Plane Selection Guide

Cycle	Description	Feedrate
XY AXIS MILLING	MANUAL FACE PROGRAMMING	G98
G112	CARTESIAN TO POLAR	G98

# G18 Plane Selection Guide

Cycle	Description	Feedrate
G81	SIMPLE DRILLING	G98
G82	SIMPLE DRILL W/ DWELL	G98
G83	DRILL W/ PECK	G98
G95	RIGID TAP	G99
G186	RIGID TAP LEFT HAND	G99
X-C MILLING	FACE MILLING	G98

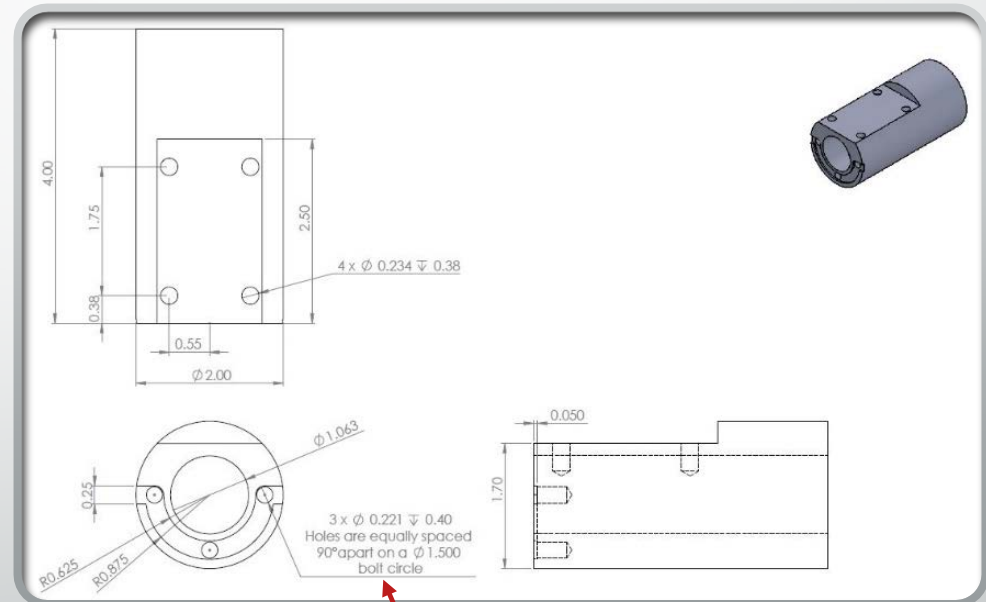


# G19 Plane Selection Guide

Cycle	Description	Feedrate
G241	CROSS DRILL	G98
G242	CROSS DRILL W/ DWELL	G98
G243	CROSS DRILL W/ PECK	G98
G195	RIGID TAP	G99
G196	RIGID TAP LEFT HAND	G99

# Face and Axial Drilling

- Drilling the three holes on the face of the part can be done with the regular **G80** series drilling canned cycles.
- The part will be programmed without a peck because of the depth.



# Face and Axial Drilling

- **G81** is programmed the same way it would be on a mill or a lathe.
  - Note: A "C" value can be used as a position move during a canned cycle.
- The depth is controlled by Z.
  - The plain needs to be set with **G18**.

```
M01 (DRILLING THE FACE .221 HOLES)
G40 G80 G00 G98
G18
T707
M154
P3000 M133
M14
C90.
Z1.
X1.5 M08
Z0.1
G81 Z-.4663 R.1 F8.0
C180.
C270.
G80
G00 Z1. M09
M135
M15
G53 Y0.
G53 X0.
G53 Z0.
```

# Face and Axial Drilling

- If **G82** is used, a **P** value needs to be entered for a dwell after the depth is reached.
- If **G83** is used, a **Q** value needs to be entered to specify the peck amount.

```
M01 (DRILLING THE FACE .221 HOLES)
G40 G80 G00 G98
G18
T707
M154
P3000 M133
M14
C90.
Z1.
X1.5 M08
Z0.1
G81 Z-.4663 R.1 F8.0
C180.
C270.
G80
G00 Z1. M09
M135
M15
G53 Y0.
G53 X0.
G53 Z0.
```

# G95 Live tooling rigid tap (face)

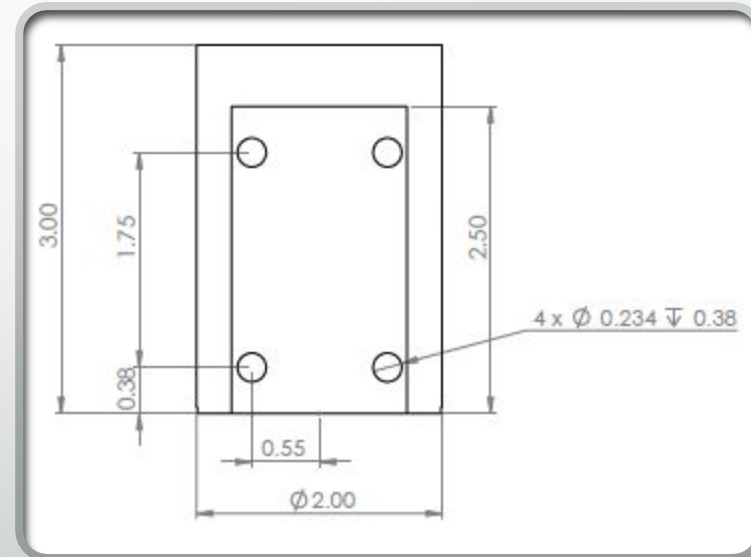
- *G95* Live Tooling Rigid Tapping is an axial tapping cycle similar to *G84* Rigid Tapping in that it uses the *F*, *R*, *X* and *Z* addresses. However, it has the following differences:
  - The control must be in *G99* Feed per Revolution mode in order for tapping to work properly.
  - An *S* (spindle speed) command must have been issued prior to the *G95*.
  - The *X* Axis must be positioned between machine zero and the center of the main spindle, do not position beyond spindle center.
- *G95 C45. Z-0.5 R0.5 F0.05 (Tap to Z-0.5)*
  - \* *C* - C-Axis absolute motion command (optional)
  - F* - Feed Rate
  - R* - Position of the R plane
  - S* - RPM, called prior to *G95*
  - W* - Z-axis incremental distance
  - X* - Optional Part Diameter X-axis motion command
  - \* *Y* - Y-axis motion command
  - Z* - Position of bottom of hole
  - \* indicates optional

# G186 Reverse Live tooling rigid tap (face)

- Similar to G95 but used for tapping left hand threads. The same rules apply to G186 as G95:
  - The control must be in G99 Feed per Revolution mode in order for tapping to work properly.
  - An S (spindle speed) command must have been issued prior to the G95.
  - The X Axis must be positioned between machine zero and the center of the main spindle. Do not position beyond spindle center.
- G95 C45. Z-0.5 R0.5 F0.05 (Tap to Z-0.5)
  - \* C - C-Axis absolute motion command (optional)
  - F - Feed Rate
  - R - Position of the R plane
  - S - RPM, called prior to G95
  - W - Z-axis incremental distance
  - X - Optional Part Diameter X-axis motion command
  - \* Y - Y-axis motion command
  - Z - Position of bottom of hole
  - \* indicates optional

# OD and Radial Drilling

- Drilling on the OD of the part can be done with the **G240** series of canned cycles.
  - These are designed to work with radial tool holders.
- Y and Z will position the tool and X will control depth on these tool paths.



# OD and Radial Drilling

- Cross drilling takes more thought and calculation for a successful toolpath.
  - The depth is controlled by X for these paths.
    - X is read by the machine as a diameter.
    - For this to work, more math will need to be done.

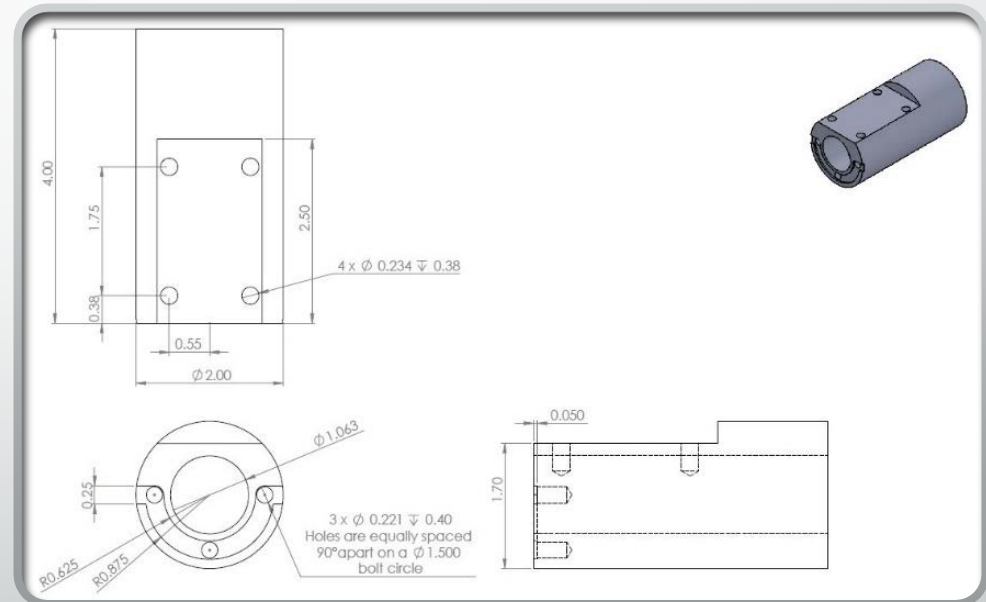
Where did this depth come from?

```
X2.5 M08  
G98  
G241 R2.5 X0.640 F10.  
Y-0.55  
Z-2.125  
Y0.55  
G80 G00  
X3. M09  
M135
```



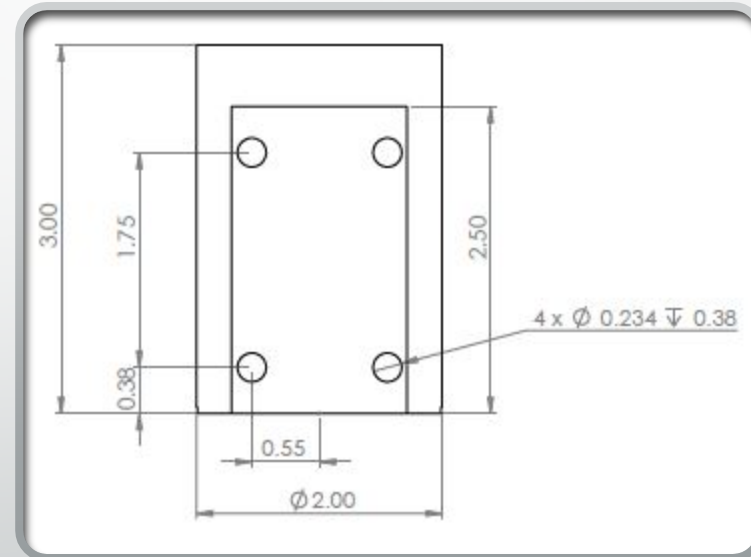
# Calculating Drill Depth

- The flat is at a dimension of 1.700 from the opposite side.
- This means the flat is .300 deep
  - This depth needs to be doubled (.600) to find the diameter on the lathe where the holes start.
- **2.000 - .600 depth = 1.400**
  - Diameter for the starting drill depth.



# Calculating Drill Depth

- The print states the holes are .38 deep.
  - This is from the flat
- This value will need to double (.760)
  - Drilling to a diameter because we are on a lathe.
- To get the final drill depth, subtract the final drill depth from the starting diameter:
  - $1.400 - .760 = .640$  Diameter



# OD and Radial Drilling

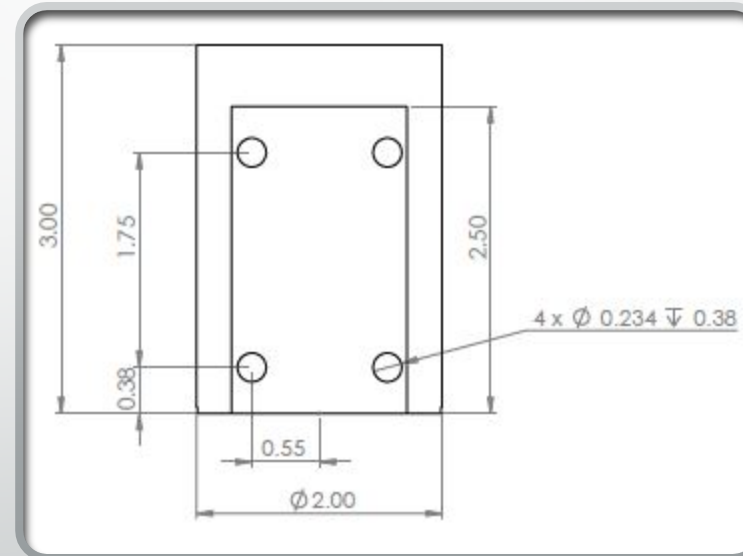
- **G19** is used because Y and Z are the positioning axis.
- This makes X the retract axis for the canned cycle.
- If **G18** was used, the automatic retract would happen in Z and result in a broken drill.

```
X2.5 M08  
G98  
G241 R2.5 X0.640 F10.  
Y-0.55  
Z-2.125  
Y0.55  
G80 G00  
X3. M09  
M135
```

# OD and Radial Drilling

## Radial Drilling using G241, G242, G243:

- Let's look at drilling holes on the diameter of the part:
  - Use the **G19** plane.
  - A Radial live tooling holder will be needed.
  - Remember that the machine still reads diameter .
    - Depths and retracts still need to be programmed to diameters and not radii.

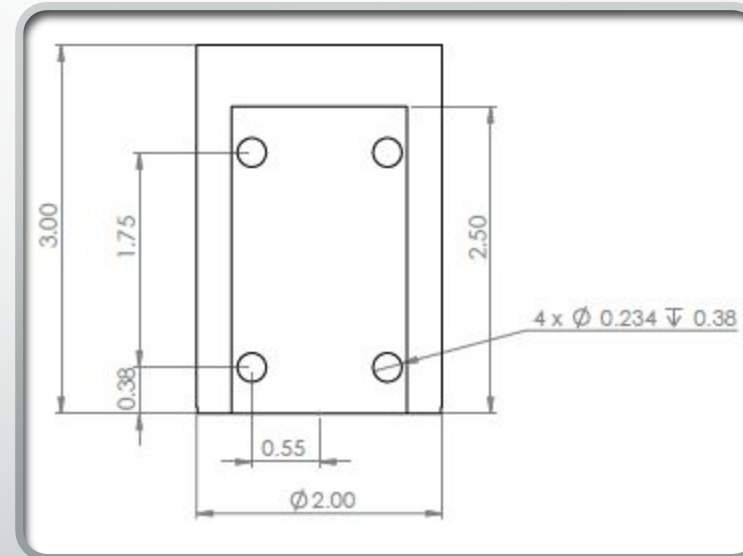


# OD and Radial Drilling

## Radial Drilling using G241:

**G241 X2.1 Y0.125 Z-1.3 C35. R4. F20**

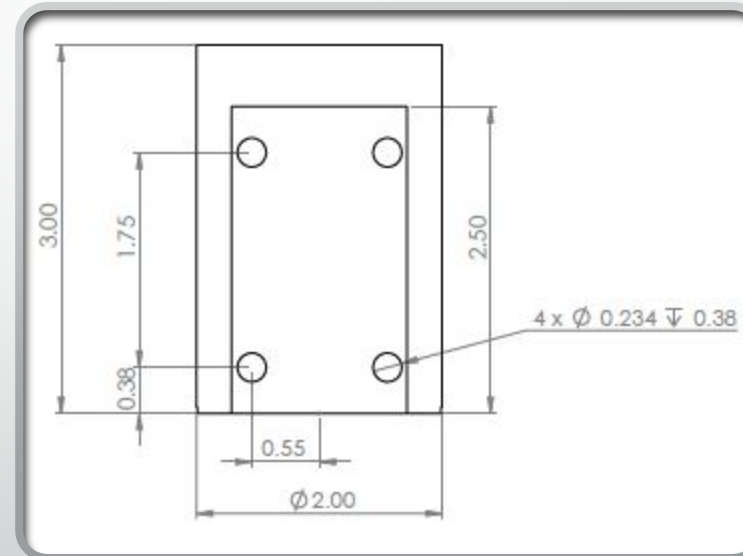
- C C-axis absolute motion command
- F Feed Rate Inch/minute
- R Position of the R plane (diameter)
- \*X Position of bottom of hole (diameter)
- \*Y Y-axis absolute motion command
- \*Z Z-axis absolute motion command
- \* indicates optional



# OD and Radial Drilling

## Radial Drilling using G241:

M154 (Engage C Axis)  
M133 P2500 (Live Tooling On, 2500 RPM)  
G19 (Y-Z Plane Selection)  
G98 (IPM)  
G00 X5. Z-0.75 Y0  
G241 X2.1 Y0.125 Z-1.3 C35. R4. F20. (Drill to X 2.1)



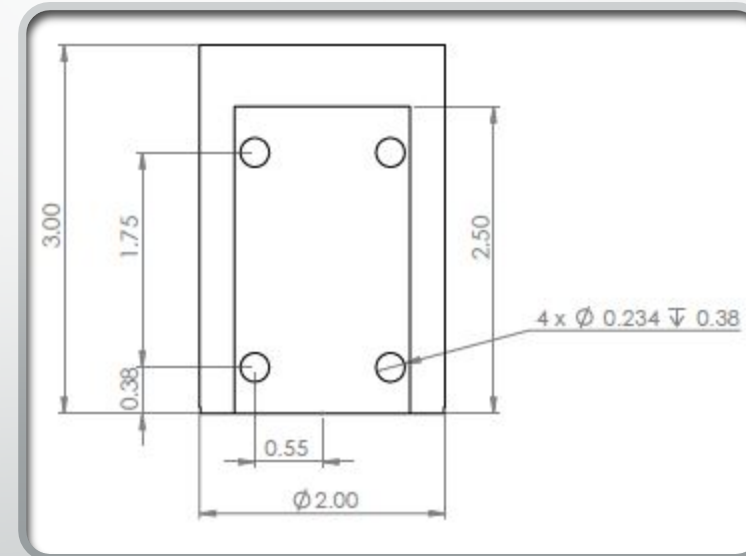
# OD and Radial Drilling

## Radial Drilling using G242:

### G242 Radial Spot Drill Canned Cycle (Group 09)

Drill and dwell canned cycle

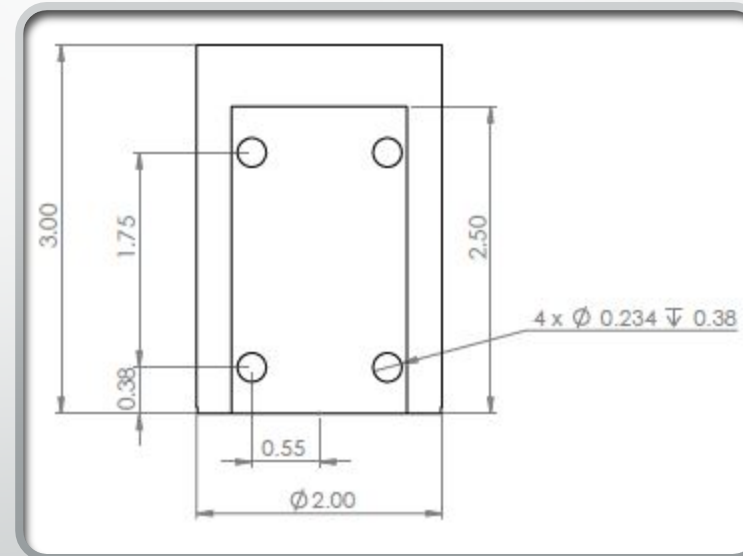
- C C-axis absolute motion command
- F Feed Rate
- P The dwell time at the bottom of the hole
- R Position of the R plane (Diameter)
- \*X Position of bottom of hole (Diameter)
- \*Y Y-axis motion command
- \*Z Z-axis motion command
- \* indicates optional



# OD and Radial Drilling

## Radial Drilling using G242:

```
M154 (Engage C Axis)  
M133 P2500 (2500 RPM)  
G19 (Y-Z Plane Selection)  
G98 (IPM)  
G00 X5. Z-0.75 Y0  
G242 X2.1 Y0.125 Z-1.3 C35. R4. P0.5 F20. (Drill to X 2.1)
```



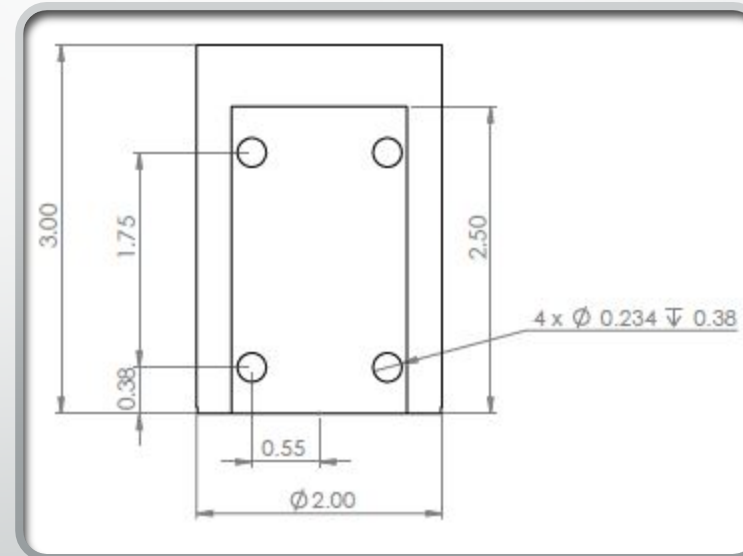


# OD and Radial Drilling

## Radial Drilling using G243:

### G243 Radial Normal Peck Drilling Canned Cycle (Group 09)

- C C-axis absolute motion command
- F Feed Rate (G98 In/mn)
- \*I Size of first cutting depth
- \*J Amount to reduce cutting depth each pass
- \*K Minimum depth of cut
- \*P The dwell time at the bottom of the hole
- \*Q The cut-in value, always incremental
- R Position of the R plane (Diameter)
- \*X Position of bottom of hole (Diameter)
- \*Y Y-axis absolute motion command
- \*Z Z-axis absolute motion command
- \* indicates optional

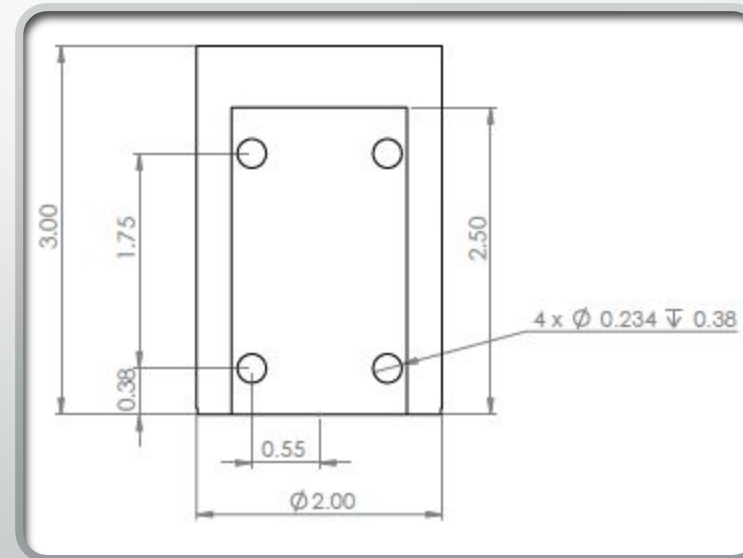


# OD and Radial Drilling

## Radial Drilling using G243

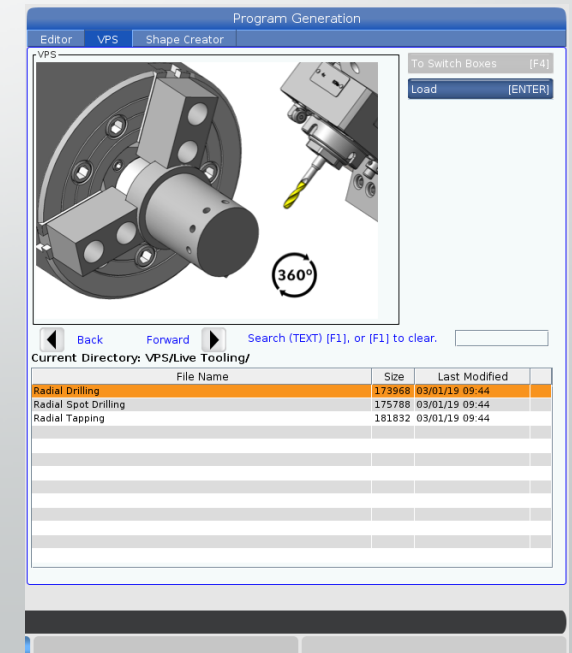
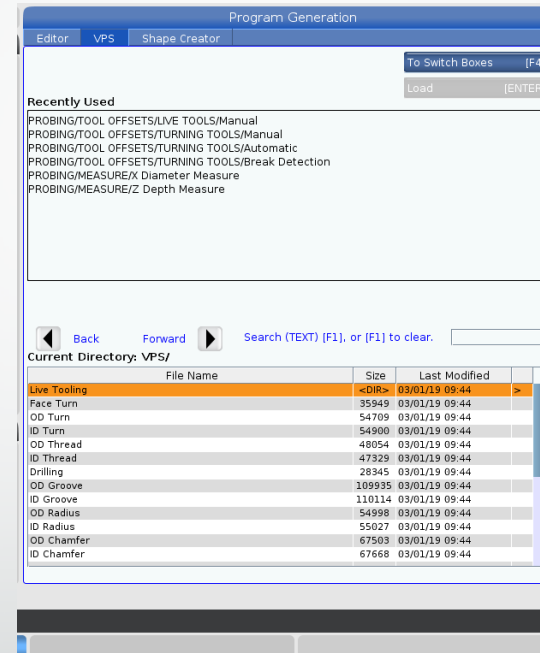
```
M154 (Engage C Axis)  
M133 P2500 (2500 RPM)  
G19  
G98 (IPM)  
G00 X5. Z-0.75 Y0  
G243 X2.1 Y0.125 Z-1.3 C35. R4. Q0.25 F20. (Drill to X 2.1)
```

- Setting 22 is the amount to feed in X to get the same point at which the retract occurred.



# Radial drilling w/VPS

- VPS can be used to program radial drilling toolpaths.
- Live tooling is the first option in the VPS
- Once live tooling is entered there are three options
  - Radial drilling
  - Radial spot drilling
  - Radial tapping



# Radial drilling w/VPS

Operation: MEM | 07:34:21 | Program Generation

MEM | Memory/O00000.nc | N451

000000;

Radial Drilling

Run in MDI [CYCLE START]  
Generate Gcode [F4]  
Reset [ORIGIN]  
Devices [F2]  
Jog Axis+[HAND JOG]

Variable	Value	Ranges
TOOL_NUMBER	5	[1 - 24]
TOOL_OFFSET_NUMBER	5	[1 - 99]
TOOL_OFFSET_IS_ZERO		

Back

Main Spindle

STOP

Overrides

Spindle Speed: 0 RPM  
Spindle Power: 0.0 KW  
Surface Speed: 0 FPM  
Chip Load: 0.00000 IPT  
Feed Rate: 0.0000 IPR  
Active Feed: 0.0000 IPR

Feed: 100%  
Spindle: 100%  
Rapid: 100%

Spindle Load(%) 0%

Setup Door Open Power Save

Input: |

Current active TOOL NUMBER is default. Alternate TOOL NUMBER can be selected

- After entering the radial drilling option, the tool number will need to be entered.
- After the tool number is entered, the rest of the command fields will appear.

# Radial drilling w/VPS

- There are two pages of variables to input.
- If there is more than one hole, the number of holes will have to be defined.
- The C start angle for the first hole needs to be called out.

Program Generation

Editor VPS Shape Creator

Radial Drilling

Run in MDI [CYCLE START]  
Generate Gcode [F4]  
Reset [ORIGIN]  
Devices [F2]  
Jog Axis+ [HAND JOG]

SINGLE HOLE = S      MULTIPLE HOLES = M

Variable	Value	Ranges
TOOL_NUMBER	5	[1 - 24]
TOOL_OFFSET_NUMBER	5	[1 - 99]
WORK_OFFSET	54	[54 - 59]
FLOOD_COOLANT	8	8 9
DRILL_START_POINT	2.5	[0.25 - 8.9]
Z_RAPID_APPROACH	0.2	[0.01 - 2.0]
X_RAPID_APPROACH	0.2	[0.01 - 2.0]
DRILL_RPM	1500	[5 - 6000]
SINGLE_MULTIPLE	S	S M
C_AXIS_START_ANGLE	0.0	[0.0 - 360.00]
Z_AXIS_DISTANCE	0.0	[0.0 - 14.0]

Enter (S) for single hole or (M) for multiple holes equally spaced

Program Generation

Editor VPS Shape Creator

Radial Drilling

Run in MDI [CYCLE START]  
Generate Gcode [F4]  
Reset [ORIGIN]  
Devices [F2]  
Jog Axis+ [HAND JOG]

# OF HOLES?

Back

Variable	Value	Ranges
TOOL_NUMBER	5	[1 - 24]
TOOL_OFFSET_NUMBER	5	[1 - 99]
WORK_OFFSET	54	[54 - 59]
FLOOD_COOLANT	8	8 9
DRILL_START_POINT	2.5	[0.25 - 8.9]
Z_RAPID_APPROACH	0.2	[0.01 - 2.0]
X_RAPID_APPROACH	0.2	[0.01 - 2.0]
DRILL_RPM	1500	[5 - 6000]
SINGLE_MULTIPLE	M	S M
C_AXIS_START_ANGLE	0.0	[0.0 - 360.00]
NUMBER_OF_HOLES	3	[2 - 180]

Enter the NUMBER OF HOLES equally spaced

# Radial drilling w/VPS

MDI N451

```
(Radial Drilling);  
( SAFETY LINE BELOW );  
G00 G54 G18 G40 G80 G97 G99;  
(TOOL = 5 / OFFSET = 5);  
(WORK OFFSET = 54);  
(DRILL RPM = 1500);  
(C-AXIS START ANGLE = 0.0);  
(# OF EQUALLY SPACED HOLES = 3);  
T505;  
G54;  
G98 (FEED PER MINUTE);  
G00 Z0.2;  
M133 P1500;  
M08;  
G00 C0. X2.9;  
G00 Y0.;  
G241 R2.7 X2.3 Z-0.5 F8.;  
C120.;  
C240.;  
G80;  
M09;  
G99 M135;  
G00 G53 Y0.;  
G53 X0.;  
G53 Z0.;  
( END RADIAL DRILLING CYCLE );  
M01;
```

Main Spindle

STOP

Spindle Speed: 0 RPM  
Spindle Power: 0.0 KW  
Surface Speed: 0 FPM

Overrides

Chip Load: 0.00000 IPT  
Feed Rate: 0.00000 IPR  
Active Feed: 0.00000 IPR

Feed: 100%  
Spindle: 100%  
Rapid: 100%

Spindle Load(%) 0%

Setup Door Open Power Save

Input:

Program Generation

Editor VPS Shape Creator

Radial Drilling

Run in MDI [CYCLE START]  
Generate Gcode [F4]  
Reset [ORIGIN]  
Devices [F2]  
Jog Axis+[HAND JOG]

Program

1: Insert to Clipboard  
2: Output to MDI  
3: Create New Program

Exit [CANCEL]

Parameter	Value	Ranges
Z_A	0.1	[0.0 - 14.0]
R_PLANE	0.1	[0.0 - 3.2]
DRILL_DEPTH	0.1	[0.0 - 1.5625]
FEEDRATE_IPM	8.	[0.25 - 40.00]
RETRACT_X_HOME	Y	Y N
RETRACT_Z_HOME	Y	Y N
END_M_CODE	1	0 1 30

Enter value: Stop (0), Optional Stop (1), End Program (30)

- After all the variables are input, F4 can be used to generate the code. In this case, option 2 will be used to output the code to MDI.

# Radial drilling w/VPS

MDI Edit: MDI 07:36:35

MDI N451

**Radial Drilling:**  
( SAFETY LINE BELOW );  
G00 G54 G18 G40 G80 G97 G99;  
(TOOL = 5 / OFFSET = 5);  
(WORK OFFSET = 54);  
(DRILL RPM = 1500);  
(C-AXIS START ANGLE = 0.0);  
(# OF EQUALLY SPACED HOLES = 3);  
T505;  
G54;  
G98 (FEED PER MINUTE);  
G00 Z0.2;  
M133 P1500;  
M08;  
G00 C0. X2.9;  
G00 Y0.;  
G241 R2.7 X2.3 Z-0.5 F8.;  
C120.;  
C240.;  
G80;  
M09;  
G99 M135;  
G00 G53 Y0.;  
G53 X0.;  
G53 Z0.;  
( END RADIAL DRILLING CYCLE );  
M01;

Active Codes

G00	Rapid Motion
G99	Feed Per Revolution
G40	Cancel Tool Nose Compensation
G80	Cycle Cancel
G54	Work Offset #54

Active Tool

Tool: 5	Offset: 5
Type: None	
Tool Group: -----	
Max Load: 0	
Life: 100%	

Coolant

Off

1/1

0/1

Spindles

	Main Spindle	Live Tool
Spindle Speed: (RPM)	0	0
Spindle Power: (KW)	0.0	0.0
Spindle Load: (%)	0%	0%
Surface Speed: (FPM)	0.0000	0.0000
Chip Load:	0.00000	0.00000
Spindle Override:	100%	100%
Direction:	Stop	Stop

Main Spindle

STOP

Overrides

Feed: 100%	Spindle: 100%	Rapid: 100%
------------	---------------	-------------

Spindle Load (%) 0%

Positions Program G54 T505

Axis	Position (IN)	Load
X	0.0000	0%
Y	0.0000	0%
Z	0.0000	0%
B	-0.2	0%
C	-----	0%

Timers And Counters

This Cycle:	0:00:05
Last Cycle:	0:00:05
Remaining	0:00:00
M30 Counter #1:	92
M30 Counter #2:	92
Loops Remaining:	0

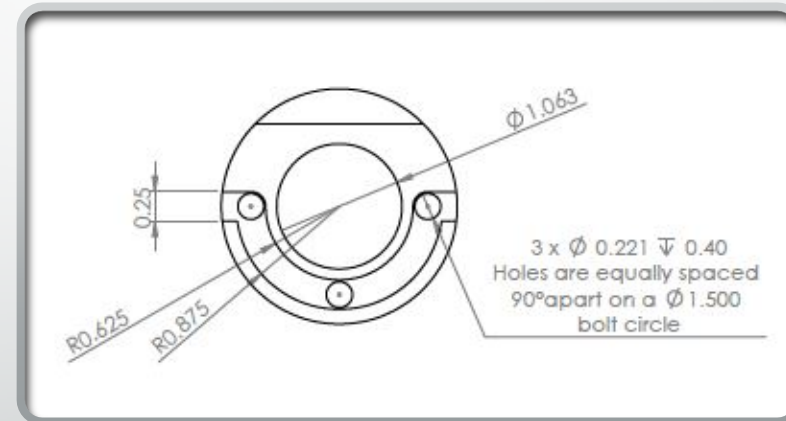
Setup Door Open Power Save

Input:

- Once the program is output and the tools geometry offsets are set, the program is ready to run.

# Axial or Face Milling

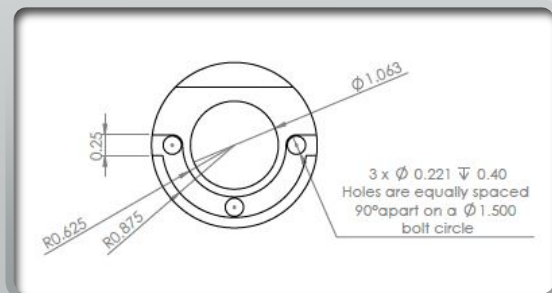
- Let's review the slot on the front face of this part.
- This can be cut two ways:
  - With X & C programming.
  - With **G112** Cartesian to Polar programming.





# Milling the Face Slot

- We have set the entrance of the groove to  $90^\circ$  and the exit at  $270^\circ$ .
- A  $\frac{1}{4}$ " Endmill is used to mill the slot in one pass at the desired depth.
- **G98** is used because of the milling operation.



Entry point



```
M133 P3000  
Z0.1  
X2.5  
C90.  
G98 G01 Z-0.05 F10. M08  
X1.5  
Exit point → C270.  
X2.5  
G00 X3.5 M09  
M135
```

Exit point




# Milling the Face Slot

- It is important to note that the feedrate is set to IPM.
- The C axis will also be rotating while cutting.
  - This is a rotational move, not a linear move.
- The Haas control will convert linear feeds to rotation feeds automatically.
- For the control to do this properly, **setting 102** must be set to the diameter that is going to cut.
  - If this diameter is wrong, the feed conversion will not be precise.

```
M133 P3000  
Z0.1  
X2.5  
C90.  
G98 G01 Z-0.05 F10. M08  
X1.5  
C270.  
X2.5  
G00 X3.5 M09  
M135
```

Programmed  
feedrate



# Milling the Face Slot

- Set the entrance of the groove to  $90^\circ$  and the exit at  $270^\circ$ .
- A  $\frac{1}{4}$ " Endmill is used to mill the slot in one pass at the desired depth.
- **G98** is used because of the milling operation.

```
M133 P3000  
Z0.1  
X2.5  
C90.  
G98 G01 Z-0.05 F10. M08 ← Start of slot  
X1.5 ← Move to Depth  
C270. ← Start cutting slot  
X2.5 ← End of slot  
G00 X3.5 M09 ← Exit part  
M135
```