Welcome to Trident Machine Tools

Lathe Program Troubleshooting

Lathe Program Troubleshooting

- This one day course is designed to provide the user with a basic understanding of lathe programming and troubleshooting common program issues.

Schedule

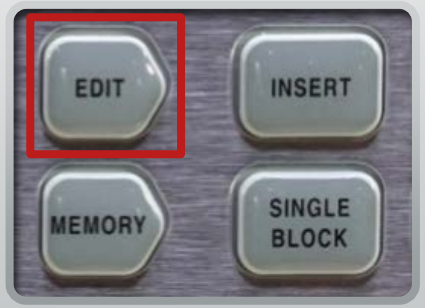
- Introductions
- VPS overview
  - VPS demo
- Break
- Program structure
- Canned cycles
- Lunch
- Canned cycles
- Break
- Program trouble shooting
- Questions

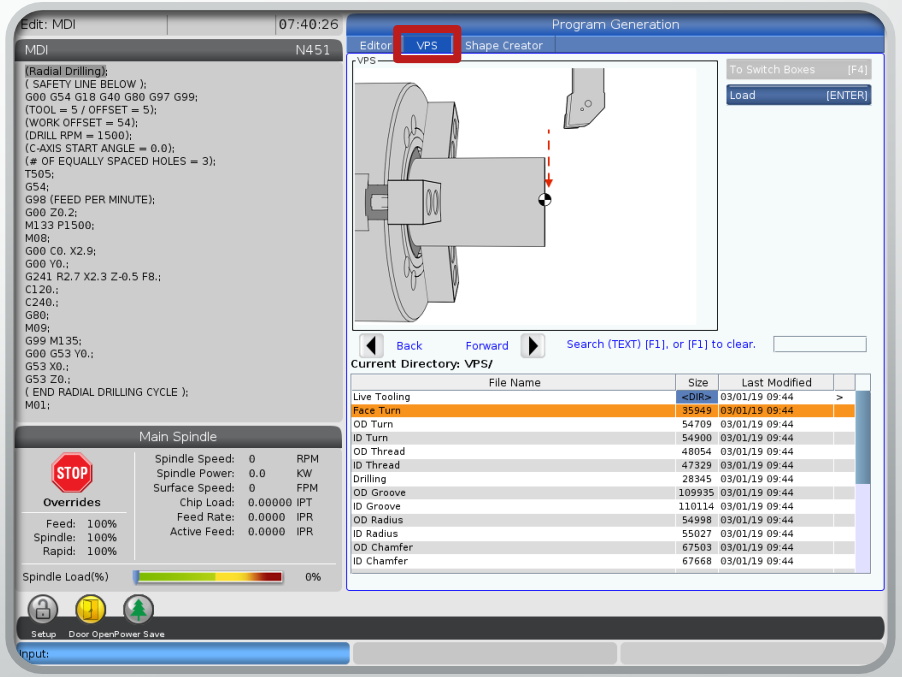
VPS Programming Overview

- VPS allows for conversational programming. This entails inputting values in a menu to output the needed code for the particular situation.
- For a lathe this might entail canned profile cuts, facing, drilling, among other options.

VPS Example

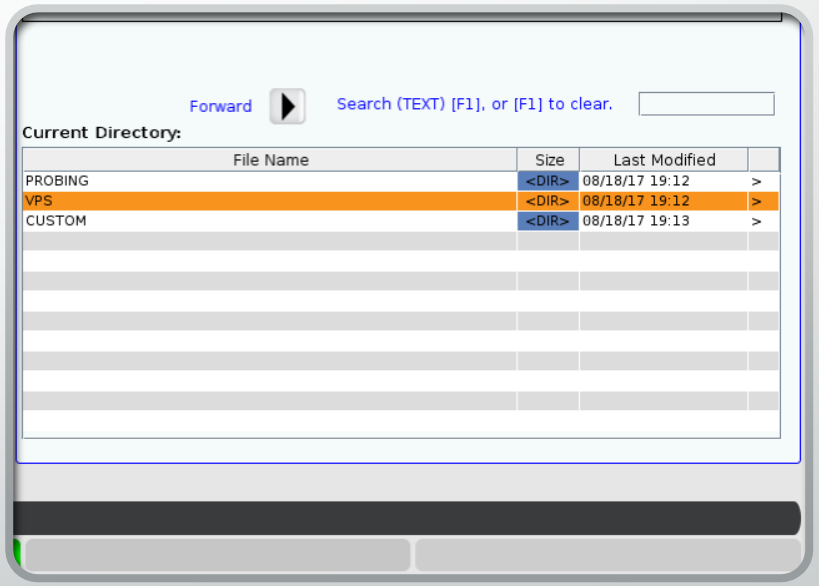
- The following is an example in how to make a VPS program, specifically, contour milling.
- Start by selecting the "Edit" button, then shift over to VPS from editor.





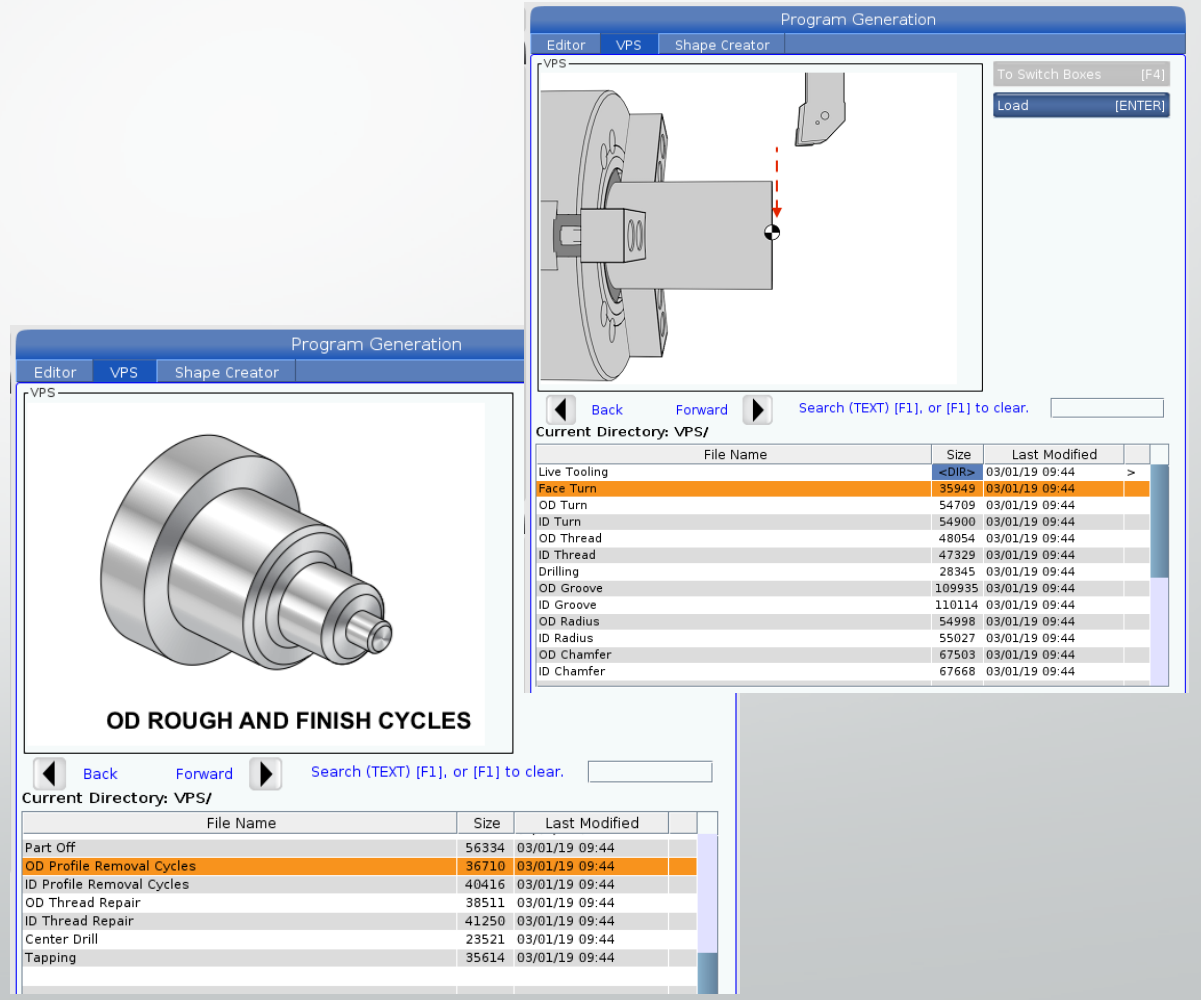
VPS Example

- Cursor down to VPS in the menu and select it.
  - Don't forget that the arrow keys are used to navigate the Haas menus.



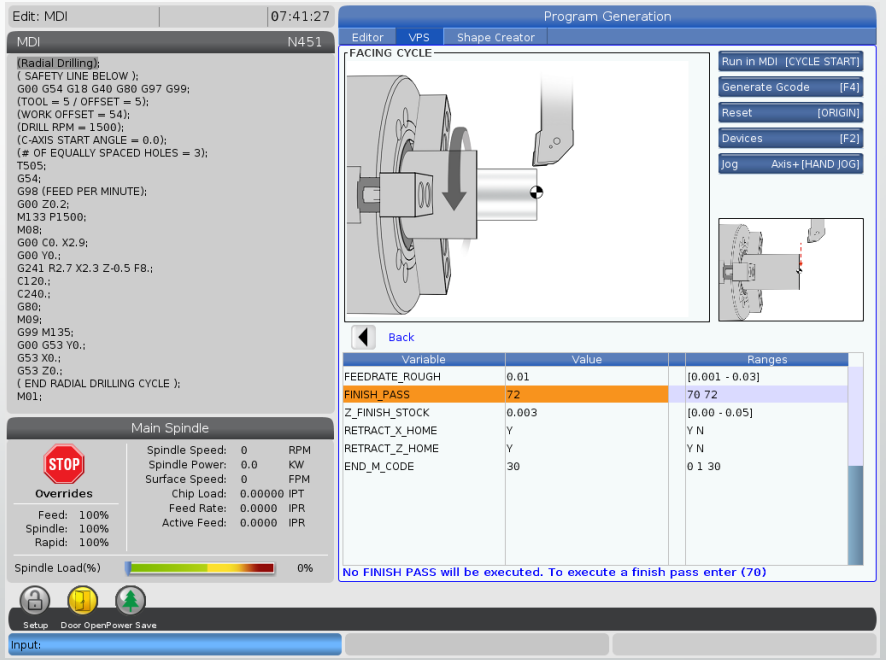
VPS Example

- The following are cycles that can be programmed using VPS. For demonstration we will use “Face Turn”.
- Use the right arrow key to enter the cycle.



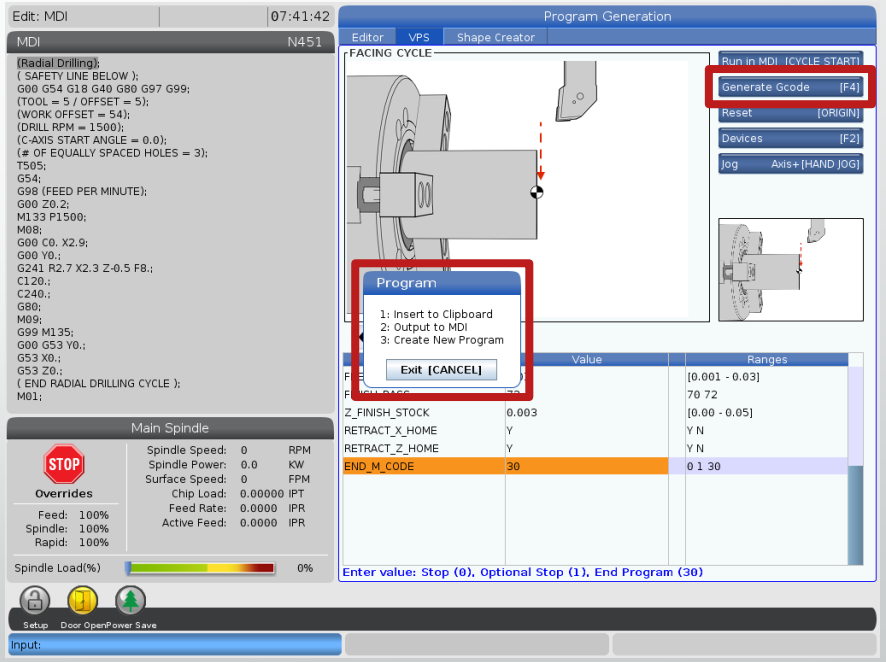
VPS Example

- Start filling out the variables in the form.
- The prompts will walk the operator through each inquiry.



VPS Example

- After the variables are filled in, it should look something like this.
- Select F4 , generate code to bring up the output menu.
- Select “2”, which will output the code to MDI.

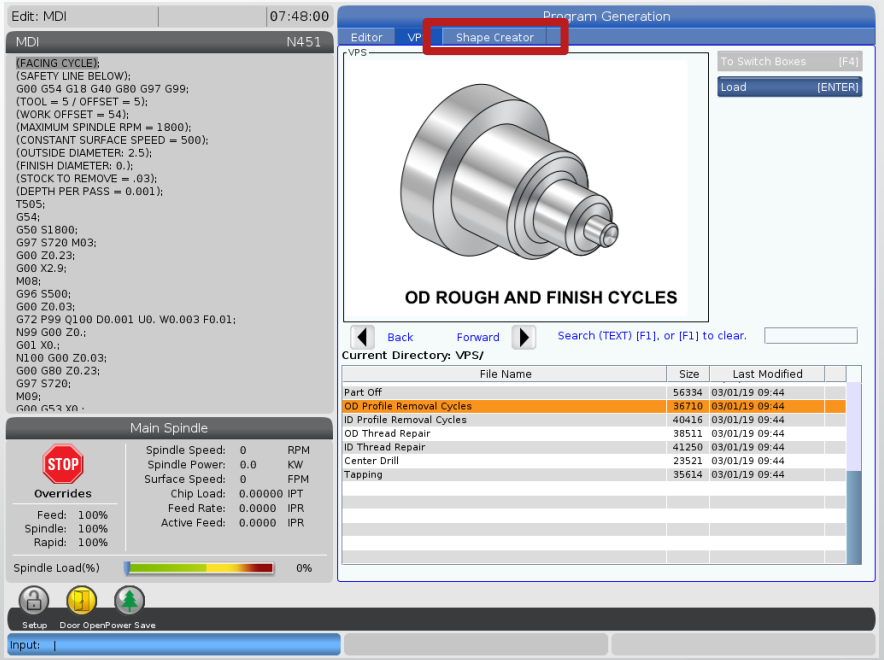


VPS Example

- The program is then output to MDI.

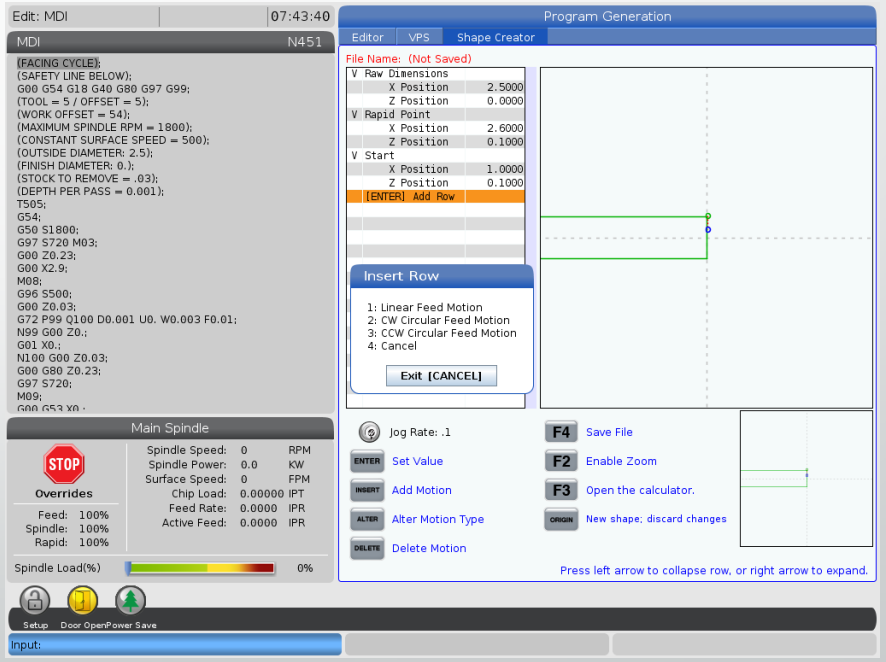
OD Profile Removal Cycle

- Another cycle option is the OD profile removal. This cycle needs the profile defined using the shape creator. Shape creator can be found in the tab next to VPS.
- The shape or OD profile must be created before the OD profile removal cycle can be made.
  - Enter shape creator using the arrow keys on the control.



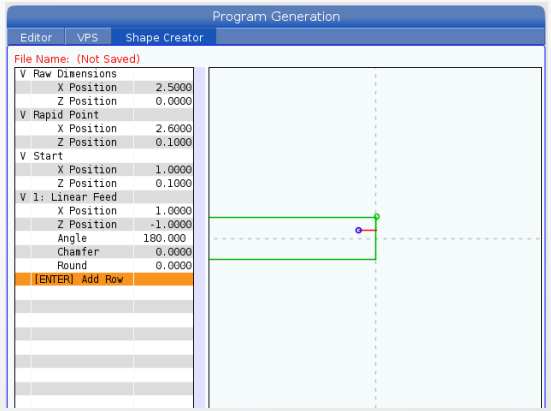
Shape Creator

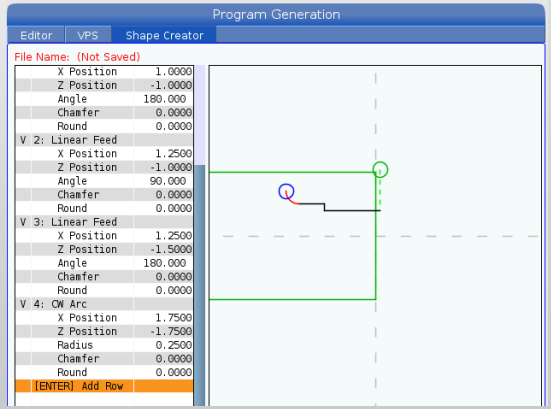
- Once in shape creator, the following will need to be defined:
  - Raw Dimension
  - Rapid Point
  - Start Point
- After those three inputs are defined “Insert” can be used to insert a move.



Shape Creator

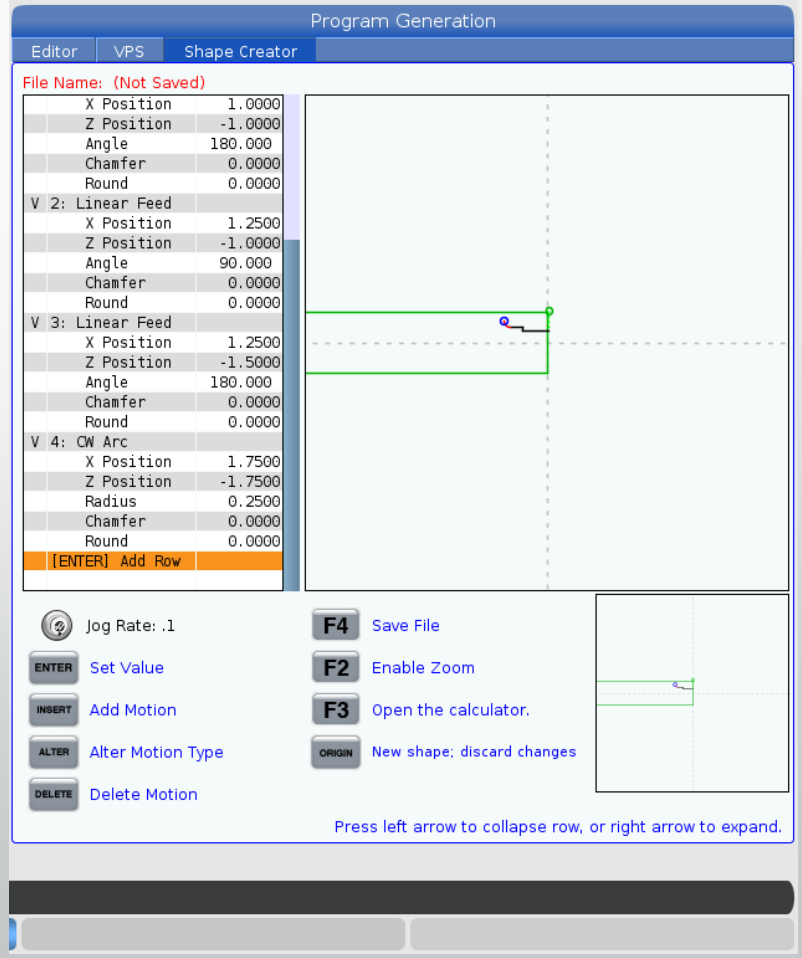
- Linear and circular moves can be input by using “insert”. Each move requires a new line.





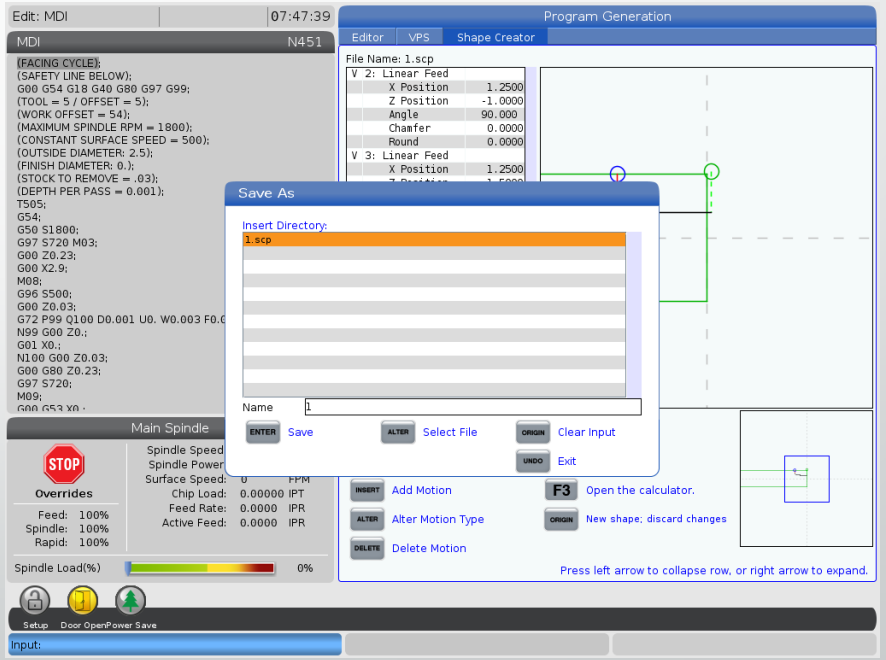
Shape Creator

- Shape creator has numerous options that can be selected. Motions can be altered and deleted. F2 can zoom in on the image if greater detail is needed.



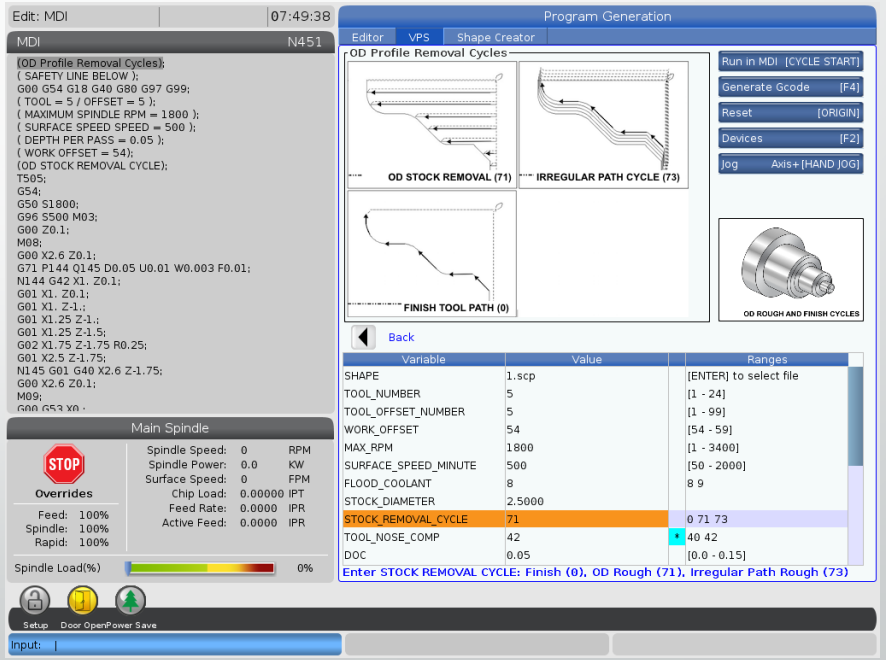
Shape Creator

- Once the profile is defined, it needs to be saved. This can be done using F4, a new screen will come up and require a file name to save as.
  - In this case the profile was called 1.



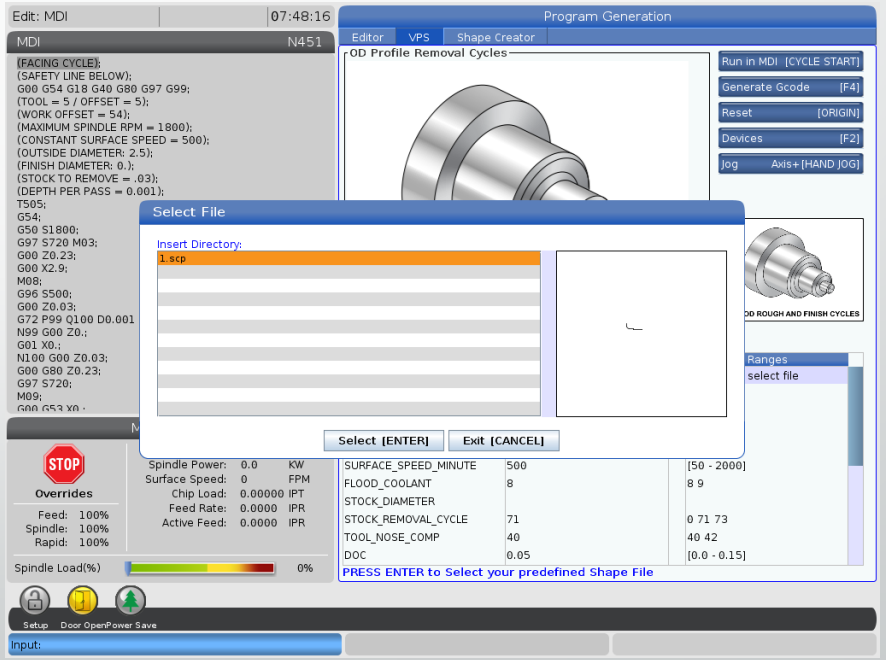
OD Profile Removal Cycle

- With the profile now defined, the OD profile removal cycle can be defined.
- This cycle can be entered through the VPS page.



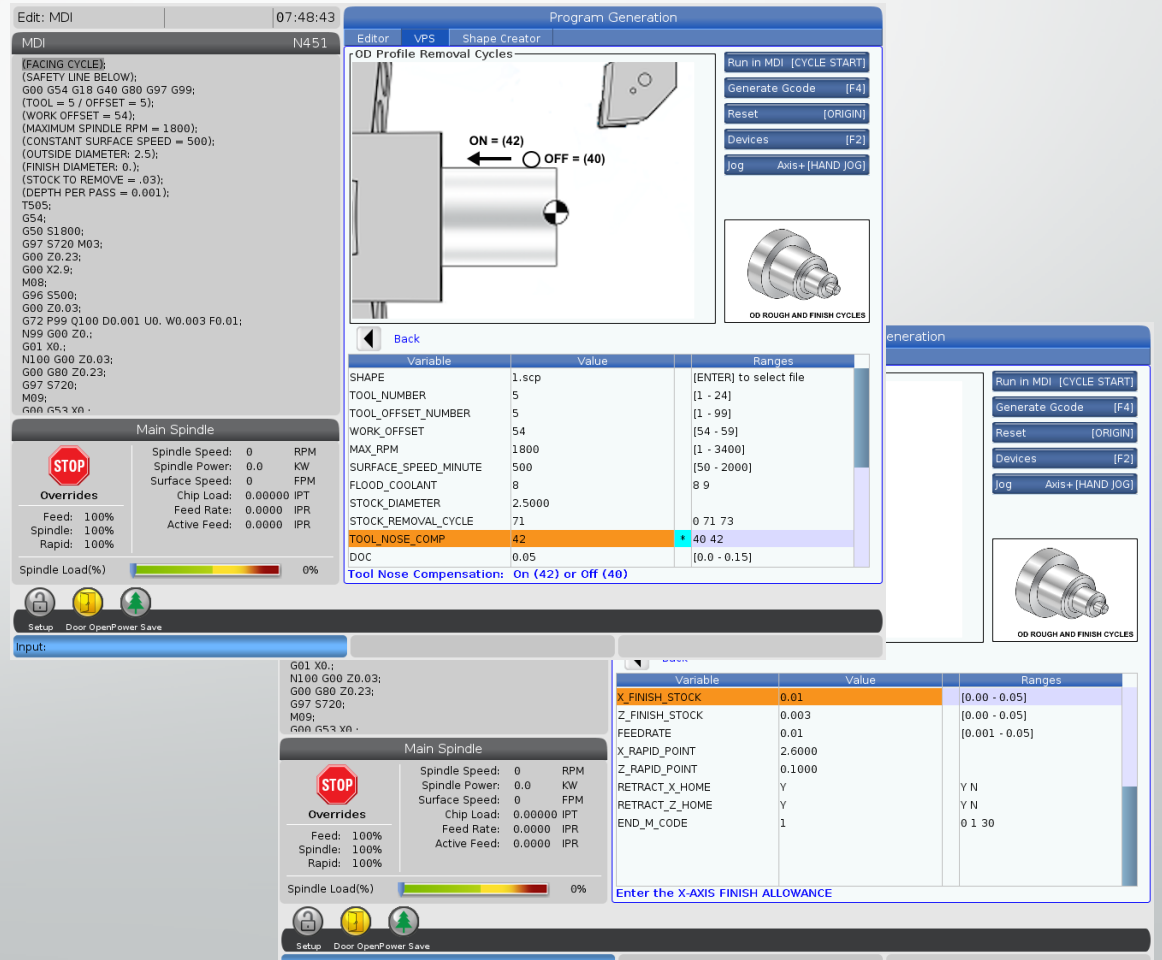
OD Profile Removal Cycle

- The first input required is the shape. Use the “enter” key to select the saved shapes.
  - In this case the saved # 1 shape is the only saved shape. Once highlighted, use “enter” again to select it.



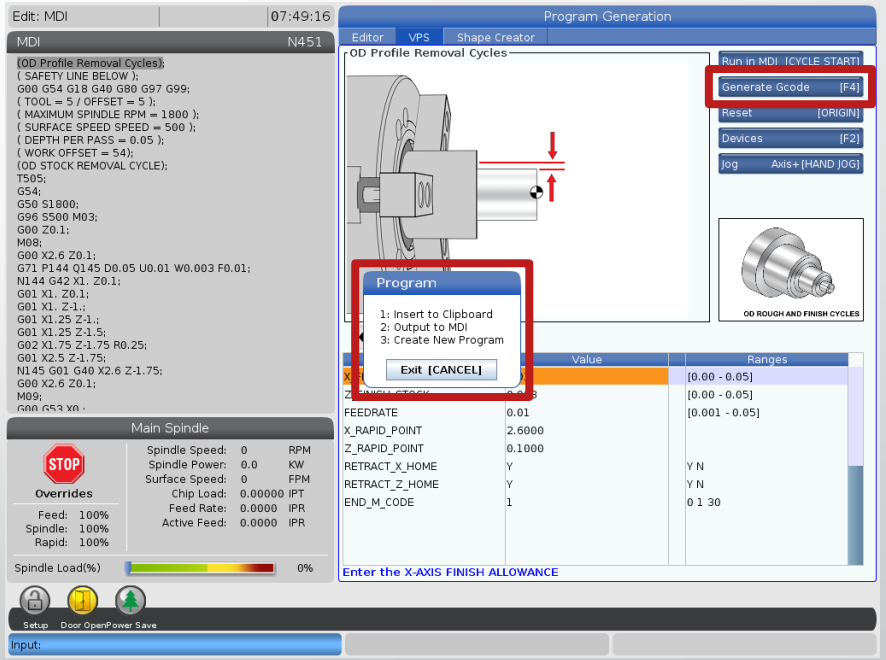
OD Profile Removal Cycle

- There are two pages of variables to fill out for the cycle. Use the up and down arrow keys to scroll through the variables.



OD Profile Removal Cycle

- After the variables are filled in, it should look something like this.
- Select F4 , generate code to bring up the output menu.
- Select “2”, which will output the code to MDI.



OD Profile Removal Cycle

- The program is then output to MDI.

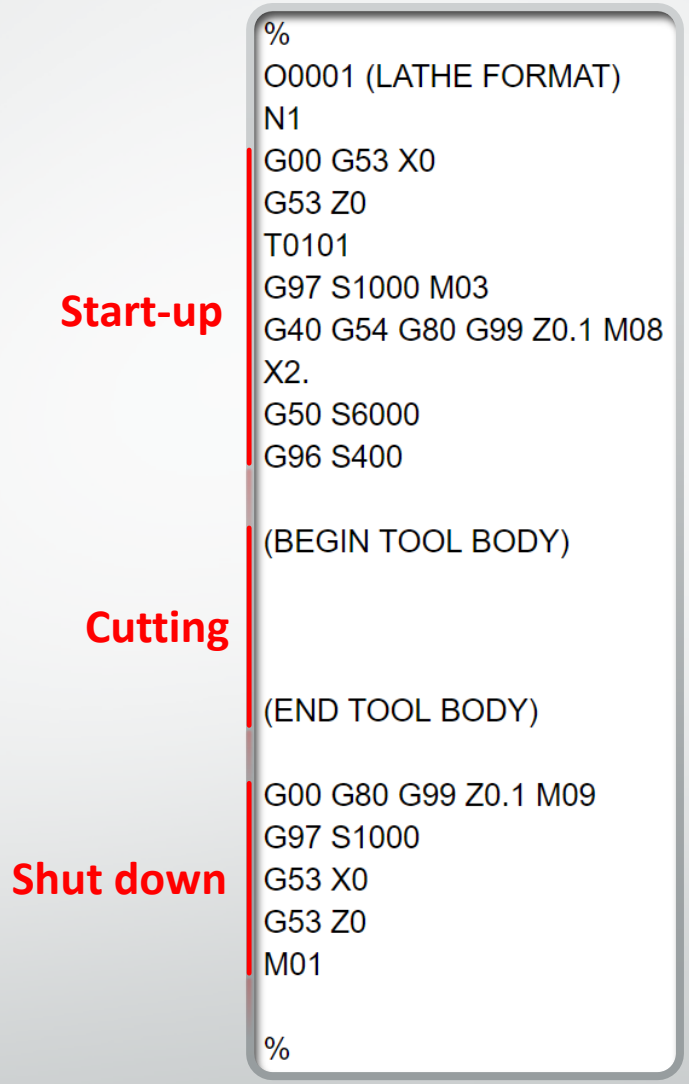
Program Structure

- When starting a new program the material, stock size, PRZ, and tool list should be included in the beginning of the program.
- This verifies that the correct stock, tools, and PRZ have been set-up.
- Each operation should have a note of what is happening. This helps the person operating the machine.

```
%
O0001 (LATHE FORMAT)
N1
G00 G53 X0
G53 Z0
T0101
G97 S1000 M03
G40 G54 G80 G99 Z0.1 M08
X2.
G50 S6000
G96 S400

(BEGIN TOOL BODY)



(END TOOL BODY)

G00 G80 G99 Z0.1 M09
G97 S1000
G53 X0
G53 Z0
M01

%
```

Program Structure

- Programs are broken up into 3 basic sections:
  - Start-up
    - In the example program to the right, line 4-11 will be the same sequence for each tool path we program. The variables such as RPM, feed rate and positions are the only thing that change.
  - Cutting
    - In the example program, this would be lines 12-13. This will vary in length and complexity depending upon the operation.
  - Shut Down
    - In the example program, this would be lines 14-18. These lines will have the same format for each tool that is programmed.



Tool Path Startup

- The first 8 lines of each tool path contain the start up of the tool. This should be done for each toolpath.
- The format is as follows:
  - Set to rapid, return X Home
  - Return Z home
  - Tool change/offset
  - Turn RPM on to s1,000 for approach
  - Safe start-up line, approach part in Z
  - Approach part in X
    - X and Z should be on separate lines to avoid the turret colliding with the tailstock.
  - Limit max spindle speed
  - Turn spindle on to Constant Cutting Speed (CCS).

```
%
O0001 (LATHE FORMAT)
N1
G00 G53 X0
G53 Z0
T0101
G97 S1000 M03
G40 G54 G80 G99 Z0.1 M08
X2.
G50 S6000
G96 S400

(BEGIN TOOL BODY)



(END TOOL BODY)

G00 G80 G99 Z0.1 M09
G97 S1000
G53 X0
G53 Z0
M01

%
```

Tool Path Startup

- Safe Start-up
  - This line is used to prepare the machine for cutting:
    - G40 – Cancel tool nose radius comp.
    - G54 – Active work offset
    - G80 – Cancel canned cycle
    - G99 – Sets feed to IPR
    - Z0.1 – Approach part in Z
    - M08 – Turn coolant on


```
%
O0001 (LATHE FORMAT)
N1
G00 G53 X0
G53 Z0
T0101
G97 S1000 M03
G40 G54 G80 G99 Z0.1 M08
X2.
G50 S6000
G96 S400

(BEGIN TOOL BODY)



(END TOOL BODY)

G00 G80 G99 Z0.1 M09
G97 S1000
G53 X0
G53 Z0
M01

%
```

Tool Path Startup

- T606
  - This calls up tool 1 and offset 1.
  - An M6 is not required for the tool to change.
- G97 S1000 M03
  - G97 turns the spindle at a direct spindle speed of 1000 RPM.
- G50 S6000
  - G50 sets the max spindle speed because G96 CCS mode will be used.
  - S6000 is the limiting spindle speed.
    - G50 can only be overridden by another G50 spindle command.
- G96 S400
  - This sets the CCS to 400.
    - CCS allows the spindle RPM to change based on the diameter of the part. This method uses the surface speed to calculate the proper RPM of the part based off the tools X diameter position when turning.

```
%
O0001 (LATHE FORMAT)
N1
G00 G53 X0
G53 Z0
T0101
G97 S1000 M03
G40 G54 G80 G99 Z0.1 M08
X2.
G50 S6000
G96 S400

(BEGIN TOOL BODY)



(END TOOL BODY)

G00 G80 G99 Z0.1 M09
G97 S1000
G53 X0
G53 Z0
M01

%
```

Shut Down

- After the cutting is complete, the tool path can be shut down. This is done in the final 5 lines of the program. The shut down procedure places the machine in rapid, turns off the spindle and coolant (if used), sends Z then X and Y home (separately to avoid collisions), and then uses an optional stop.
  - G00 G80 G99 Z.1 M09
    - This puts the machine back into rapid, cancels canned cycles, sets the machine to feed per revolution, retracts Z to .1", and turns coolant off.
  - G97 S1000
    - This turns the spindle on to a constant rpm of 1000 RPM.
  - G53 X0.
    - G53 sends the axis home
    - X0. indicates the axis to send home
    - Note that X is homed first to avoid possible collision
  - G53 Z0.
    - G53 sends the axis home
    - Z0. indicates the axis to send home
  - M01
    - Optional stop is used so the program will stop before the next tool path

```
%
O0001 (LATHE FORMAT)
N1
G00 G53 X0
G53 Z0
T0101
G97 S1000 M03
G40 G54 G80 G99 Z0.1 M08
X2.
G50 S6000
G96 S400

(BEGIN TOOL BODY)



(END TOOL BODY)

G00 G80 G99 Z0.1 M09
G97 S1000
G53 X0
G53 Z0
M01

%
```

Writing Safe Programs

- Regardless of the number of tools that are being used, each operation should be written as a separate program.
- This means that each tool has the proper safety start-up line, calls the tool change each time, and the cycle ends with the tool going to X & Z home.
- Writing programs this way ensures that any operation in the program can be re-run without issue.
  - I.E.- if a tool is used twice in a row in a program and a person does not write in a tool change for the second operation, a machinist cannot rerun the second operation without manually changing the tool.

G2 and G3 Calculations

Circular Interpolation

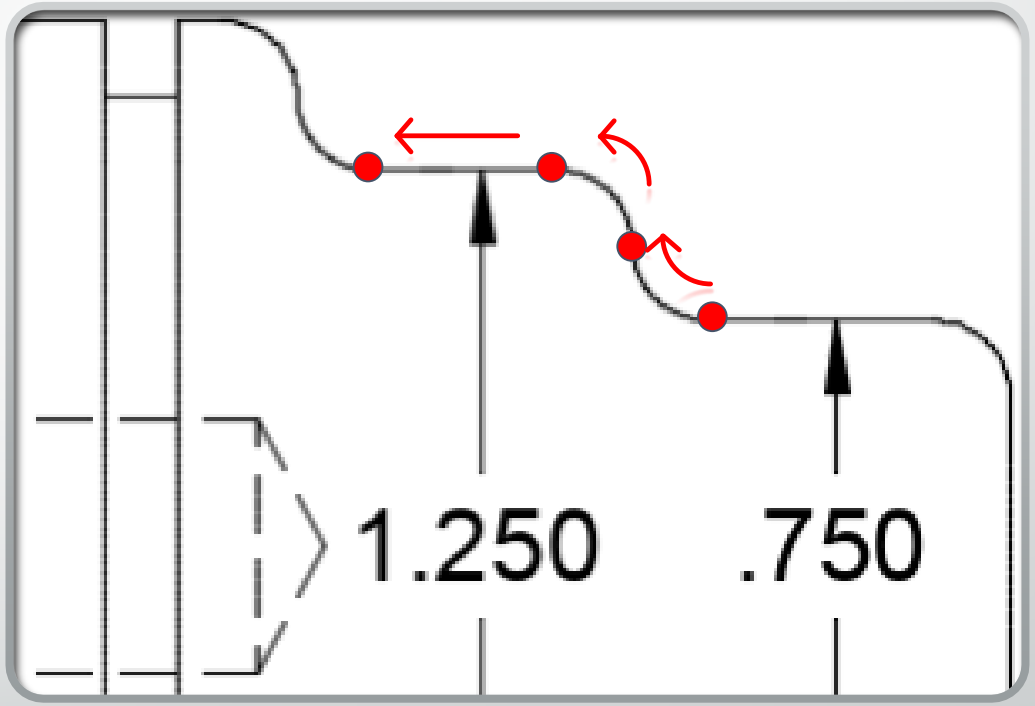
- In addition to G01 straight line movements there are:
- **G02** - Curve clockwise
- **G03** - Curve counterclockwise

- G01, G02, and G03 are modal commands. Meaning they stay active until they are overwritten.
- Circular toolpaths have an additional R value, this is the size of the radius the tool should take when traveling to the next location.

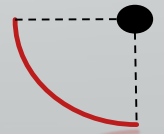
G2 and G3 Calculations

Defining the profile:

- Here is an example of two circular moves using the radius method for programming
- Starting at X.75 Z-.5:
  - G02 X1.0 Z-.625 R.125
  - G03 X1.25 Z-.75 R.125
  - G01 Z-1.057
- Notice that after a circular move there has to be a G01 in order to do a straight move.

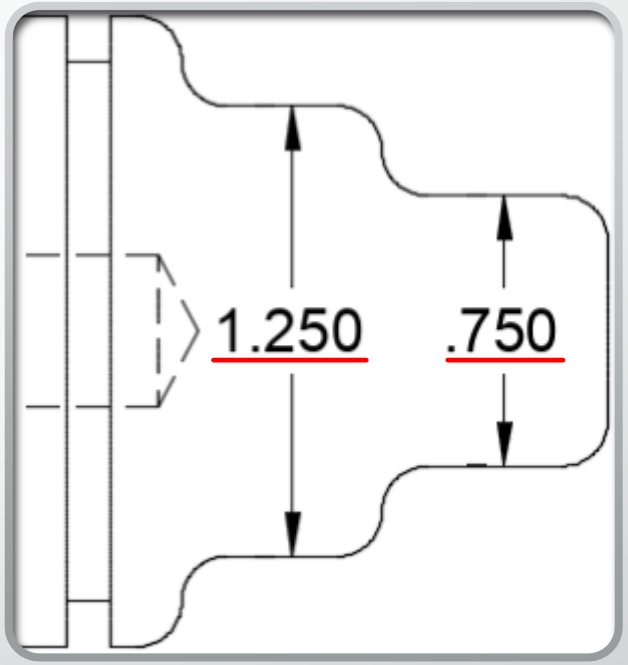


- With radius method programming, the center point of the arc bisects the start and end points of the arc.



G2 and G3 Calculations

- Notice that the circular moves are programmed off the diameter of the part, just like linear moves.
- This means that the X circular moves need to be doubled, this is because the move is defined as a radius but the machine reads diameter movements.



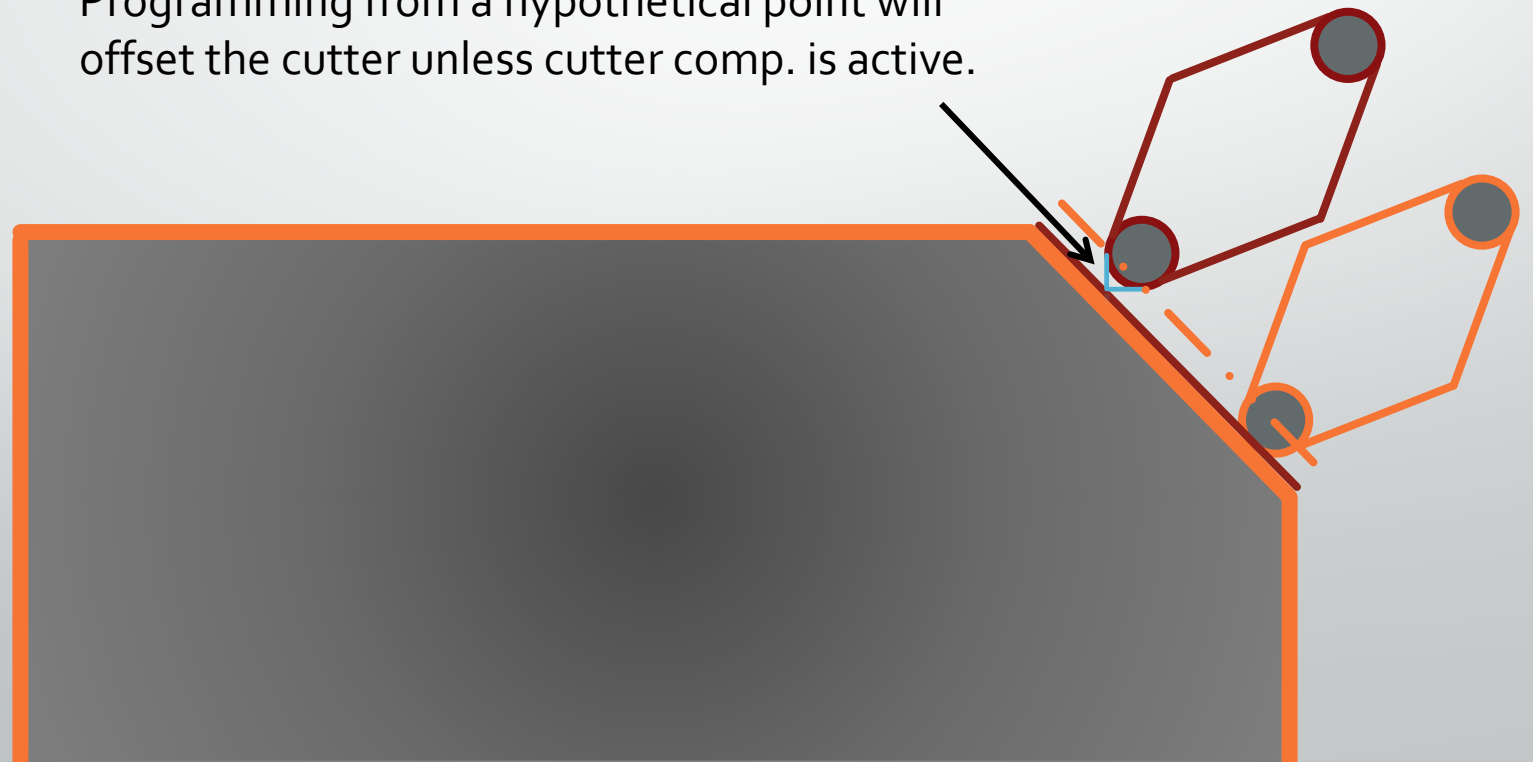

```
G71 P10 Q20 U.04 W.002 D.075 F.008
N10 X0.
G01 Z0.
X.5
G03 X.75 Z-.125 R.125
G01 Z-.5
G02 Z-.625 X1.0 R.125
G03 Z-.75 X1.25 R.125
G01 Z-1.057
```

Tool Nose Radius Compensation

- Tool nose radius compensation is used to offset the tool by the radius of the tool nose. This is needed because all tools have a rounded point, while the part is programmed for a perfectly square point.
- Tool nose radius compensation uses G41 and G42.
  - G41 – Tool nose comp. left (looking in direction of cut)
  - G42 Tool nose comp. right (looking in direction of cut)

Tool Nose Radius Compensation

Programming from a hypothetical point will offset the cutter unless cutter comp. is active.



Tool Nose Radius Compensation

```
%
O00019 (G71 TURNING + G42 TNRC)
(SAMPLE PROGRAM NOT FOR USE !!)

N1 (TOOL AT SAFE INDEX POSITION)
G00 G53 X0.
G53 Z0.
T101
G97 S1000 M03
G00 G40 G54 G80 G99 Z0.1 M08
X3.35
(CONSTANT SURFACE SPEED)
G50 S2500
G96 S350

(BEGIN TOOL BODY)

G71 P100 Q200 D0.1 F0.016 U0.01 W0.005
N100 G01 G42 X1.5 F0.004
Z0.
X1.6
X2. Z-0.2
Z-3.8
X2.4 Z-4.
X2.6
G03 X3. Z-4.2 R0.2
G01 Z-5.
N200 X3.25
(RGH CYCLE END)

(END TOOL BODY)
G00 G80 G99 Z0.1 M09
G97 S1000
(TOOL AT SAFE INDEX POSITION)
G53 X0.
G53 Z0.
M30
%
```

Canned Cycles

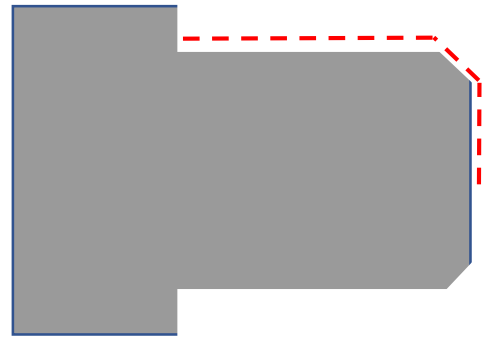
- Canned cycles are like toolpath templates, they use set values to complete toolpaths with minimal programming.
- There are multiple milling canned cycles for operations such as drilling and tapping.
- Canned cycles remain active until they are cancelled.

G71 Rough Turning

- G71 is used to rough material from the part profile while leaving a defined amount for a finish pass.
- A G71 Cycle looks like this:
  - G71 P10 Q20 U.020 W.003 D.075 F.008

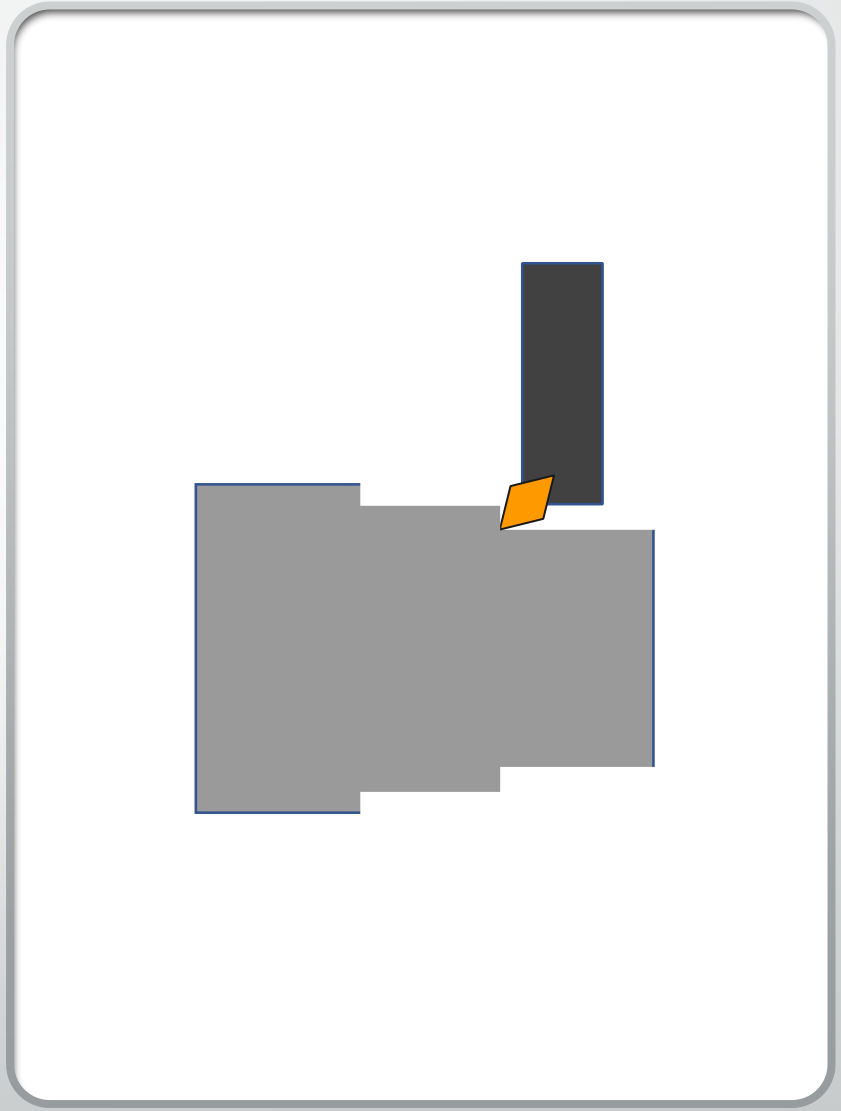
G71 Canned Roughing Cycle

- After the tool is in position, the cycle can be called up. It looks like this:
- G71 P10 Q20 U.020 W.003 D.075 F.008
  - **U.020** - The incremental amount of material left on the X diameter. This is left for the finishing pass. This is defined in incremental because there can be multiple diameters to leave stock on.
  - **W.003** - The incremental amount of material left on the Z face. This is left for the finishing pass. This is defined in incremental because there can be multiple shoulders to leave stock on.



G71 Canned Roughing Cycle

- After facing is complete, the OD of the part is rough turned followed by a finish toolpath. Lathes use canned cycles for turning operations.
- Canned cycles are an efficient way to remove material with minimal programming effort.



```
%
O00017 (G71 TURNING)
(SAMPLE PROGRAM NOT FOR USE !!)

N1 (START TOOL AT SAFE INDEX POSITION)
G00 G53 X0.
G53 Z0.
T101
G97 S1000 M03
G00 G40 G54 G80 G99 Z0.1 M08
X3.35
(CONSTANT SURFACE SPEED)
G50 S2500
G96 S350

(BEGIN TOOL BODY)
(RGH. TURN CYCLE)
N7 G71 P100 Q200 D0.1 F0.016 U0.01 W0.005
N100 G01 X1.5 F0.004
Z0.
X1.6
X2. Z-0.2
Z-3.8
X2.4 Z-4.
X2.6
G03 X3. Z-4.2 R0.2
G01 Z-5.
N200 X3.25 G40
(RGH. & FIN. CYCLE END)

(END TOOL BODY)

G00 G80 G99 Z0.1 M09
G97 S1000
(TOOL AT SAFE INDEX POSITION)
G53 X0.
G53 Z0.
M30
%
```

G71 Canned Roughing Cycle

- For a canned roughing cycle to work, the profile of the part must be defined. When activated, the cycle reads the profile then removes material based on the defined variables within the cycle.

- The first thing needed for the roughing cycle to work is the starting point. As a rule of thumb, the tool point should be within .1” of the face and .050” of the largest diameter.
  - On this part, the starting point would be X3.35, Z.1.

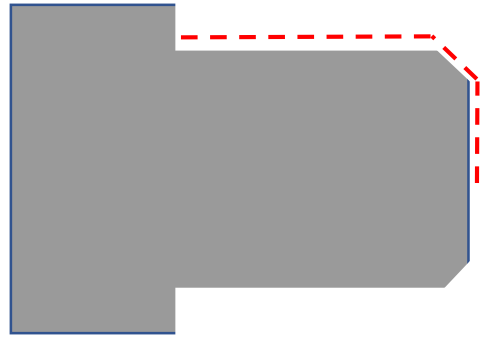
```
%
O00017 (G71 TURNING)
(SAMPLE PROGRAM NOT FOR USE !!)

N1 (START TOOL AT SAFE INDEX POSITION)
G00 G53 X0.
G53 Z0.
T101
G97 S1000 M03
G00 G40 G54 G80 G99 Z0.1 M08
X3.35
(CONSTANT SURFACE SPEED)
G50 S2500
G96 S350

(BEGIN TOOL BODY)
(RGH. TURN CYCLE)
N7 G71 P100 Q200 D0.1 F0.016 U0.01 W0.005
N100 G01 X1.5 F0.004
Z0.
X1.6
X2. Z-0.2
Z-3.8
X2.4 Z-4.
X2.6
G03 X3. Z-4.2 R0.2
G01 Z-5.
N200 X3.25 G40
(RGH. & FIN. CYCLE END)

(END TOOL BODY)

G00 G80 G99 Z0.1 M09
G97 S1000
(TOOL AT SAFE INDEX POSITION)
G53 X0.
G53 Z0.
M30
%
```

G71 Canned Roughing Cycle

- After the tool is in position, the cycle is called up. It looks like this:
- G71 P100 Q200 D.1 F.016 U.01 W.005
  - **P100** - Starting block number of the profile. This can be any number you choose but must match the N number in the program.
  - **Q200** - Ending block number of the profile. This can be any number you choose but must match the N number in the program.

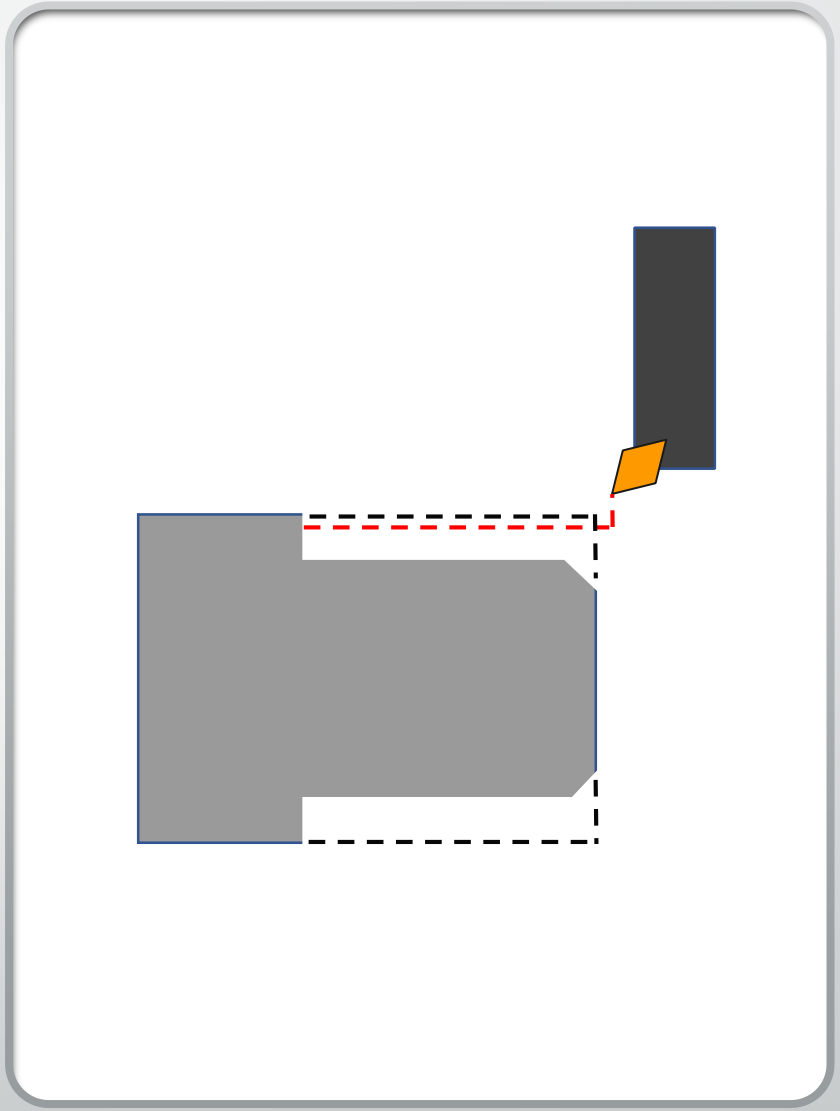
G71 Canned Roughing Cycle

- After the tool is in position, the cycle can be called up. It looks like this:
- G71 P100 Q200 D.1 F.016 U.02 W.003
  - **U.02** - The incremental amount of material left on the X diameter. This is left for the finishing pass. This is defined in incremental because there can be multiple diameters to leave stock on.
  - **W.03** - The incremental amount of material left on the Z face. This is left for the finishing pass. This is defined in incremental because there can be multiple shoulders to leave stock on.



G71 Canned Roughing Cycle

- After the tool is in position, the cycle can be called up. It looks like this:
- G71 P100 Q200 D.1 F.016 U.02 W.003
  - **D.1** - The depth of cut taken per pass. This can be adjusted based on the material, cutting too, and part features.
  - **F.016** - Feed in IPR to be used during the roughing cycle. This can be changed based off the material or cutting tool specifications.



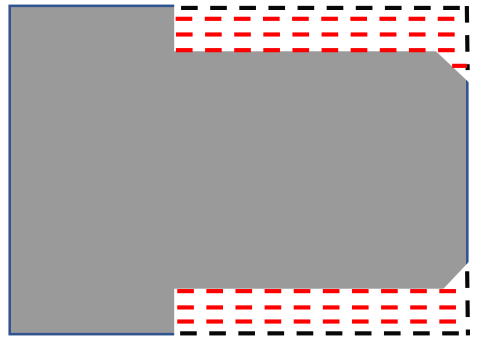
```
%
O00017 (G71 TURNING)
(SAMPLE PROGRAM NOT FOR USE !!)

N1 (START TOOL AT SAFE INDEX POSITION)
G00 G53 X0.
G53 Z0.
T101
G97 S1000 M03
G00 G40 G54 G80 G99 Z0.1 M08
X3.35
(CONSTANT SURFACE SPEED)
G50 S2500
G96 S350

(BEGIN TOOL BODY)
(RGH. TURN CYCLE)
N7 G71 P100 Q200 D0.1 F0.016 U0.01 W0.005
N100 G01 X1.5 F0.004
Z0.
X1.6
X2. Z-0.2
Z-3.8
X2.4 Z-4.
X2.6
G03 X3. Z-4.2 R0.2
G01 Z-5.
N200 X3.25 G40
(RGH. & FIN. CYCLE END)

(END TOOL BODY)

G00 G80 G99 Z0.1 M09
G97 S1000
(TOOL AT SAFE INDEX POSITION)
G53 X0.
G53 Z0.
M30
%
```

G71 Canned Roughing Cycle

- Defining the profile:
  - The profile needs to be defined after the cycle is activated. This profile must happen between the starting and ending block numbers as defined by the roughing cycle.
  - The profile needs to start with an X negative move from the starting position and end with a final X move in the positive direction.

G71 Canned Roughing Cycle

- G71 P100 Q200 D.1 F.016 U.01 W.005
  - The cycle will read through the profile block numbers and start roughing the part. The first depth of cut is based from the tool starting position.
  - If the tool starts too low the cut will be too deep, likewise, if the tool is too far above the part, it will cut air.



```
%
O00021 (G72 FACING)
(SAMPLE PROGRAM NOT FOR USE !!)

N1 (TOOL AT SAFE INDEX POSITION)
G00 G53 X0.
G53 Z0.
T101
G97 S1000 M03
G00 G40 G54 G80 G99 Z0.1 M08
X6.1
(CONSTANT SURFACE SPEED)
G50 S2500
G96 S350

(BEGIN TOOL BODY)

(RGH. FACE CYCLE)
G72 P100 Q200 D0.1 F0.008 U0.01 W0.005
N100 G00 Z-1.
G01 X2.4 F0.0035
G03 X2. Z-0.8 R0.2
G01 Z-0.7
G02 X1.6 Z-0.5 R0.2
G01 X1.
Z-0.2
X0.6 Z0.
X-0.032
N200 Z0.1
(RGH CYCLE END)

(END TOOL BODY)

G00 G80 G99 Z0.1 M09
G97 S1000
(TOOL AT SAFE INDEX POSITION)
G53 X0.
G53 Z0.
M30

%
```

G72 Rough Facing Cycle

- G72 is used to rough material from the part profile while leaving a defined amount for a finish pass.
- A G72 Cycle looks like this:
  - G72 P100 Q200 D.1 F.008 U.01 W.005

G72 Rough Facing Cycle

- After the tool is in position, the cycle can be called up. It looks like this:
- G72 P100 Q200 D.1 F.008 U.01 W.005
  - **U.01** - The incremental amount of material left on the X diameter. This is left for the finishing pass. This is defined in incremental because there can be multiple diameters to leave stock on.
  - **W.005** - The incremental amount of material left on the Z face. This is left for the finishing pass. This is defined in incremental because there can be multiple shoulders to leave stock on.

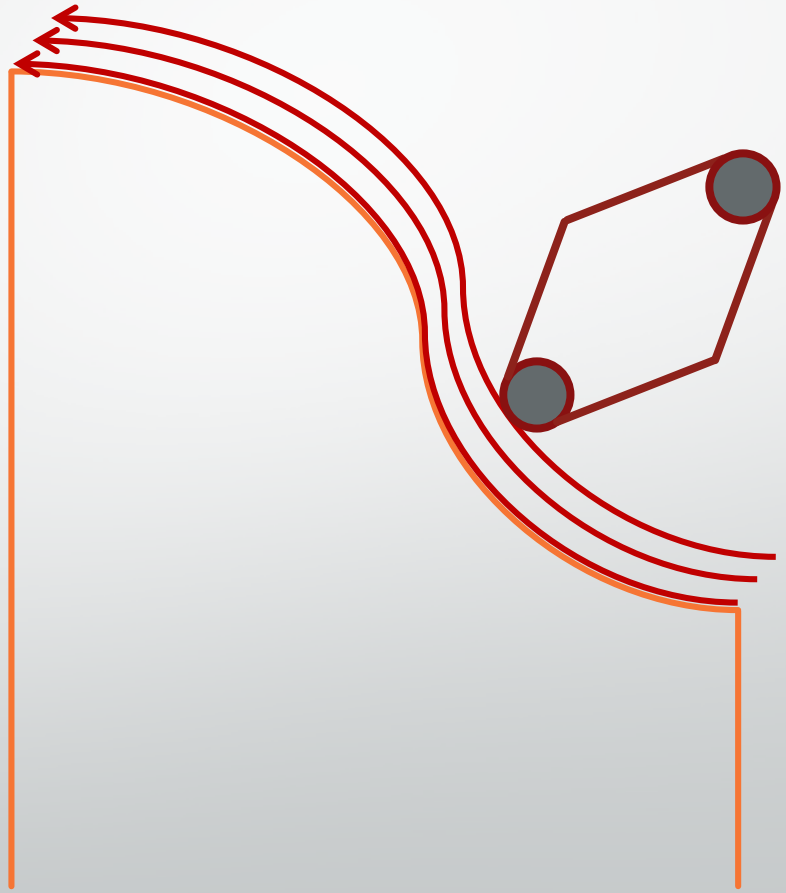
G73 Irregular Path Stock Removal Cycle

- G73 Roughs material from a profile with irregular contours. The main difference from G71 Canned roughing is that instead of roughing all stock material, it makes a set number of passes along the prescribed profile. This eliminates wasted passes for situations like turning castings where the profile is rough in.
- A G73 Cycle looks like this:
  - G73 D.1 I.06 K.06 P100 Q200

G73 Irregular Path Stock Removal Cycle

- G73 D.1 I.06 K.06 P100 Q200
  - D.1 – Number of passes
  - I.06 – X-axis distance and direction from first cut to last, radius
  - K.06 – Z-axis distance and direction from first cut to last
  - P100 – Starting block number
  - Q200 – Ending block number

G73 Irregular Path Stock Removal Cycle



G73 Irregular Path Stock Removal Cycle

```
%
O00023 (G73 PATTERN REPEAT)
(SAMPLE PROGRAM NOT FOR USE !!)

N1 (START TOOL AT SAFE INDEX POSITION)
G00 G53 X0.
G53 Z0.
T101
G97 S1000 M03
G00 G40 G54 G80 G99 Z0.1 M08
X6.1
(CONSTANT SURFACE SPEED)
G50 S2500
G96 S350

(BEGIN TOOL BODY)

(RGH.PATTERN REPEAT CYCLE)
G73 P100 Q200 I0.4 K0.4 D4 U0.01 W0.005 F0.008
N100 G00 Z-1.
G01 X2.4 F0.0035
G03 X2. Z-0.8 R0.2
G01 Z-0.7
G02 X1.6 Z-0.5 R0.2
G01 X1.
Z-0.2
X0.6 Z0.
X-0.032
N200 Z0.1
(RGH CYCLE END)

(END TOOL BODY)

G00 G80 G99 Z0.1 M09
G97 S1000
(TOOL AT SAFE INDEX POSITION)
G53 X0.
G53 Z0.
M30
%
```

G70- Canned Finish Cycle

- Canned finish cycles work with GG71,G72 and G73. After the part is roughed, a finish pass must be made.

- Finishing cycles are similar to roughing cycles in that they follow the same profile, just with a different purpose.
  - Finishing cycles usually include slower feeds, as well as tools with smaller included angles and nose radii. This improves the surface finish

```
%
O00019 (G71 TURNING + G42 TNRC)
(SAMPLE PROGRAM NOT FOR USE !!)

N1 (TOOL AT SAFE INDEX POSITION)
G00 G53 X0.
G53 Z0.
T101
G97 S1000 M03
G00 G40 G54 G80 G99 Z0.1 M08
X3.35
(CONSTANT SURFACE SPEED)
G50 S2500
G96 S350

(BEGIN TOOL BODY)

G71 P100 Q200 D0.1 F0.016 U0.01 W0.005
N100 G01 G42 X1.5 F0.004
Z0.
X1.6
X2. Z-0.2
Z-3.8
X2.4 Z-4.
X2.6
G03 X3. Z-4.2 R0.2
G01 Z-5.
N200 X3.25
(RGH CYCLE END)

(END TOOL BODY)
G00 G80 G99 Z0.1 M09
G97 S1000
(TOOL AT SAFE INDEX POSITION)
G53 X0.
G53 Z0.
M01

N2 (TOOL AT SAFE INDEX POSITION)
G00 G53 X0.
G53 Z0.
T303
G97 S1000 M03
G00 G40 G54 G80 G99 Z0.1 M08
X3.35
(CONSTANT SURFACE SPEED)
G50 S2500
G96 S350

(BEGIN TOOL BODY)

G70 P100 Q200 F.008

(END TOOL BODY)
G00 G80 G99 Z0.1 M09
G97 S1000
(TOOL AT SAFE INDEX POSITION)
G53 X0.
G53 Z0.
M30
%
```

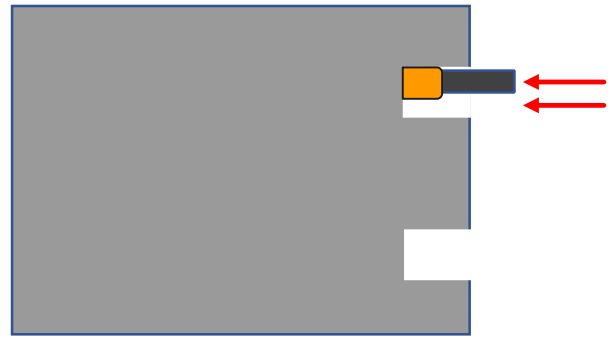
G70- Canned Finish Cycle

- Finish passes utilize a G70 canned finishing cycle, which looks like this:
  - G70 P10 Q20 F.005
    - This is the same profile as the roughing, but is being utilized for finishing. Once activated, the cycle will read through the program until the block numbers are found.
    - Since this tool is taking a finish pass, U and W are omitted from the canned cycle. This is because the part is being cut to size.
    - Almost all of the material has been removed from the profile. In this case, D or depth of cut is also not needed in the profile.
    - The feedrate can be defined based from the tool and material.

```
%
O00019 (G71 TURNING + G42 TNRC)
(SAMPLE PROGRAM NOT FOR USE !!)

N1 (TOOL AT SAFE INDEX POSITION)
G00 G53 X0.
G53 Z0.
T101
G97 S1000 M03
G00 G40 G54 G80 G99 Z0.1 M08
X3.35
(CONSTANT SURFACE SPEED)
G50 S2500
G96 S350

(BEGIN TOOL BODY)

G71 P100 Q200 D0.1 F0.016 U0.01 W0.005
N100 G01 G42 X1.5 F0.004
Z0.
X1.6
X2. Z-0.2
Z-3.8
X2.4 Z-4.
X2.6
G03 X3. Z-4.2 R0.2
G01 Z-5.
N200 X3.25
(RGH CYCLE END)

(END TOOL BODY)
G00 G80 G99 Z0.1 M09
G97 S1000
(TOOL AT SAFE INDEX POSITION)
G53 X0.
G53 Z0.
M01

N2 (TOOL AT SAFE INDEX POSITION)
G00 G53 X0.
G53 Z0.
T303
G97 S1000 M03
G00 G40 G54 G80 G99 Z0.1 M08
X3.35
(CONSTANT SURFACE SPEED)
G50 S2500
G96 S350

(BEGIN TOOL BODY)

G70 P100 Q200 F.008

(END TOOL BODY)
G00 G80 G99 Z0.1 M09
G97 S1000
(TOOL AT SAFE INDEX POSITION)
G53 X0.
G53 Z0.
M30
%
```

G74 End Face Grooving Cycle

- G74 Is used to groove the face of the part. Therefore, this toolpath consists of numerous plunges along the Z axis.
- A G74 Cycle looks like this:
  - G74 X1.375 Z-.5 I.1 K.125 F2.



G74 End Face Grooving Cycle

- G74 X1.375 Z-.5 I.1 K.125 F2.
  - X1.375 – End X position
  - Z-.5 – End Z position
  - I.1 – X-axis size of increment between peck cycles
  - K.125 – Z-axis size of increment between pecks in a cycle
  - F2. – Feed

G74 End Face Grooving Cycle

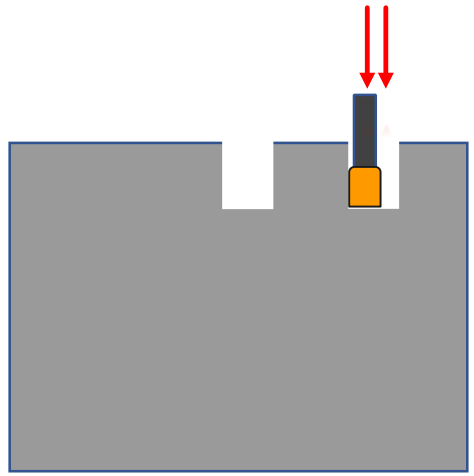
G74 X1.375 Z-.5 I.1 K.125 F2.

- X1.375 – End X position
- Z-.5 – End Z position
- I.1 – X-axis size of increment between peck cycles
- K.125 – Z-axis size of increment between pecks in a cycle
- F2. – Feed

```
N2
(TOOL AT SAFE INDEX POSITION)
G00 G53 X0.
G53 Z0.
T202
G97 S1000 M03
G00 G40 G54 G80 G99 Z0.1 M08
X1.3

(BEGIN TOOL BODY)

(RGH. GROOVING CYCLE)
G74 X0.4 Z-0.1 I0.1 K0.025 F0.004
X1.3 Z0.1
G01 Z-0.1 F0.003
X0.4
Z0.1
(END TOOL BODY)

G00 G80 G99 Z0.1 M09
G97 S1000
(TOOL AT SAFE INDEX POSITION)
G53 X0.
G53 Z0.
M30
%
```

G75 OD/ID Face Grooving Cycle

- G75 Is used to groove the diameters of the part. Therefore, this toolpath consists of numerous plunges along the X axis.
- A G75 Cycle looks like this:
  - G75 X1.9 Z-.625 I.125 K.1 F2.



G75 OD/ID Face Grooving Cycle

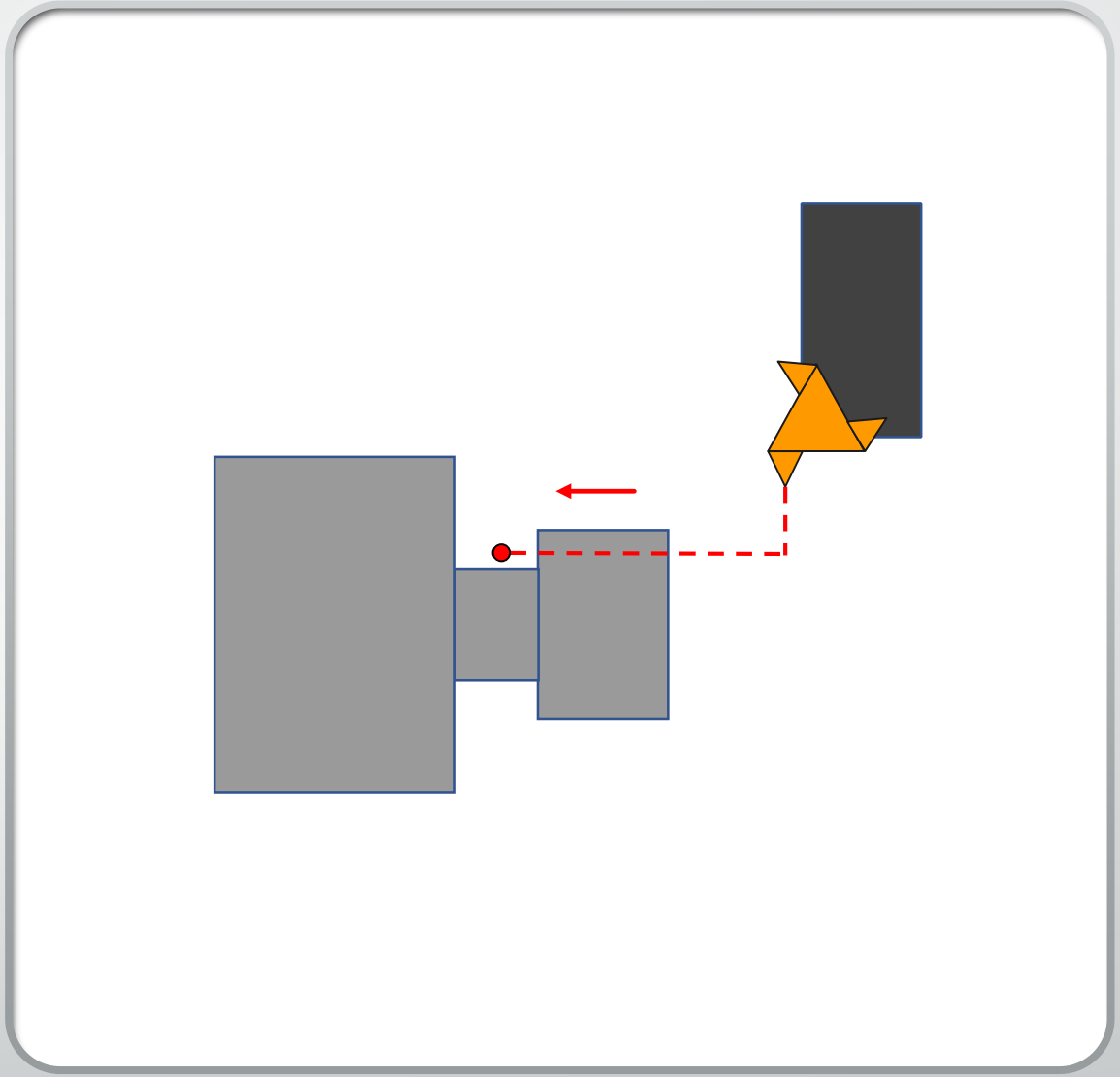
G75 X1.9 Z-.625 I.125 K.1 F2.

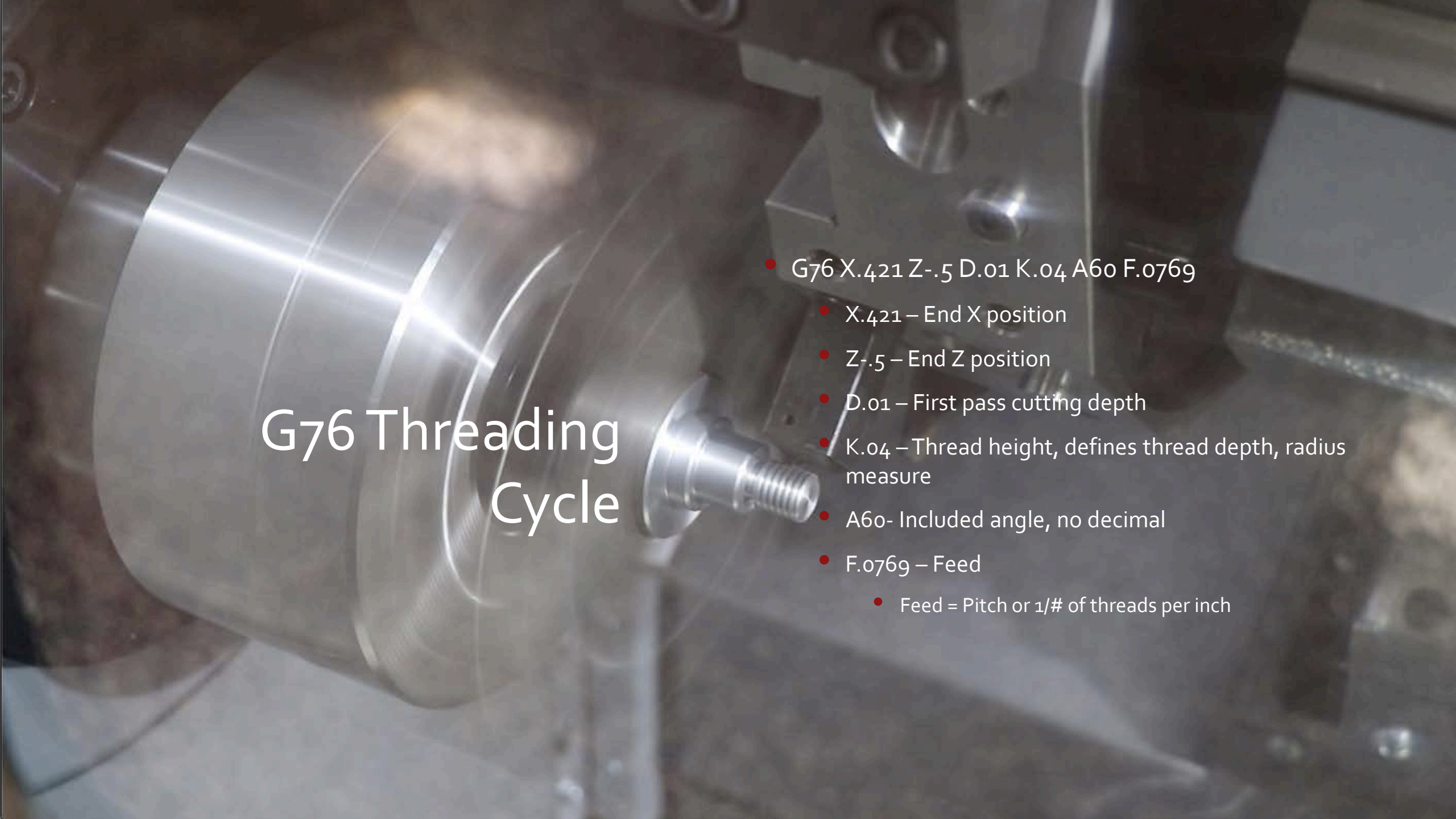
- X1.9 – End X position
- Z-.625 – End Z position
- I.125 – X-axis size of increment between peck cycles in a cycle (radius measure)
- K.1 – Z-axis size of increment between peck cycles
- F2. – Feed

```
N2
(TOOL AT SAFE INDEX POSITION)
G00 G53 X0.
G53 Z0.
T202
G97 S1000 M03
G00 G40 G54 G80 G99 Z0.1 M08
X2.1

(BEGIN TOOL BODY)

(POSITION GROOVE)
Z-1.105

(RGH. GROOVING CYCLE)
G75 X1.82 Z-1.995 I0.025 K0.1 F0.004

(FIN. GROOVE PASS)
Z-1.1
G01 X1.8 F0.003
Z-2.
X2.1
(END TOOL BODY)

G00 G80 G99 Z0.1 M09
G97 S1000
(TOOL AT SAFE INDEX POSITION)
G53 X0.
G53 Z0.
M30
%
```

G76 Threading Cycle

- G76 Is used to thread with multiple passes.
- A G76 Cycle looks like this:
  - G76 X.421 Z-.5 D.01 K.04 F.0769





G76 Threading Cycle

- G76 X.421 Z-.5 D.01 K.04 A60 F.0769
  - X.421 – End X position
  - Z-.5 – End Z position
  - D.01 – First pass cutting depth
  - K.04 – Thread height, defines thread depth, radius measure
  - A60- Included angle, no decimal
  - F.0769 – Feed
    - Feed = Pitch or 1/# of threads per inch

G76 Threading Cycle

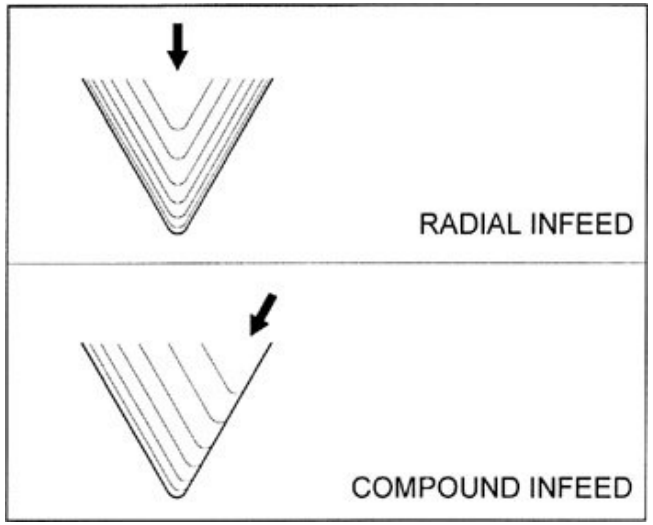
- G76 Is used to thread with multiple passes.
- A G76 Cycle looks like this:
  - G76 X.421 Z-.5 D.01 K.04 A60 F.0769

```
G00 G80 G99 Z0.1 M09
G97 S1000
(TOOL AT SAFE INDEX POSITION)
G53 X0.
G53 Z0.
M01

N2 (TOOL AT SAFE INDEX POSITION)
G00 G53 X0.
G53 Z0.
T202
G97 S1000 M03
G00 G40 G54 G80 G99 Z0.2 M08
X1.6

(BEGIN TOOL BODY)
M24
G76 X1.4386 Z-1.2 D0.0094 K0.0307 F0.05 A60 P1
(END TOOL BODY)

G00 G80 G99 Z0.2 M09
G97 S1000
(TOOL AT SAFE INDEX POSITION)
G53 X0.
G53 Z0.
M30
%
```

G76 Threading Cycle

- P can also be added to the threading cycle. P(1-4) are options and can change the tools in feed based off the numerical value.
  - *P1*:Single edge cutting, cutting amount constant
  - *P2*:Double edge cutting, cutting amount constant
  - *P3*: Single edge cutting, cutting depth constant
  - *P4*: Double edge cutting, cutting depth constant

```
G00 G80 G99 Z0.1 M09
G97 S1000
(TOOL AT SAFE INDEX POSITION)
G53 X0.
G53 Z0.
M01

N2 (TOOL AT SAFE INDEX POSITION)
G00 G53 X0.
G53 Z0.
T202
G97 S1000 M03
G00 G40 G54 G80 G99 Z0.2 M08
X1.6

(BEGIN TOOL BODY)
M24
G76 X1.4386 Z-1.2 D0.0094 K0.0307 F0.05 A60 P1
(END TOOL BODY)

G00 G80 G99 Z0.2 M09
G97 S1000
(TOOL AT SAFE INDEX POSITION)
G53 X0.
G53 Z0.
M30
%
```

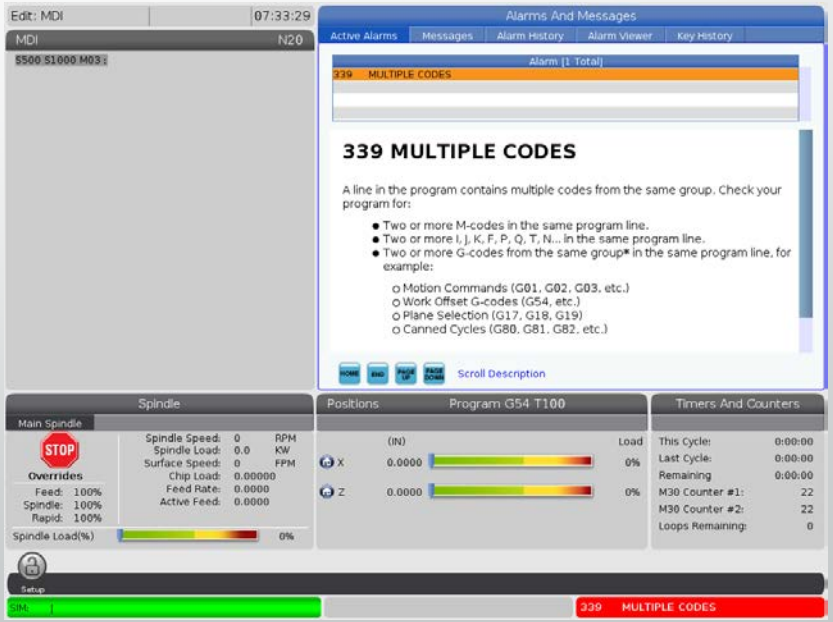


Program Trouble Shooting

- New programs often have issues, especially if they are hand programmed.
- New programs should be verified before cutting the part.
- It is a good habit to use single block and slow the rapid to 5% when running a new program.

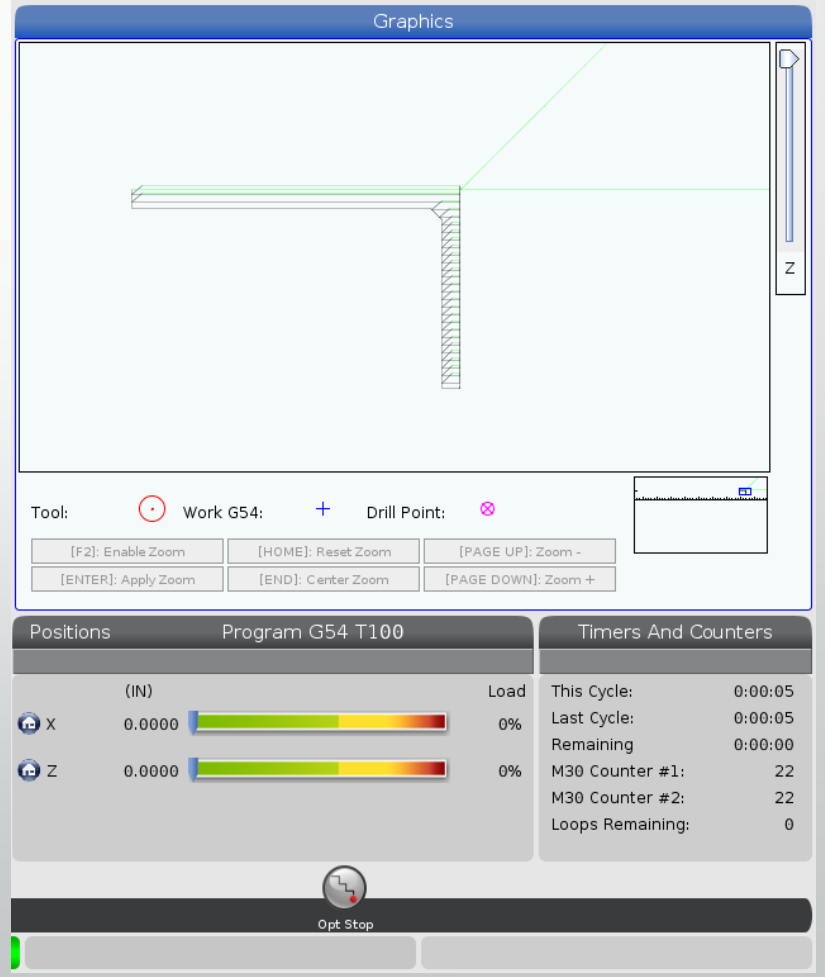
Program Trouble Shooting

- If an issue with the program does come, the machine will give an alarm.
- It is important not to hit reset right away.
  - Select the alarm button from the display keys and read the active alarm. This will give information about the issue.
  - Then go back to memory and read the line the alarm happened on. If it looks ok, read the following lines.
    - The machine has the ability to look ahead in the program. This means that the line the machine alarmed may not be the line of code with the issue.



Program Trouble Shooting

- The graphics simulation can also be used to get a 2D visual of the programmed toolpath without running the machine in memory.



Program Trouble Shooting

- When there is an alarm in a canned cycle and the issue is not apparent, a forward slash can be used. This forward slash combined with the block delete key will ignore the canned cycle line and allow the user to run through the code.
- When using this method it is also good to have single block activated on the control.