Welcome to Trident Machine Tools

Live Tool Lathe Training

Haas live tool lathe training

- This ½ 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

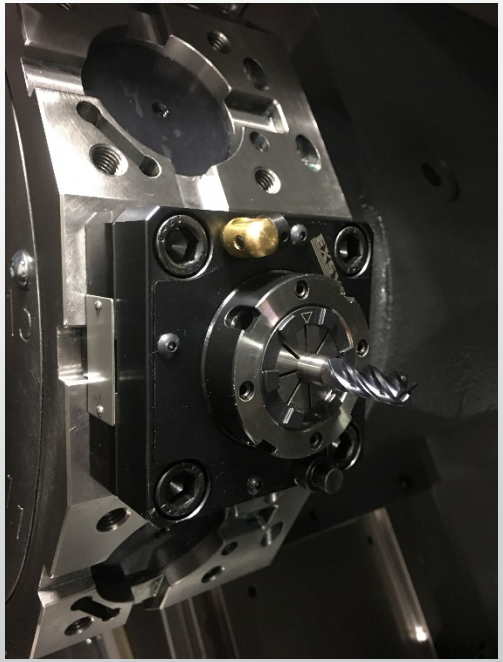
- Most CNC slant bed lathes consist of two axis.
  - X – controls diameter
  - Z – controls depth
- These machines can perform simple turning, drilling, threading, boring, etc.
- All work must be done on the centerline of the spindle.

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

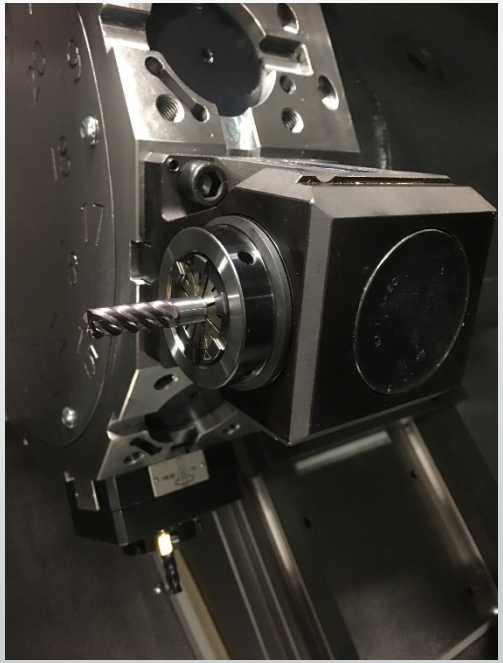
- **Radial tools**
  - Hold the cutting tool parallel to the face of the turret.
  - Designed to work on the diameter of the part performing operations:
    - Milling
    - Drilling
    - Tapping



Haas Automation Inc. "*Axial and Radial Live Tooling*." Lathe Operator's Manual. N.p.: Haas Automation Inc., 2014. 242. Print.

Types of Live Tooling

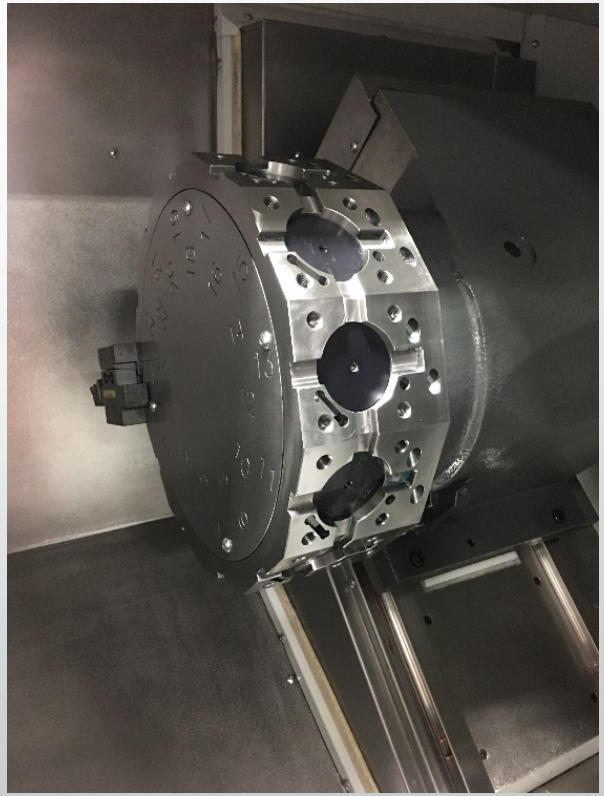
- **Axial tools**
  - Hold the cutting tool perpendicular to the face of the turret
  - Designed to work on the face of the part performing operations
    - Milling
    - Drilling
    - Tapping



Haas Automation Inc. "*Axial and Radial Live Tooling*." Lathe Operator's Manual. N.p.: Haas Automation Inc., 2014. 242. Print.

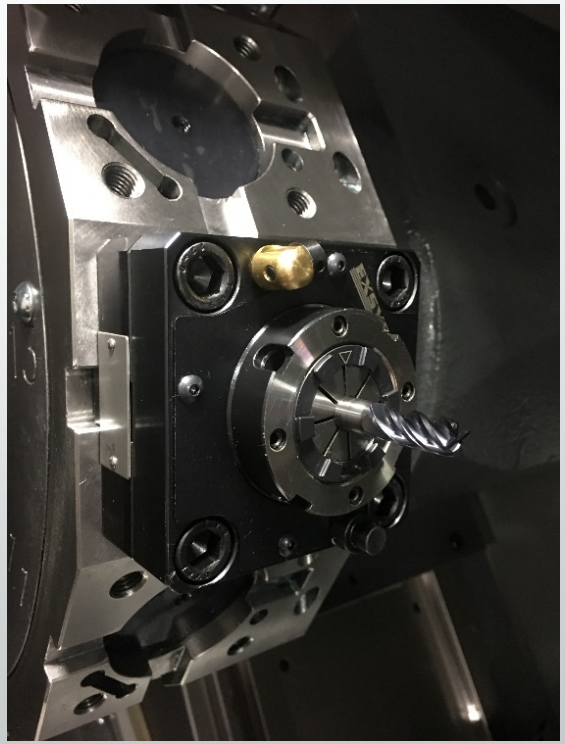
Base Mount Turret (BMT)

- 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.



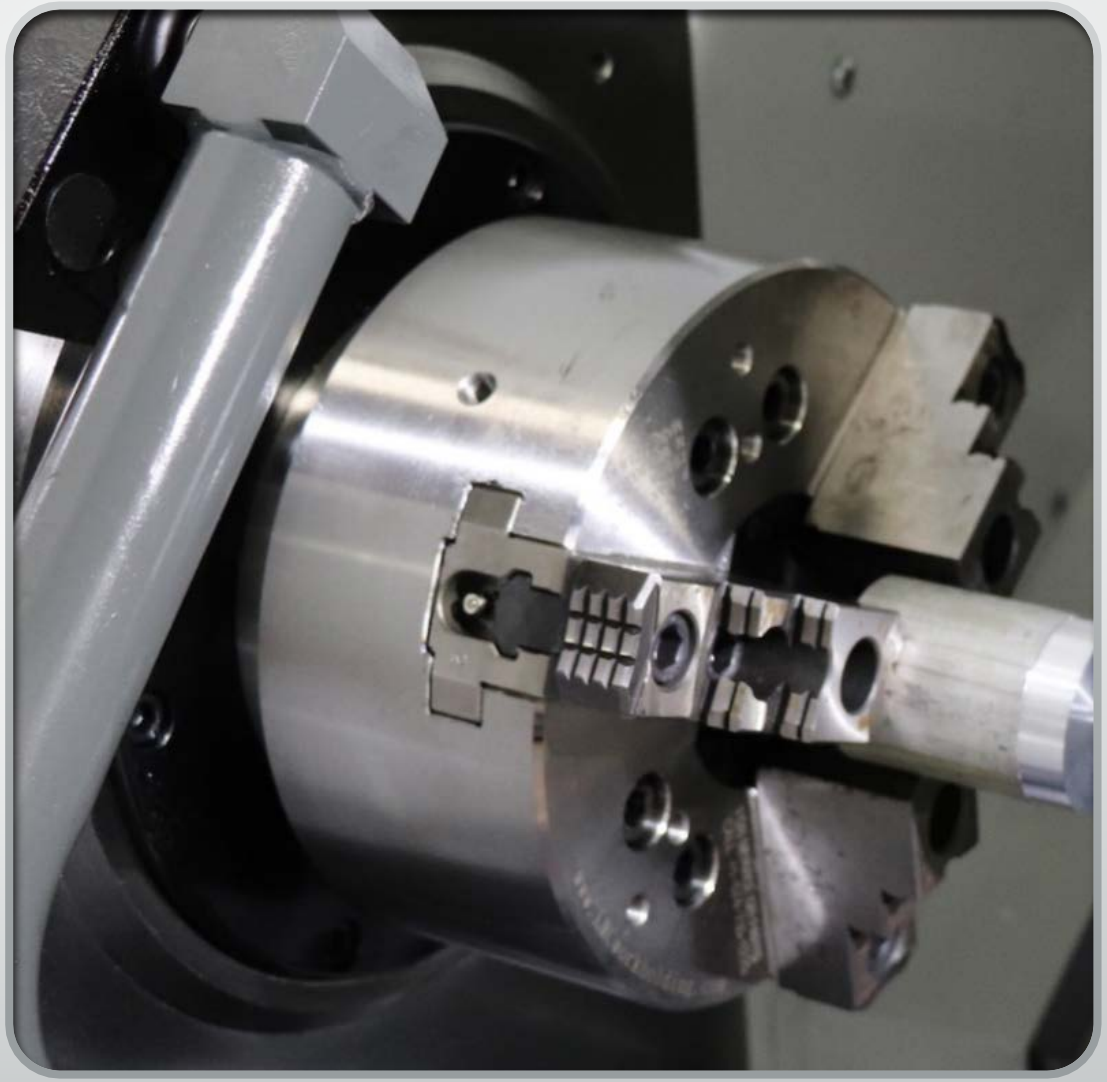
Types of Live Tooling

- **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



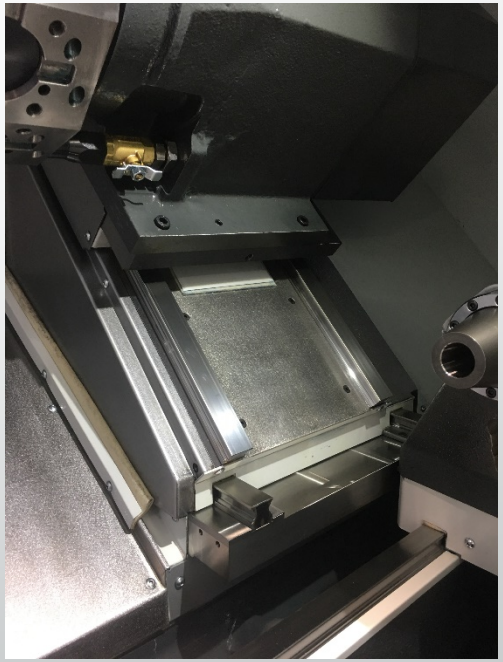
C Axis

- 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



Haas Automation Inc. "*Y axis Motion*." Lathe Operator's Manual. N.p.: Haas Automation Inc., 2014. 255. Print.

Axial Tools

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

Photo Source: Exsys Tool. N.d. Web. 29 January 2015.<http://www.exsystool.com/images/catalog/archive/haas-tooling-catalog.pdf>

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

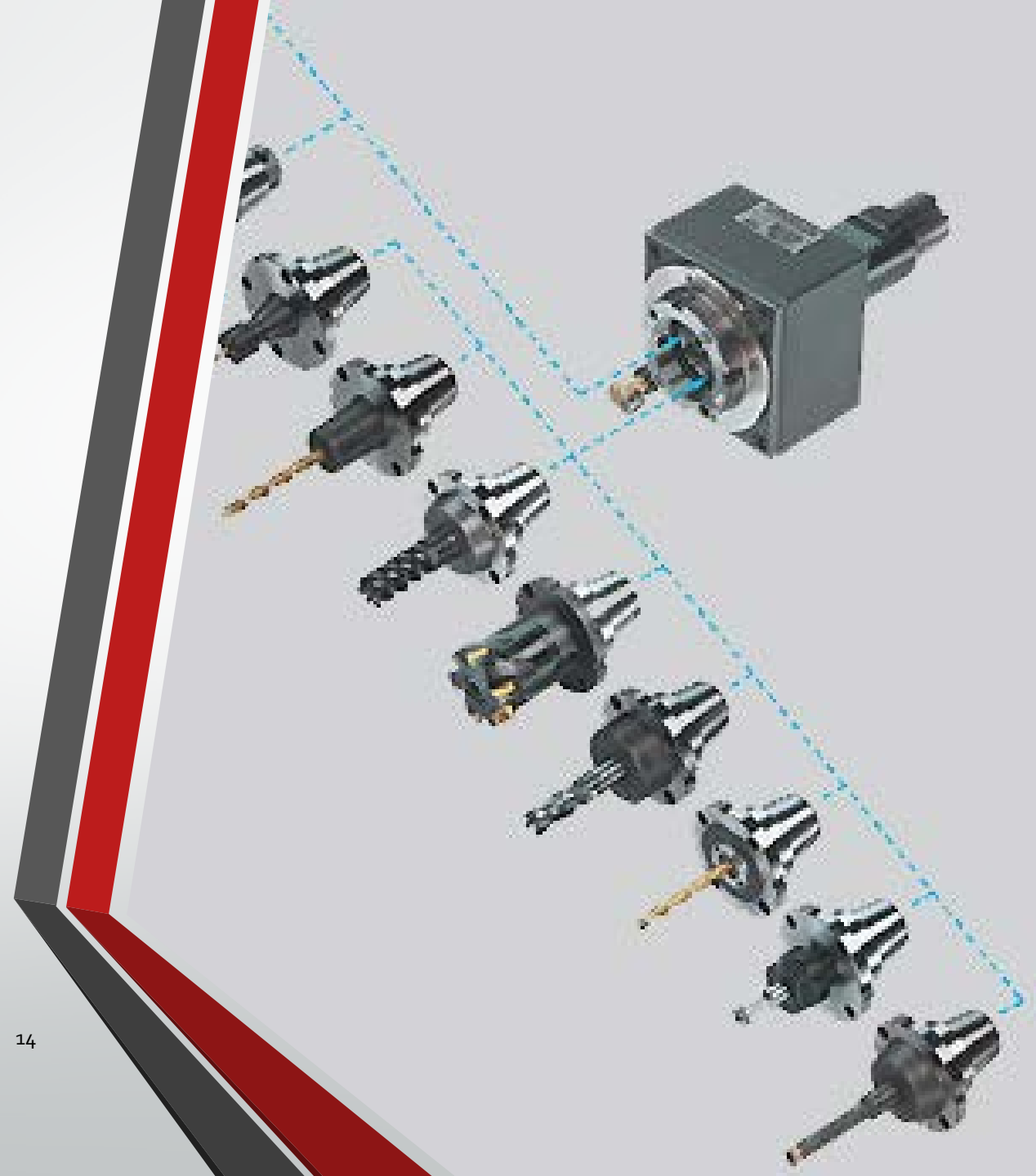
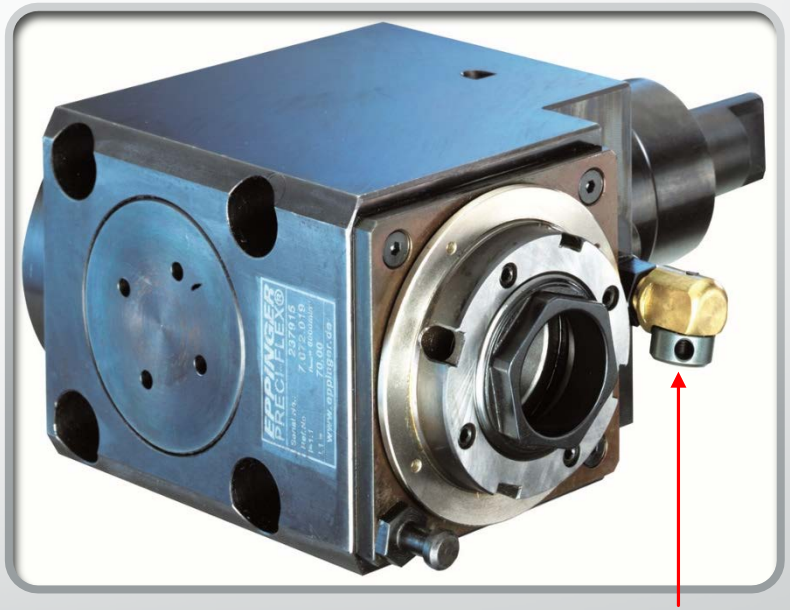


Photo Source: Exsys Tool. N.d. Web. 29 January 2015.<http://www.exsystool.com/images/catalog/archive/haas-tooling-catalog.pdf>

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.

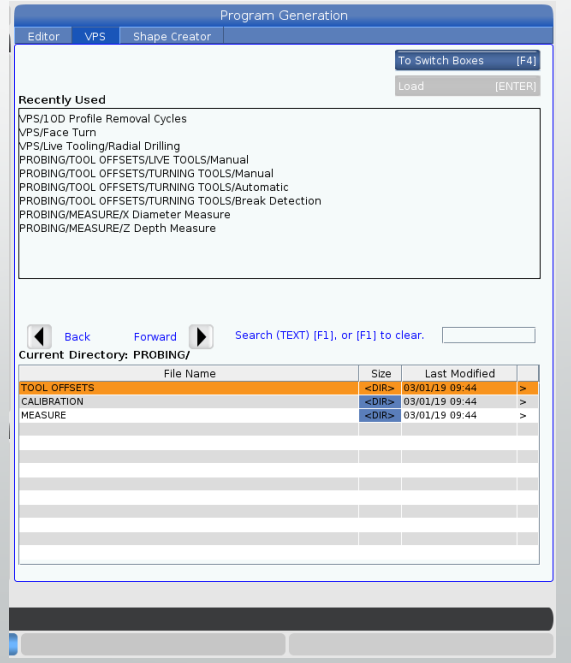


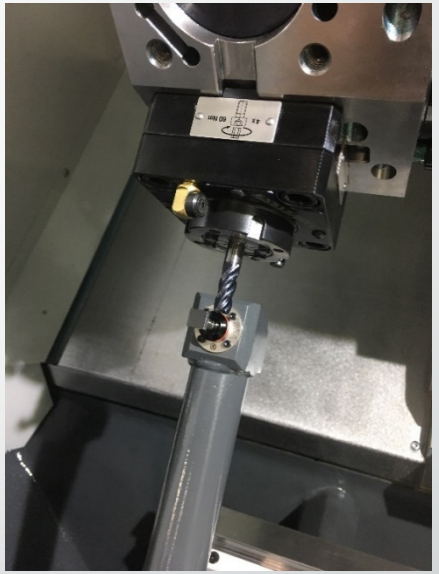
Coolant should be adjusted here when setting the tool up.

Photo Source: Exsys Tool. N.d. Web. 29 January 2015.<http://www.exsystool.com/images/catalog/archive/haas-tooling-catalog.pdf>

Live Tooling Geometry Offsets

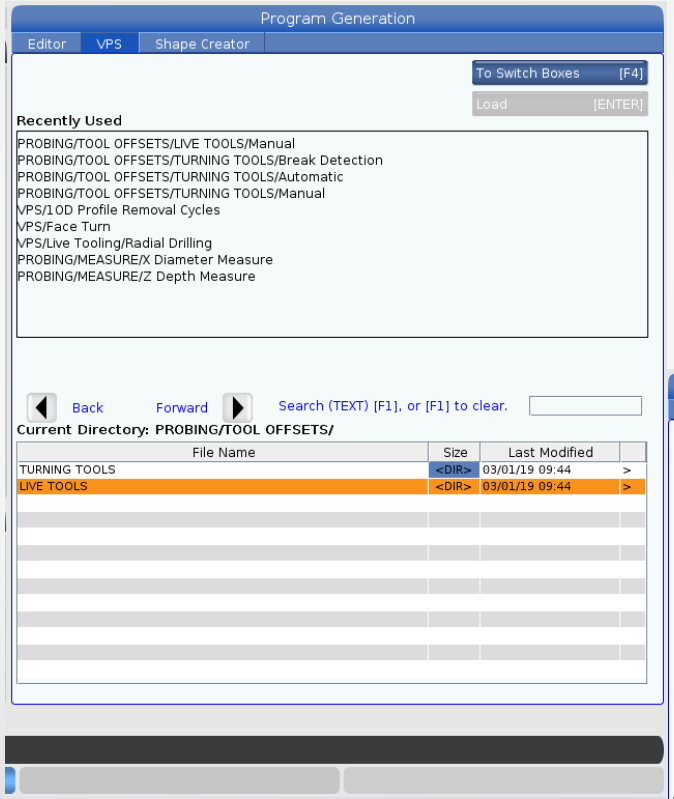
- Both radial and axial live geometry offsets can be set with the tool setter.
- Like regular turning tools, VPS can be used to set the tool offsets.

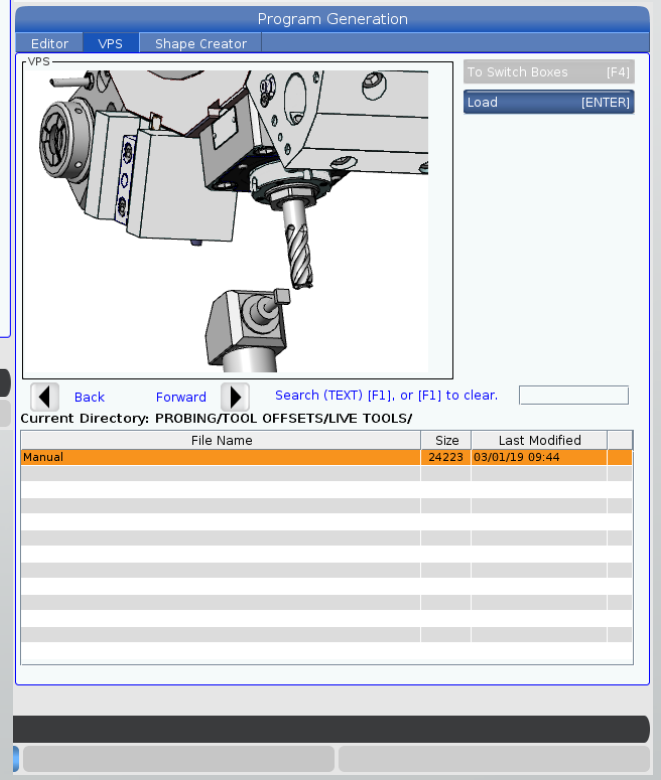




Live Tooling Geometry Offsets

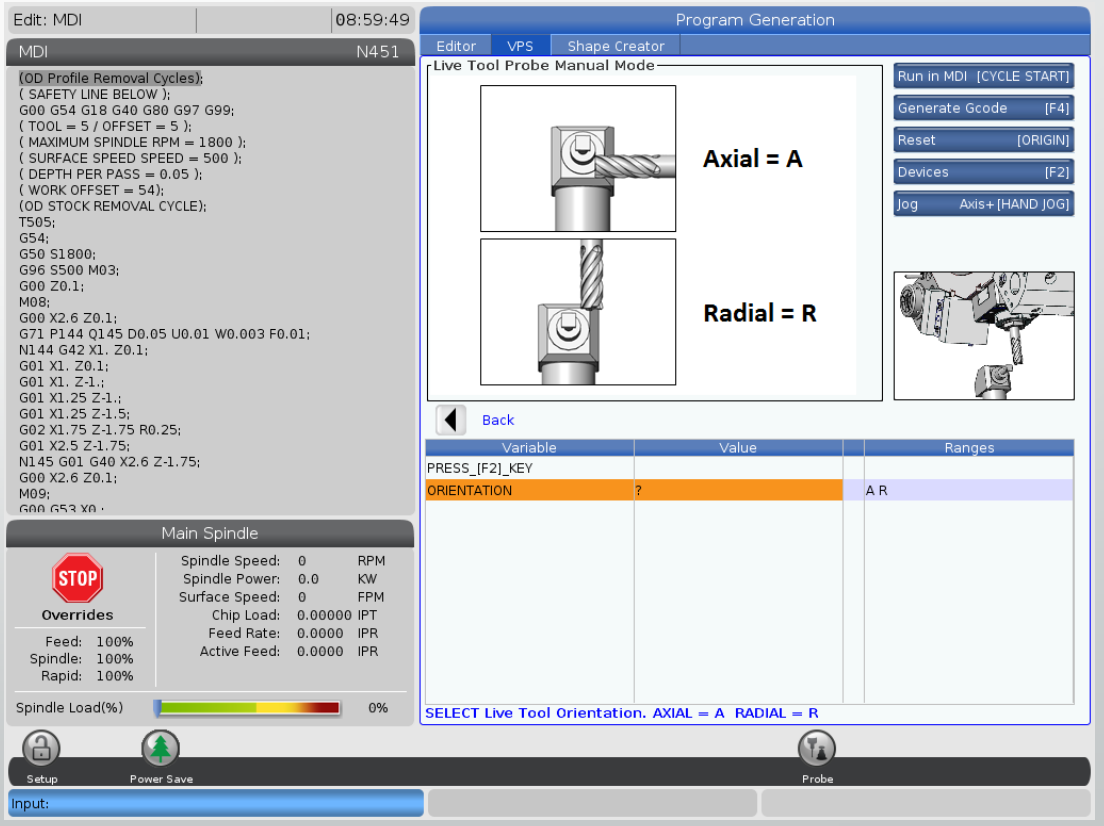
- After selecting VPS, enter the live tools option, then enter manual.
- Manual tool setting is the only option for setting live tool offsets.





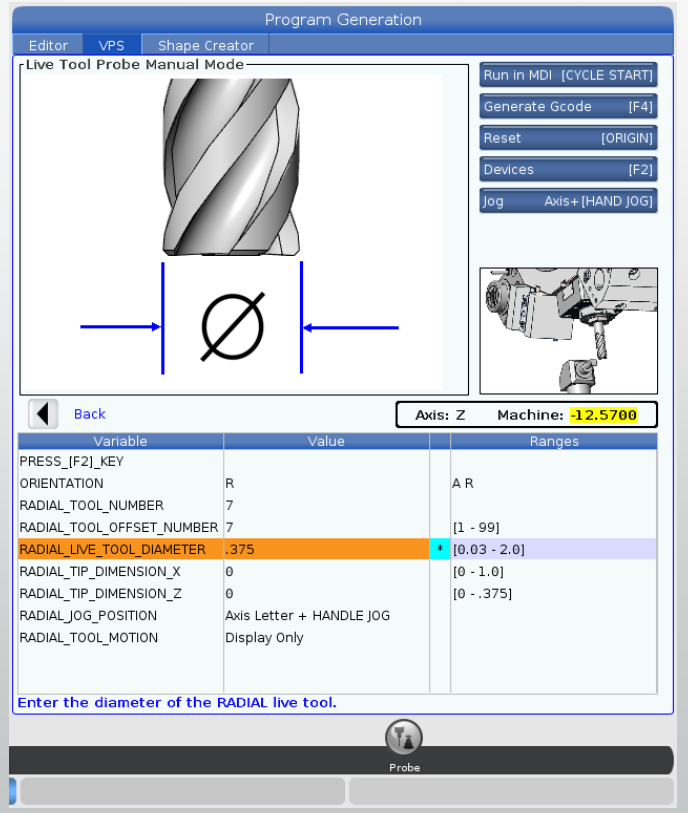
Live Tooling Geometry Offsets

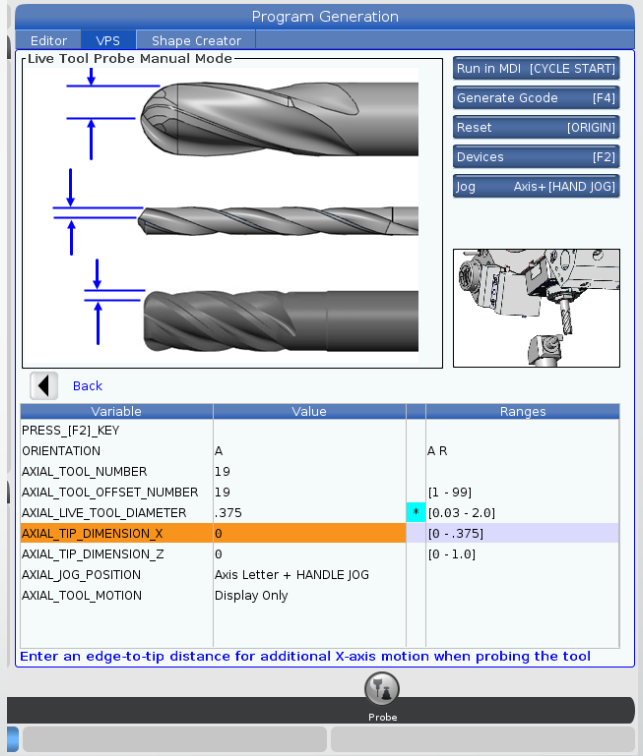
- After entering the manual offset page, the orientation of the tool needs to be established.
  - A- axial tool offset
  - R- radial tool offset



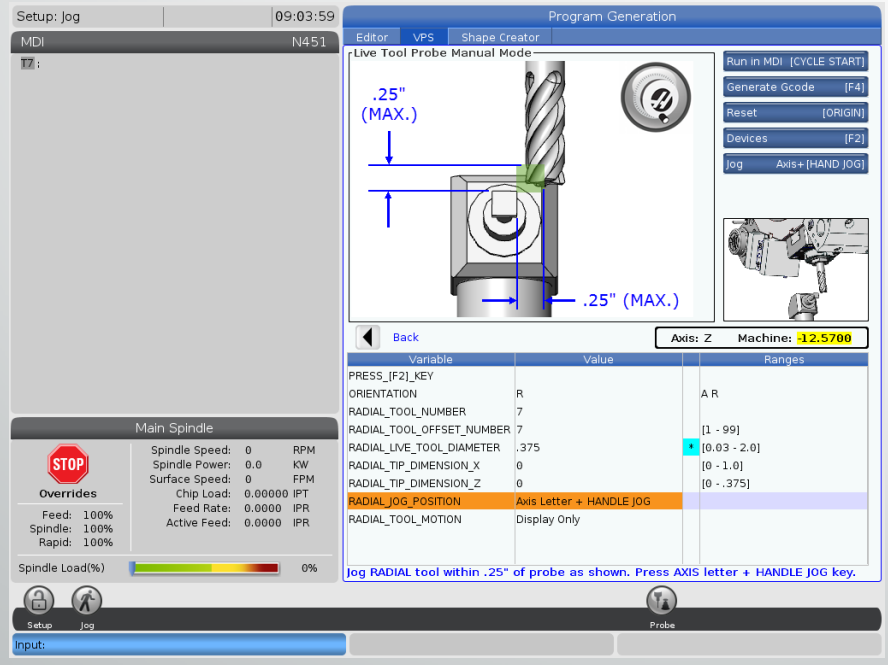
Live Tooling Geometry Offsets

- 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.





Live Tooling Geometry Offsets

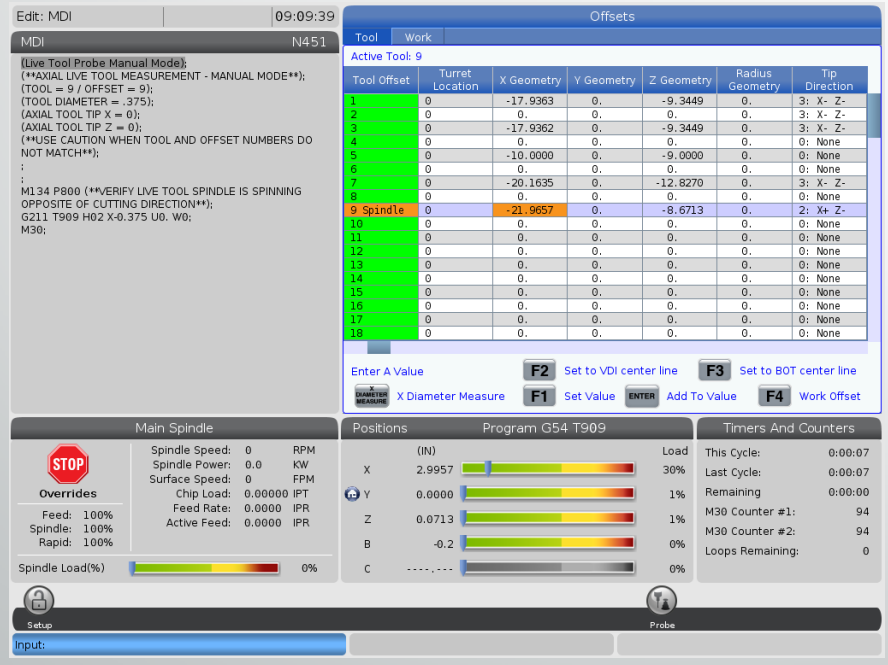


- F2 can be used to lower the tool probe.
- Selecting an axis and then pressing hand jog allows the user to hand wheel the tool to the probe without leaving the screen.
- After the tool is in place, F4 can be used to output the code. In this case, it is output to MDI.

Live Tooling Geometry Offsets

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

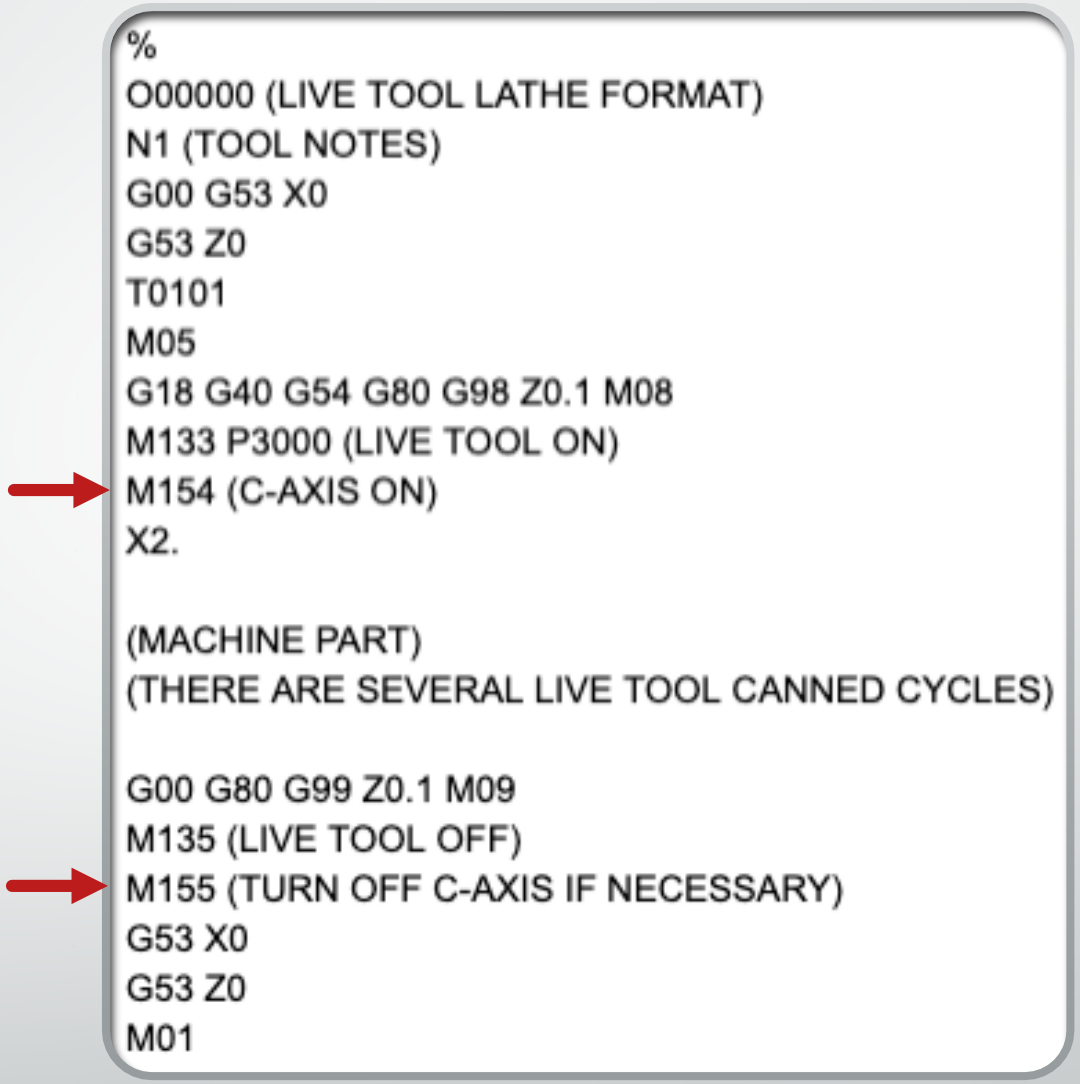
Starting the Live Tooling

- **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.
```

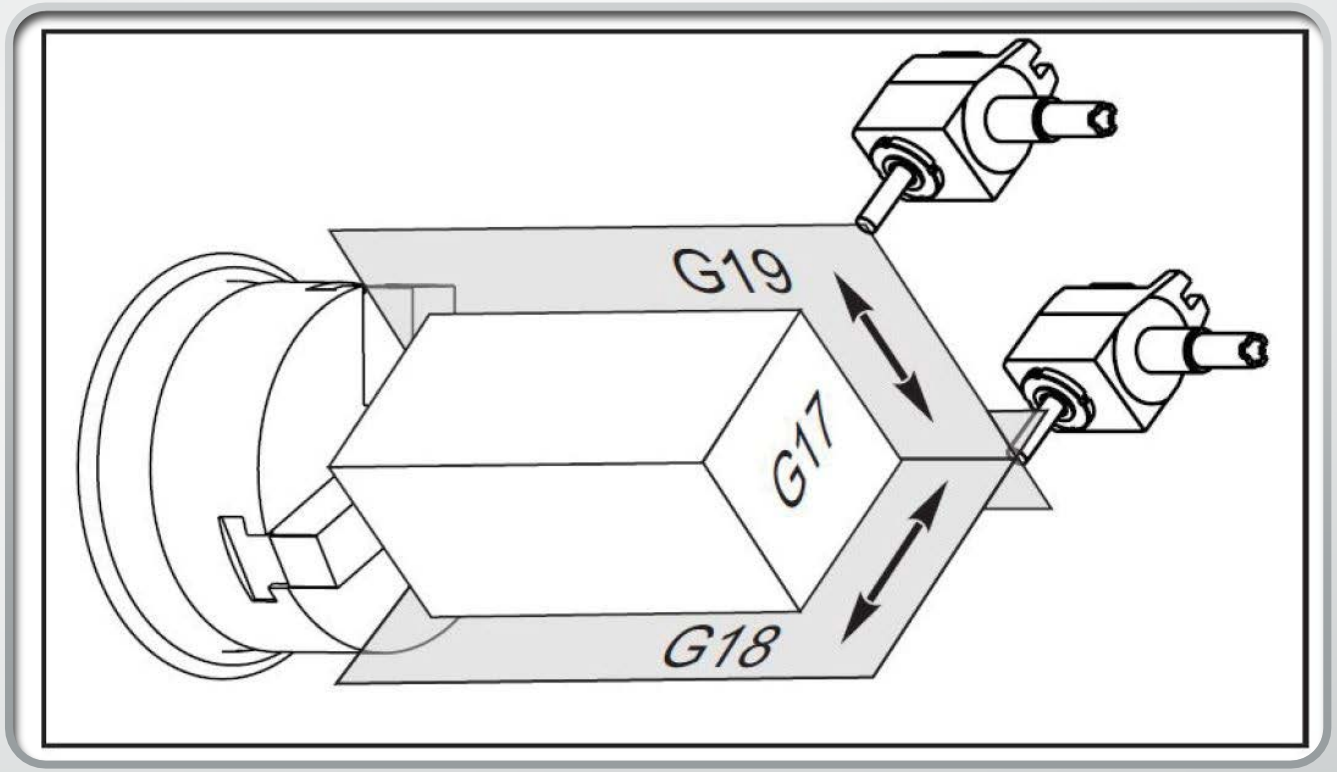
C Axis Engagement

- **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.



Plane Selection

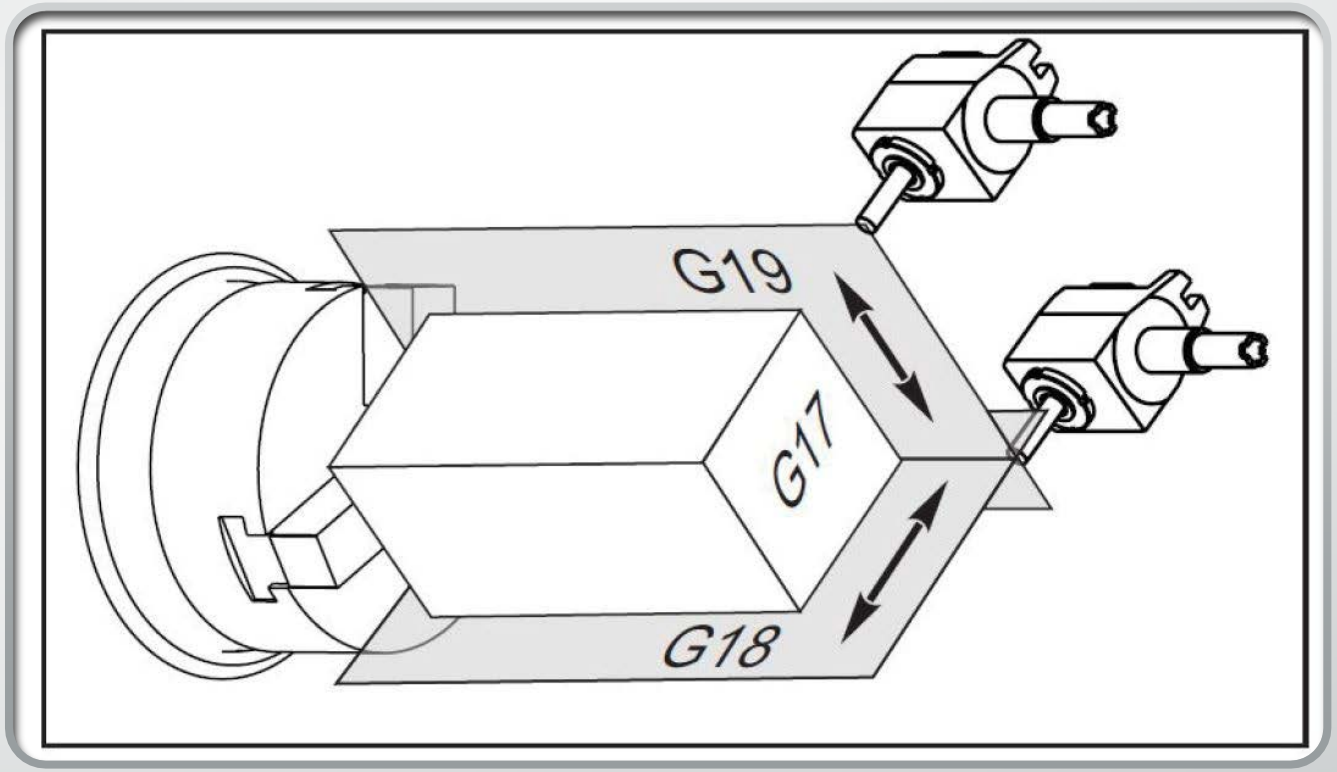
- Different planes can be used to make different canned cycles and milling tool paths work correctly on a live tooling lathe.
- **Drill or Mill** – refers to face work or axial work on a live tooling lathe.
- **Cross Mill or Cross Drill** – refers to work on the OD or radial work on a live tooling lathe.



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

Plane Selection

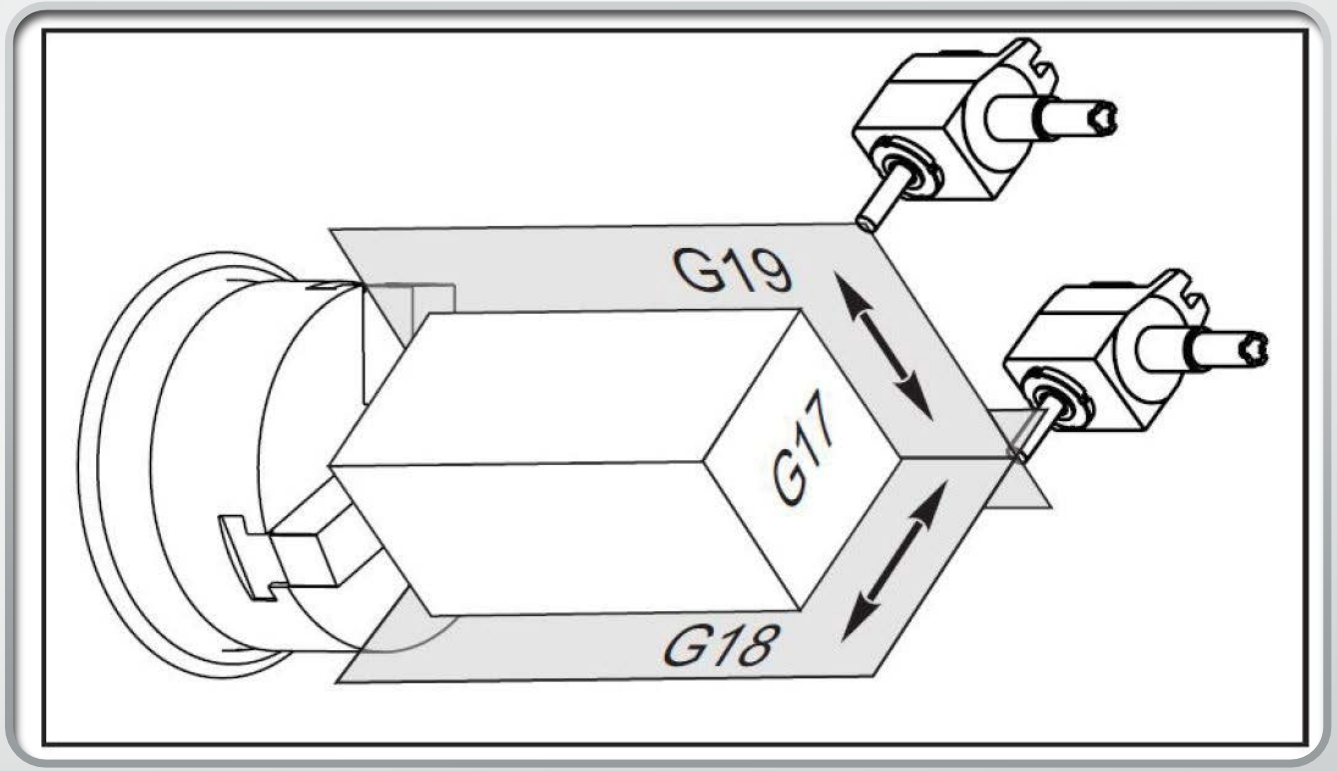
- **G17, G18, G19**
  - What are these codes?
    - **G17** – XY plane
    - **G18** – XZ plane
    - **G19** – YZ plane



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

Plane Selection

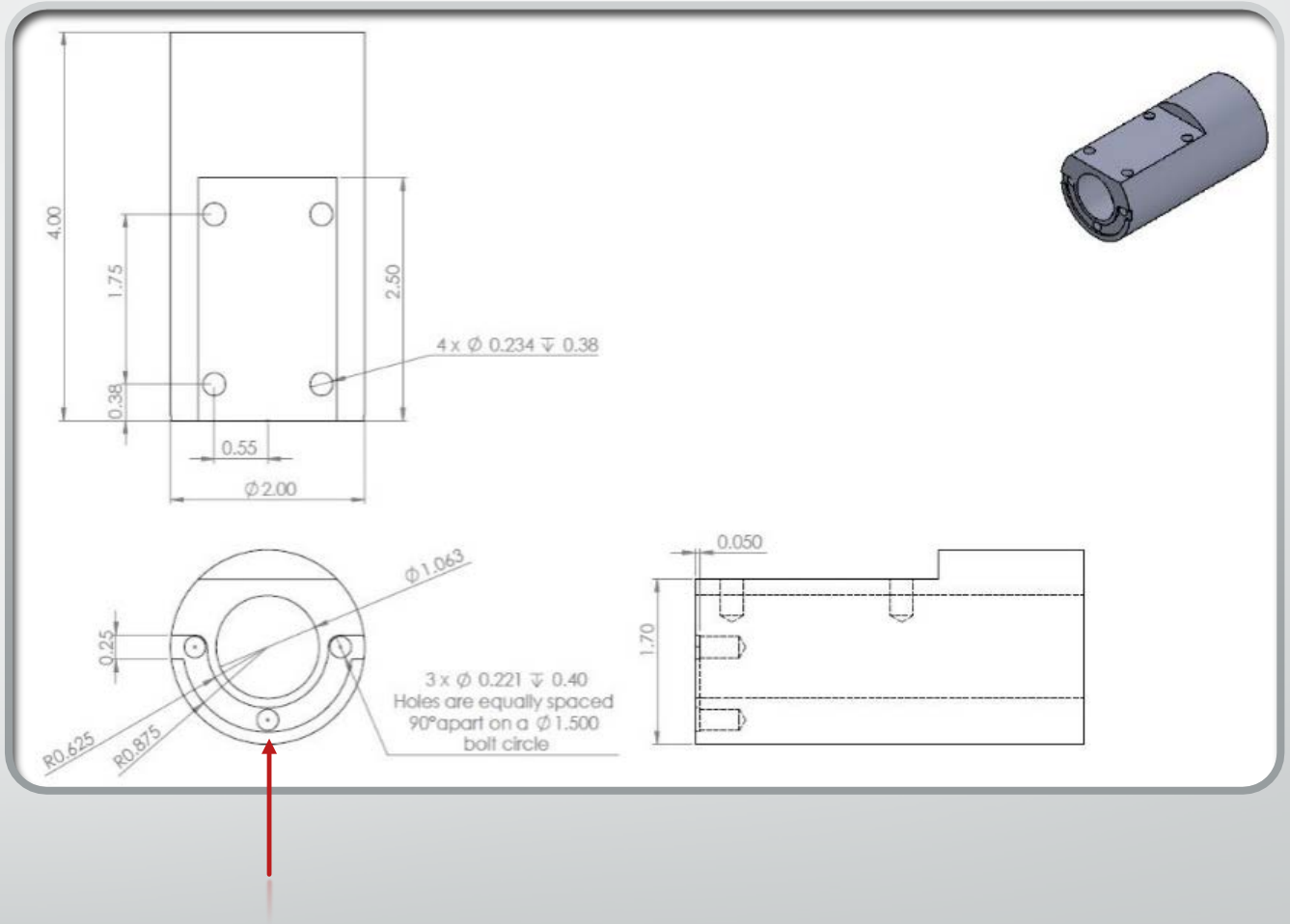
- **What do these planes mean?**
  - Plane selection (**G17, G18, G19**) determines the main plane of work.
  - Think of these planes as axis selection:
    - **G17** - the Z axis positions and the main cutting axis would be X and Y.
    - **G18** - the Y axis positions and the main cutting axis would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



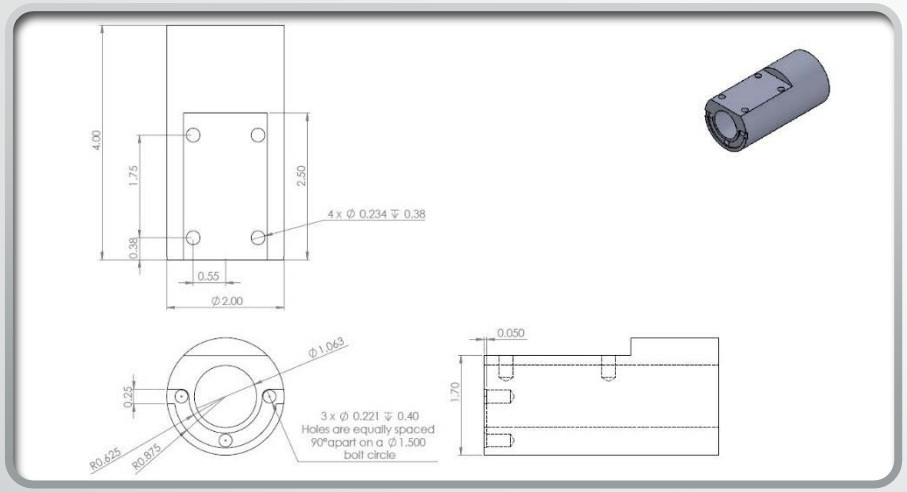
Productivity Inc. "*Plane Selection.*" *Live Tool For has Lathes*. N.p.: Productivity *Inc.*, 2012. 10. Print.

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

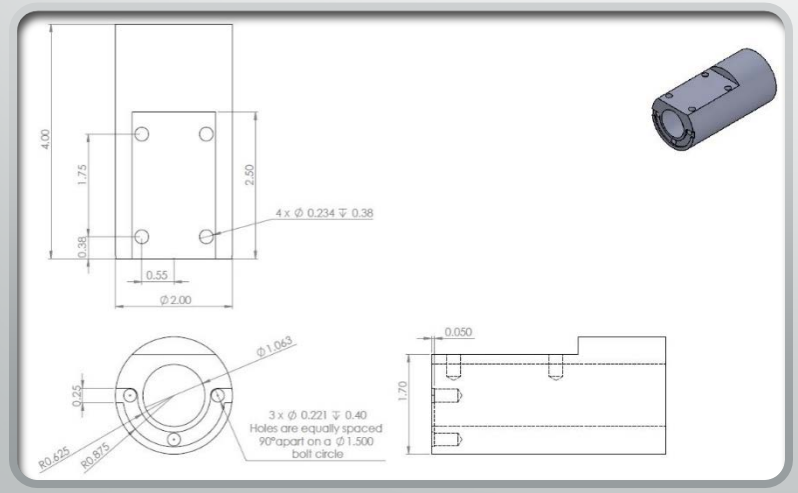


- The turning, drilling and boring for the part would all happen with the **G18** plane.
- **G18** could also be used for the XC milling for the slot on the front face.

- **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 the of part.

G17 Plane Selection Guide

Cycle	Description	Feedrate
XY AXIS MILLING	MANUAL FACE PROGARMMING	G98
G112	CARTIESAN TO POLAR	G98

G18 Plane Selection Guide

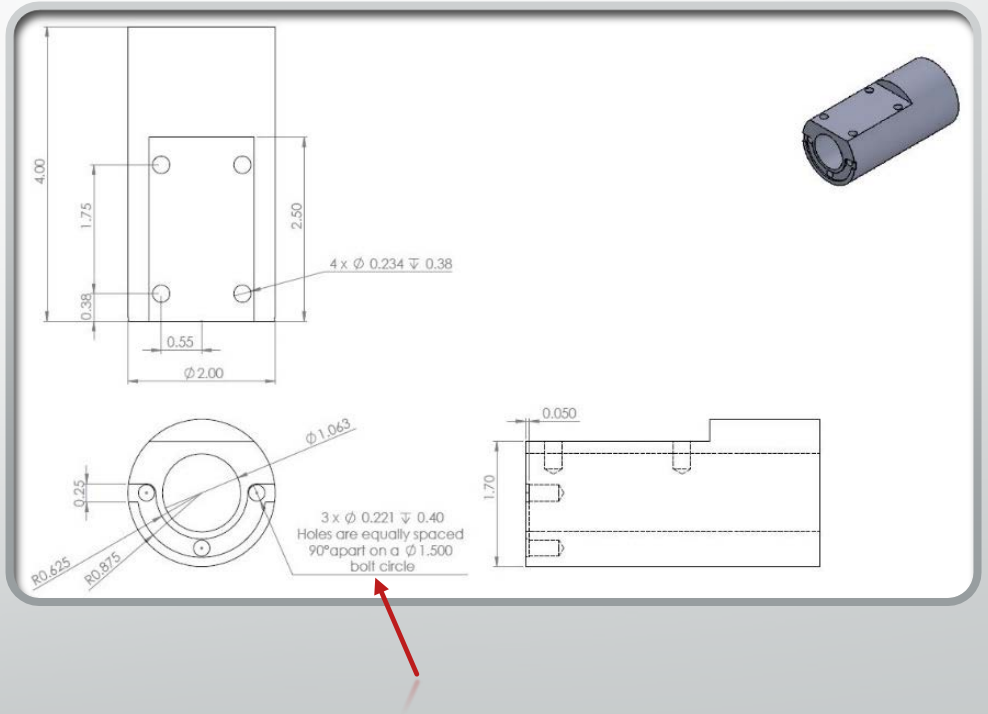
Cycle	Description	Feedrate
G81	SIMPLE DRILLING	G98
G82	SIMPLE DRILL W/ DWELL	G98
G83	DRILL W/ PECK	G98
G95	RIGID TAP	G99
G186	RIGID TAP LEFT HAND	G99
X-C MILLING	FACE MILLING	G98

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 (DRILLNG 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 (DRILLNG 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

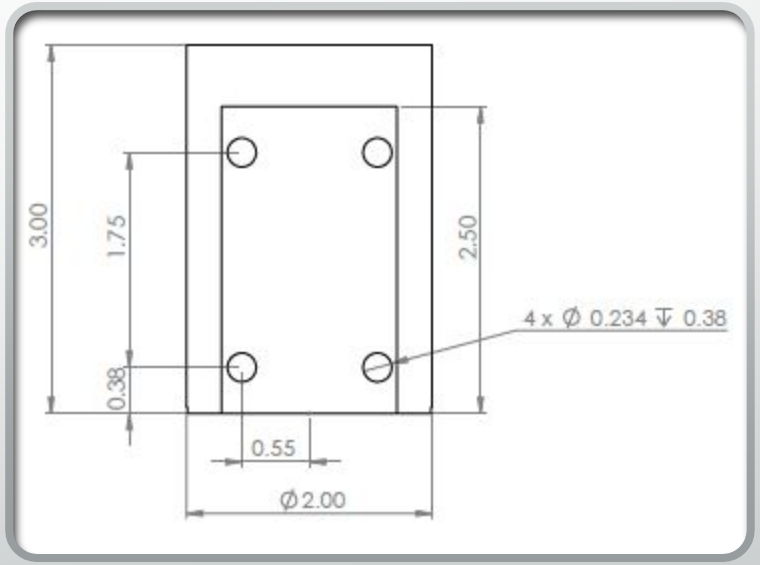
G186 Reverse Live tooling rigid tap (face)

- 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

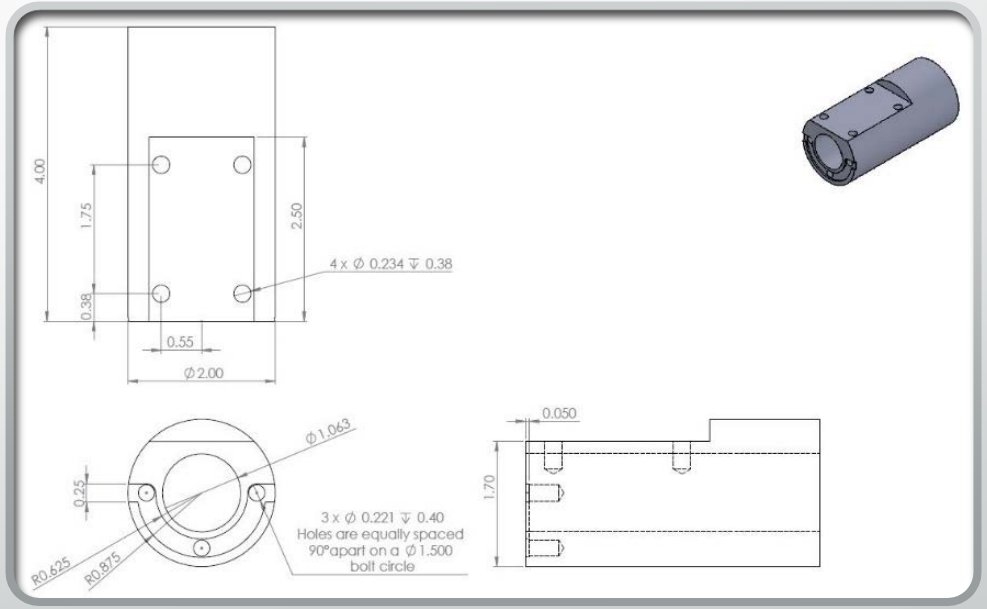
- Cross drilling takes more thought and calculation for a successful toolpath.
  - The depth is controlled by X for these paths.
    - X is read by the machine as a diameter.
    - For this to work, more math will need to be done.

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
```

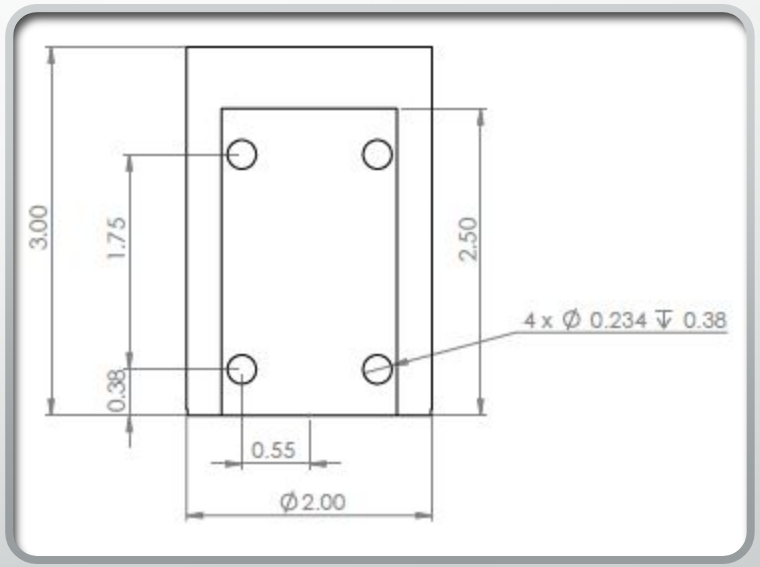
Calculating Drill Depth

- 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

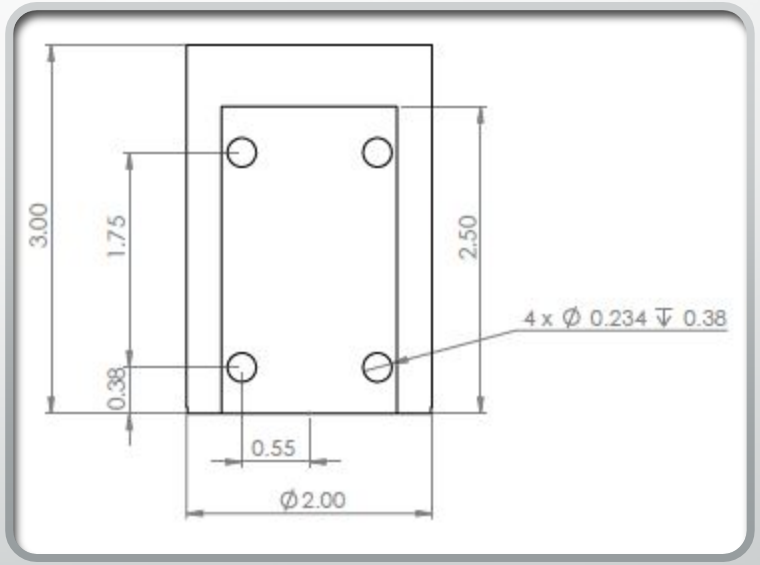
- **G19** is used because Y and Z are the positioning axis.
- This makes X the retract axis for the canned cycle.
- If **G18** was used, the automatic retract would happen in Z and result in a broken drill.

```
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 programed to diameters and not radii.

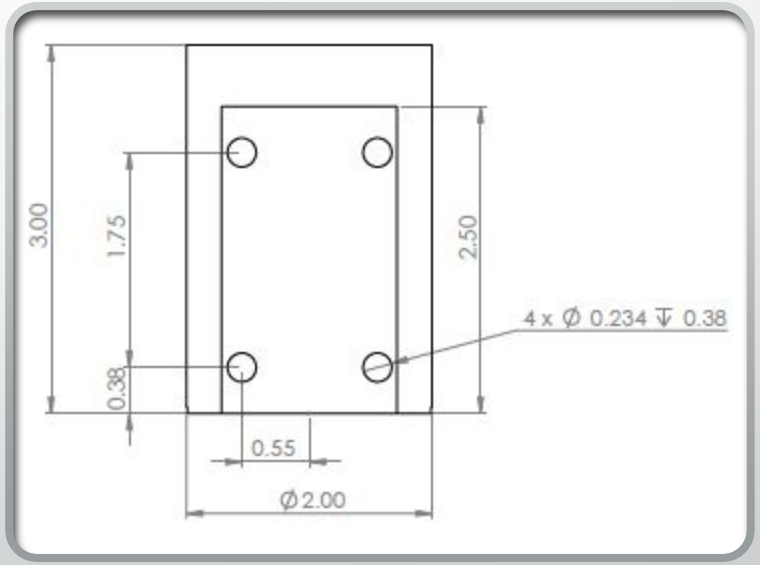


OD and Radial Drilling

Radial Drilling using G241:

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

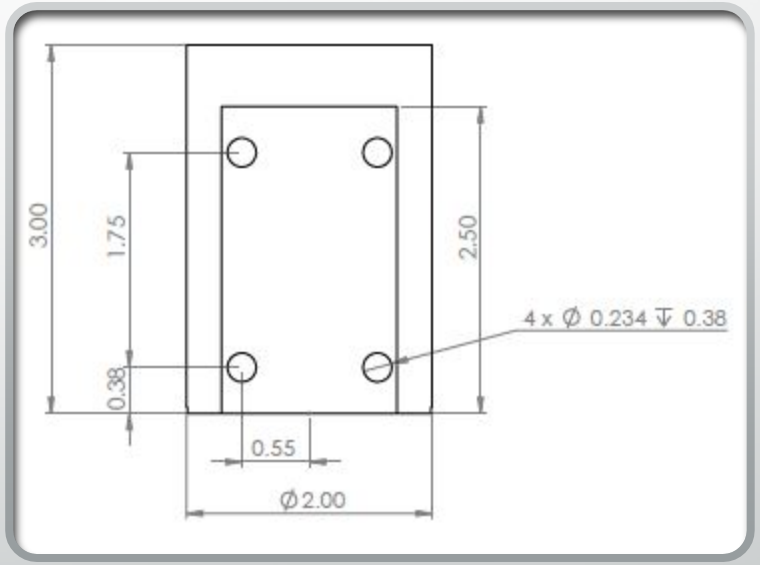
C C-axis absolute motion command
F Feed Rate Inch/mine
R Position of the R plane (diameter)
*X Position of bottom of hole (diameter)
*Y Y-axis absolute motion command
*Z Z-axis absolute motion command
* indicates optional



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)
```

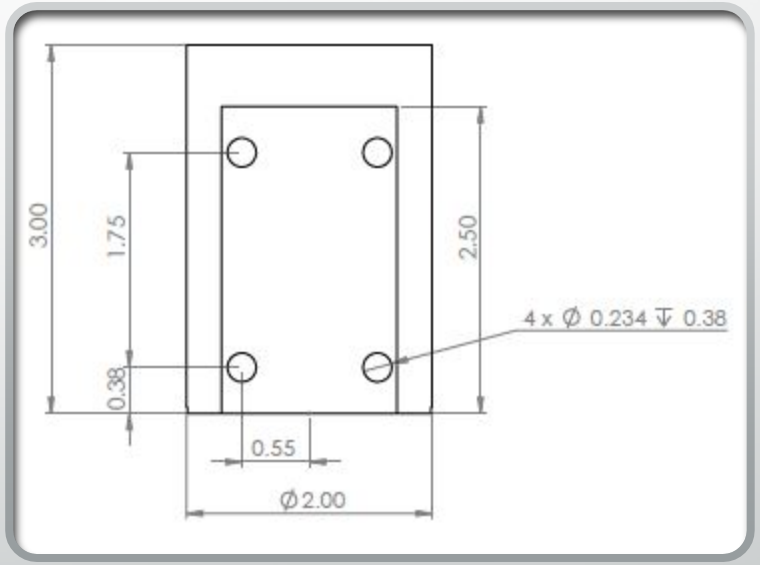


OD and Radial Drilling

Radial Drilling using G242:

G242 Radial Spot Drill Canned Cycle (Group 09)
Drill and dwell canned cycle

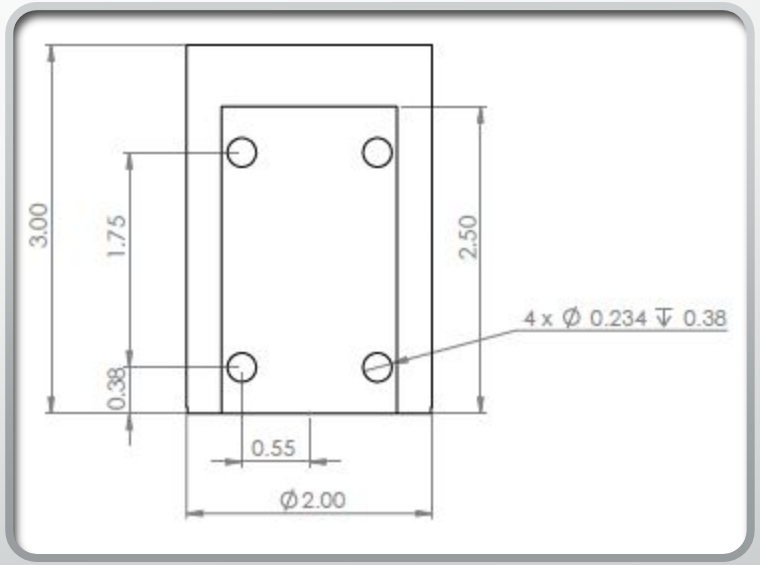
C C-axis absolute motion command
F Feed Rate
P The dwell time at the bottom of the hole
R Position of the R plane (Diameter)
*X Position of bottom of hole (Diameter)
*Y Y-axis motion command
*Z Z-axis motion command
* indicates optional



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)
```



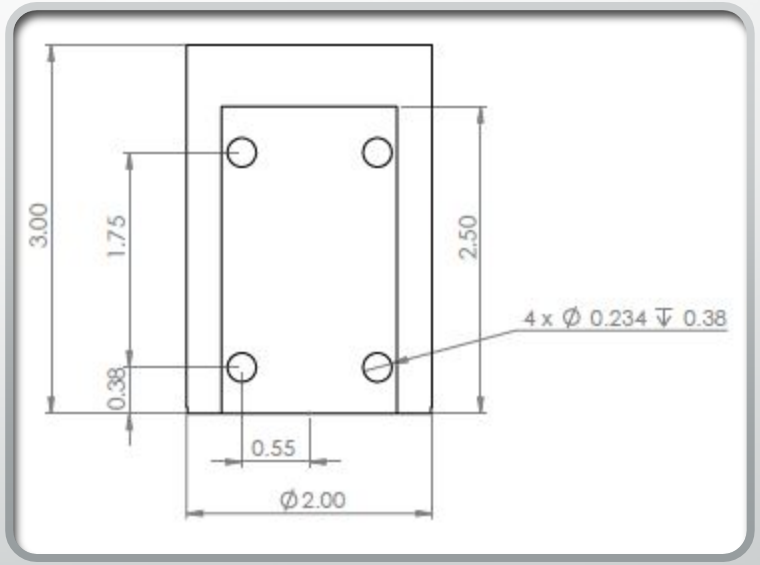
OD and Radial Drilling

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

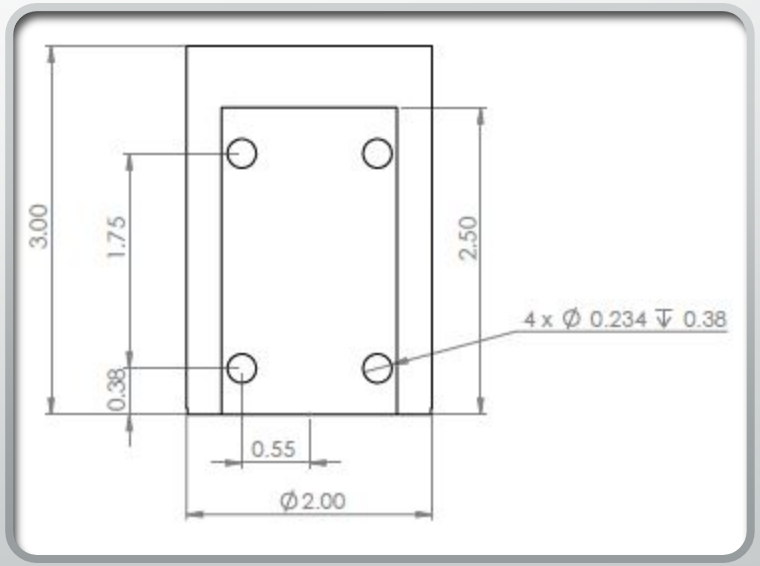


OD and Radial Drilling

Radial Drilling using G243

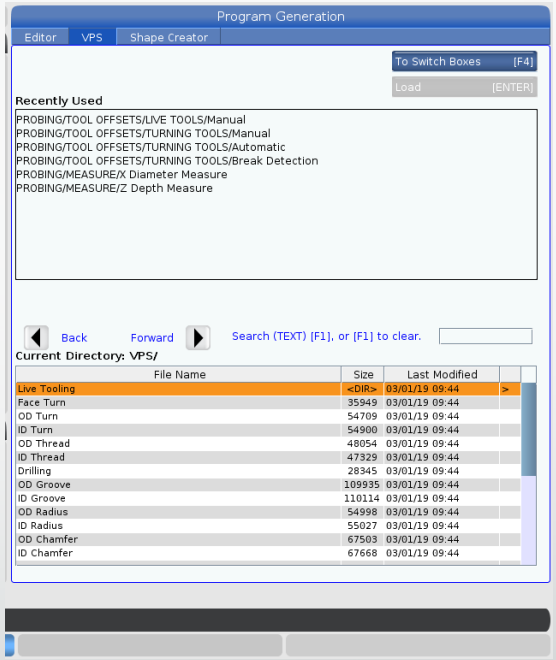
```
M154 (Engage C Axis)
M133 P2500 (2500 RPM)
G19
G98 (IPM)
G00 X5. Z-0.75 Y0
G243  X2.1  Y0.125  Z-1.3  C35.  R4.  Q0.25  F20. (Drill to X 2.1)
```

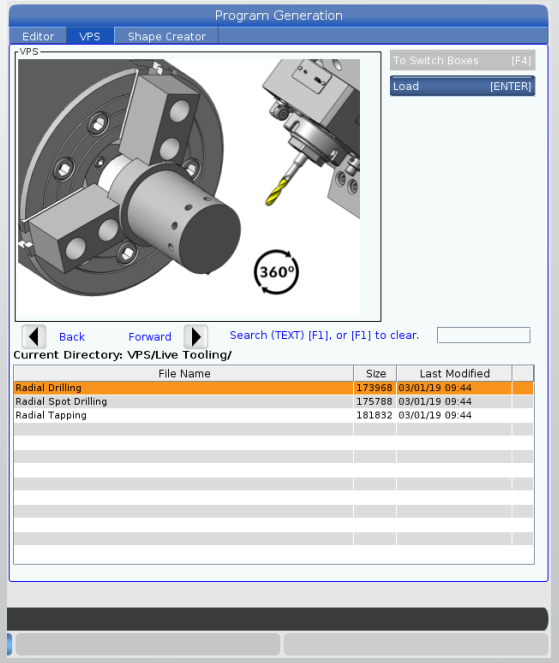
- Setting 22 is the amount to feed in X to get the same point at which the retract occurred.



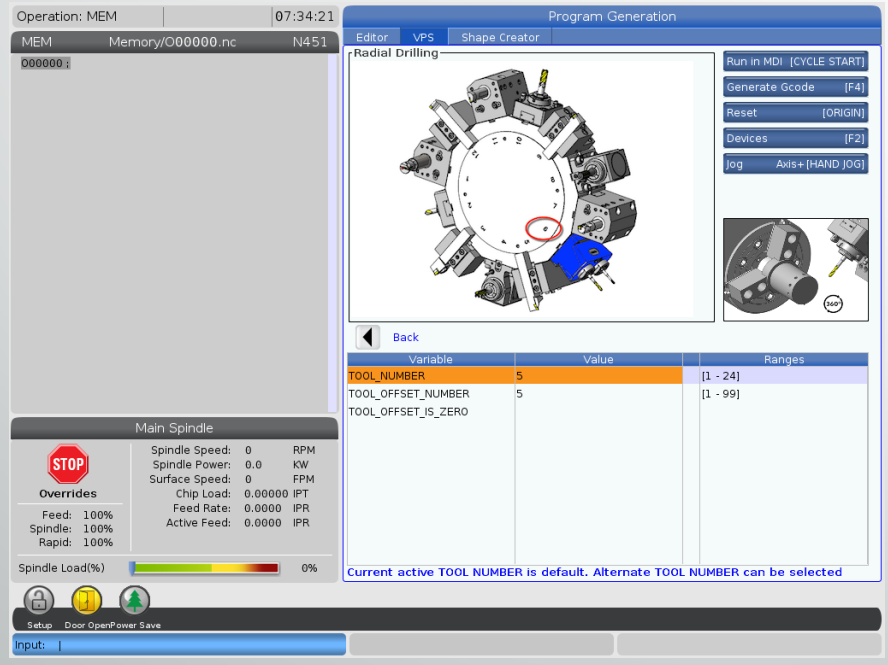
Radial drilling w/VPS

- 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





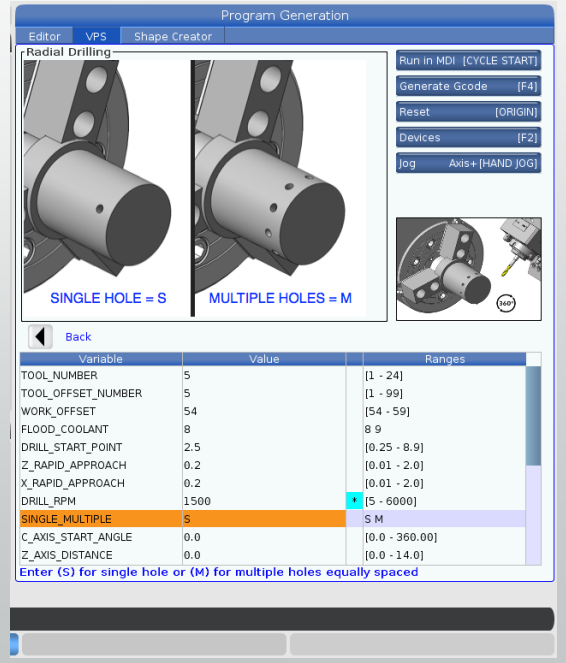
Radial drilling w/VPS

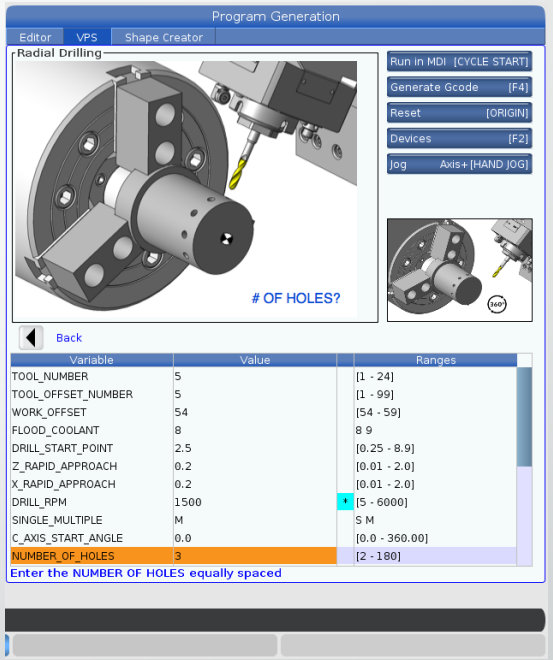


- After entering the radial drilling option, the tool number will need to be entered.
- After the tool number is entered, the rest of the command fields will appear.

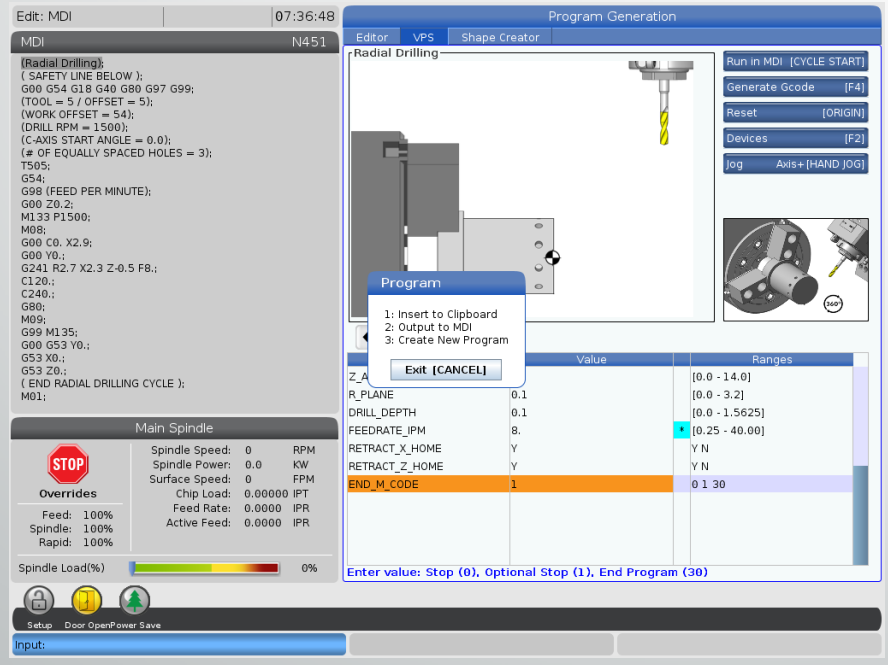
Radial drilling w/VPS

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





Radial drilling w/VPS



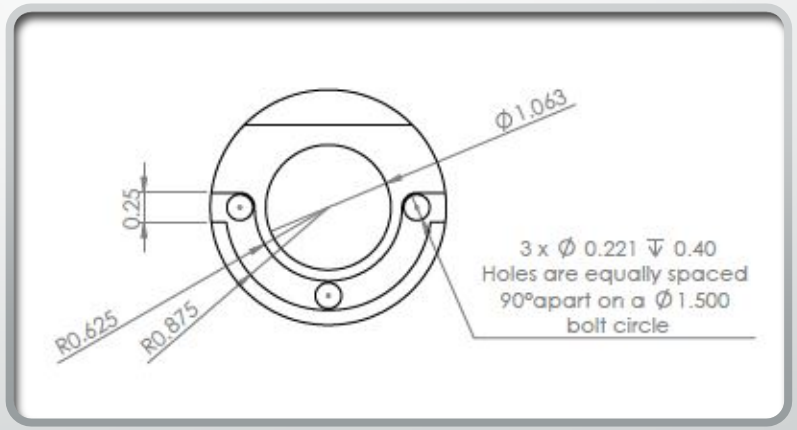
- After all the variables are input, F4 can be used to generate the code. In this case, option 2 will be used to output the code to MDI.

Radial drilling w/VPS

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

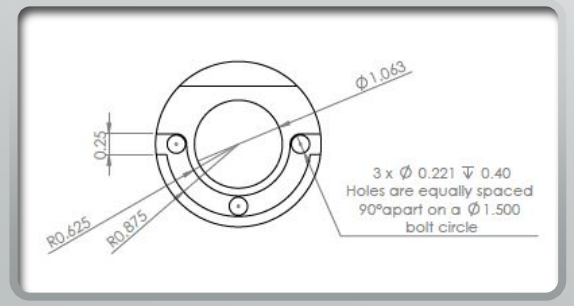
Axial or Face Milling

- 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° and the exit at 270°.
- A ¼" Endmill is used to mill the slot in one pass at the desired depth.
- **G98** is used because of the milling operation.

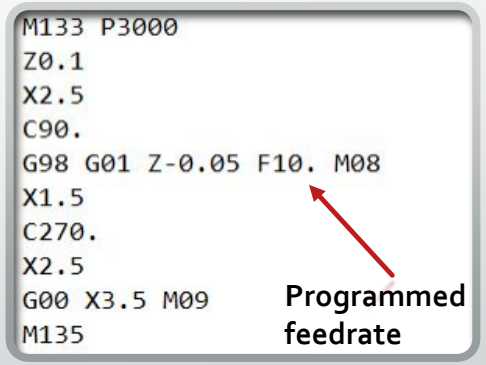


Entry point →
Exit point →

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

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.



Milling the Face Slot

- Set the entrance of the groove to 90° and the exit at 270°.
- A ¼" 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
```