

# Welcome to Trident Machine Tools Mill Program Troubleshooting



# Mill Program Troubleshooting

 This one day course is designed to provide the user with a basic understanding of basic mill programming and trouble shooting common program issues.

#### Schedule

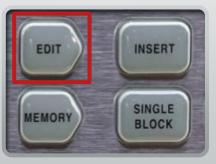
- Introductions
- VPS overview
  - VPS demo
- Break
- Program structure
- Cutter Compensation
- Canned cycles
- Lunch
- Canned cycles
- Break
- Program trouble shooting
- Questions

# 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 mill this might entail facing, drilling, contouring, among other options.

Editor VPS		F	Program Generation		
Load      Recently Used      VPS/O-Ring Groove Milling      VPS/Face Milling      VPS/Engraving      VPS/Circle Pocket Milling      VPS/Circle Contour Milling      VPS/Circle Contour Milling      VPS/Circle Contour Milling      Search (TEXT) [F1], or [F1] to clear.      Current Directory:      File Name    Size      VPS    <0IR>    08/18/1      VPS    <0IR>    08/18/1	Editor <u>VPS</u>				
Recently Used      VPS/O-Ring Groove Milling      VPS/Face Milling      VPS/Engraving      VPS/Drilling      VPS/Circle Pocket Milling      VPS/Circle Contour Milling      VPS/Circle Contour Milling      VPS/Circle Contour Milling      Search (TEXT) [F1], or [F1] to clear.      Current Directory:      File Name    Size      Las      PROBING <dir>    08/18/1      VPS    <dir>    08/18/1</dir></dir>					To Switc
Recently Used      VPS/O-Ring Groove Milling      VPS/Face Milling      VPS/Engraving      VPS/Drilling      VPS/Circle Pocket Milling      VPS/Circle Contour Milling      VPS/Circle Contour Milling      VPS/Circle Contour Milling      Search (TEXT) [F1], or [F1] to clear.      Current Directory:      File Name    Size      Las      PROBING <dir>    08/18/1      VPS    <dir>    08/18/1</dir></dir>				i	Load
VPS/Face Milling VPS/Engraving VPS/Circle Pocket Milling VPS/Circle Contour Milling VPS/Circle Contour Milling Forward Search (TEXT) [F1], or [F1] to clear. Current Directory: File Name Size Las PROBING <dir> 08/18/1 VPS <dir> 08/18/1</dir></dir>	Recently Used				Louia
Current Directory:      File Name    Size    Las      PROBING <dir>    08/18/1      VPS    <dir>    08/18/1</dir></dir>	VPS/Face Milling VPS/Engraving VPS/Drilling VPS/Circle Pocket I	Milling			
File Name    Size    Las      PROBING <dir>    08/18/1      VPS    <dir>    08/18/1</dir></dir>					
File Name    Size    Last      PROBING <dir>    08/18/1      VPS    <dir>    08/18/1</dir></dir>					
Current Directory:      File Name    Size    Las      PROBING <dir>    08/18/1      VPS    <dir>    08/18/1</dir></dir>					
File Name      Size      Las        PROBING <dir>      08/18/1        VPS      <dir>      08/18/1</dir></dir>			Search (TEXT) [F1], or	[F1] to a	lear.
PROBING <dir>      08/18/1        VPS      <dir>      08/18/1</dir></dir>	Current Director	-			
VPS <pre><pre><pre><pre><pre>VPS</pre><pre><pre>08/18/1</pre></pre></pre></pre></pre></pre>	-	File Name			Las
					00/10/1
	l				

- 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.



peration: MEM		15:45:24	Program Generation		
MEM Me	emory/000000.nc	NØ	Editor VPS		
00000(HAAS VM-2 S DATE=DD-MM-YY -0, MCX FILE - E:VINTRO T UBPLATE WITH SLOT NC FILE - E:VINTRO T UBPLATE.NC): MATERIAL - ALLMINUJ T1   37 FACE MILL   H T3   27/64 DRILL   H T6   1/2 FLAT ENDMI .5. );	UBPLATE); 204-18 TIME=HH:MM - 16 0 TOOLMAKING\HAAS VM- EMCAM); TOOLMAKING\HAAS VM-2 M INCH - 2024); 41 ); H2 );	2   TOOL DIA.	Recently Used VPS/0-Ring Groove Milling VPS/Face Milling VPS/Drilling VPS/Circle Pocket Milling VPS/Circle Contour Milling	To Switch Boxes	[F4] [ENTER]
.5); T8 1/2 FLAT ENDMI 1100 G20; 1110 G0 G17 G40 G4 FACE); 1120 T1 M6;	LL   H8 ); 19 GB0 G90; 2.993 Y1.6822 A0. 5300 M 24; 22; 28 I-6303 J.2416; 1336;		PROBING <u< td=""><td>to clear.</td><td>&gt; &gt;</td></u<>	to clear.	> >
_	Spindle			08/18/17 19:13	>
Main Spindle STOP Overrides Feed: 100% Spindle: 100% Rapid: 100%	Spindle Speed: 0 Spindle Load: 0.1 Surface Speed: 0 Chip Load: 0.1 Feed Rate: 0.1 Active Feed: 0.1	FPM 00000 0000			
pindle Load(%)		0%	1		
Setup M:			I		

#### • Cursor down to VPS in the menu and select it.

 Don't forget that the arrow keys are used to navigate the Haas menus.

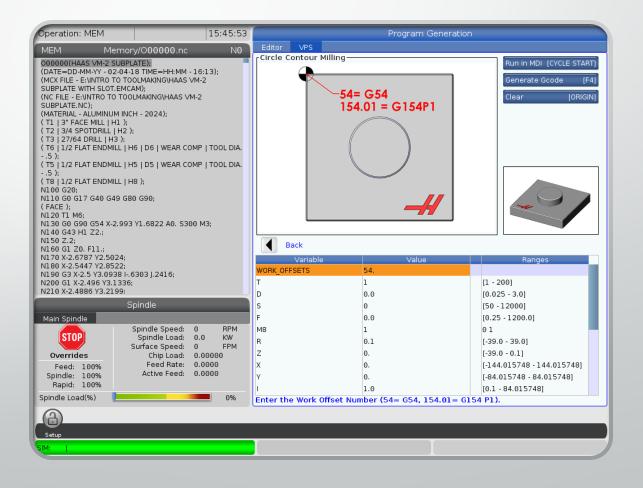


Operation: MEM		15:45:33				Program	Generation			
MEM Mem	orv/000000.nc	NO	Editor	VPS						
MEM      Memory        000000(HAAS VM-2 SUB        (DATE=DD-MM-YY - 02-04        (MCX FILE - E:INNTRO TO T        SUBPLATE WITH SLOT.EM        (NC FILE - E:INNTRO TO TO        SUBPLATE WITH SLOT.EM        (INC FILE - E:INNTRO TO TO        SUBPLATE.NC);        (MATTERIAL - ALUMINUM IN IN        (IT ] 3" FACE MILL   H1.)        (T2 ] 3/4 SPOTDRILL   H1.)        (T3 ] 1/2 FLAT ENDMILL   5.);        (T5 ] 1/2 FLAT ENDMILL   5.);        (T8 ] 1/2 FLAT ENDMILL   N100 G20;        N110 G0 G17 G40 G49 C0;        N120 TL M6;	1-18 TIME=HH:MM - OOLMAKING\HAAS V CAM); JOLMAKING\HAAS V ICH - 2024); ; ; H6   D6   WEAR COI H5   D5   WEAR COI H8 );	16:13); /M-2 4-2 MP   TOOL DIA.	Editor Recently VPS/0-Rin VPS/Face VPS/Engr VPS/Drillir VPS/Circle VPS/Circle	g Groove Milling aving g Pocket N	filling				To Switch Boxe	s [F4] [ENTER]
N120 G1 G90 G54 X-2.99 N140 G43 H1 Z2.; N150 Z.2; N160 G1 Z0. F11.; N170 X-2.6787 Y2.5024; N180 X-2.5447 Y2.5024; N190 G3 X-2.5 Y3.0938   N200 G1 X-2.496 Y3.133 N210 X-2.4886 Y3.2199;	6303 J.2416; 6;	0 M3;	Current PROBING VPS	Director			(TEXT) [F1], or	Size <dir> <dir></dir></dir>	Last Modif 08/18/17 19:12 08/18/17 19:12	>
	Spindle		CUSTOM					<dir></dir>	08/18/17 19:13	>
Main Spindle STOP Overrides Feed: 100% Spindle: 100% Rapid: 100% Spindle Load(%)	Spindle Speed: Spindle Load: Surface Speed: Chip Load: Feed Rate: Active Feed:	0 FPM 0.00000 0.0000								

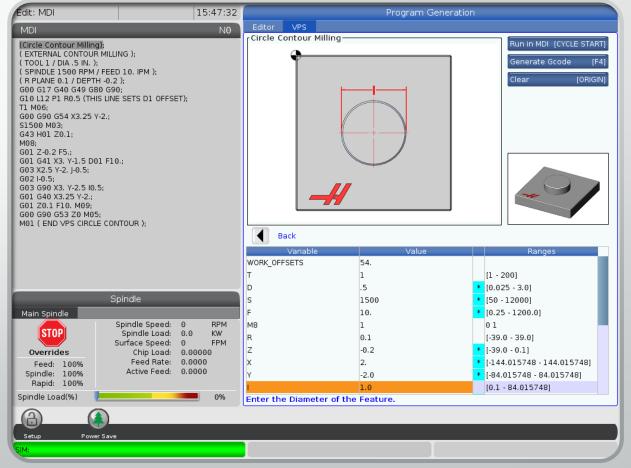
Select "Circle Contour Milling".

Operation: MEM		15:45:39	Program Ge	neration
MEM Memory	y/000000.nc	NO	Editor VPS	
000000(HAAS VM-2 SUBPLJ (DATE=DD-MM-YY - 02-04-1) (MCX FILE - E:\INTRO TO TOO SUBPLATE WITH SLOT.EMCA (NC FILE - E:\INTRO TO TOOL SUBPLATE WITH SLOT.EMCA (NC FILE - E:\INTRO TO TOOL SUBPLATE.NC); (T1 ] 3' FACE MILL   H1 ); (T3   27/64 DRILL   H3 ); (T6   1/2 FLAT ENDMILL   H2 ); (T5   1/2 FLAT ENDMILL   H3 ); (T6   1/2 FLAT ENDMILL   H4 .5 ); (T5   1/2 FLAT ENDMILL   H4 N100 G20; N110 G0 G17 G40 G49 G80 (FACE ); N120 T1 M6; N130 G90 G54 X-2.993 N140 G43 H1 Z2; N150 Z.2; N160 G1 Z0. F11; N170 X-2.6787 Y2.5024;	8 TIME=HH:MM - 16; DLMAKING\HAAS VM-2 WAY; MAKING\HAAS VM-2 H - 2024); 6   D6   WEAR COMP 5   D5   WEAR COMP 8 ); 0 G90;	] TOOL DIA.   TOOL DIA.	VPS Back Forward Search (TE) Current Directory: VPS/	To Switch Boxes [F4]
N180 X-2.5447 Y2.8522;			File Name	Size Last Modified
N190 G3 X-2.5 Y3.0938 I6 N200 G1 X-2.496 Y3.1336:			Circle Contour Milling	23150 08/18/17 19:12
N210 X-2.4886 Y3.2199:			Circle Pocket Milling	24126 08/18/17 19:12
St	oindle		Drilling	159769 08/18/17 19:12
	Jinde		Engraving Face Milling	19322 08/18/17 19:12 215766 08/18/17 19:12
Main Spindle			O-Ring Groove Milling	16478 08/18/17 19:12
S S	Spindle Speed: 0	RPM	Square Contour Milling	35420 08/18/17 19:12
STOP	Spindle Load: 0.0		Square Pocket Milling	48912 08/18/17 19:12
	Surface Speed: 0	FPM	Straight Milling	10792 08/18/17 19:12
Overrides		0000	Thread Milling	32929 08/18/17 19:12
Feed: 100%		000		
Spindle: 100%	Active reeu: 0.0	000		
Rapid: 100%				
Spindle Load(%)		0%	1	
Setup			T	
ISIM:				

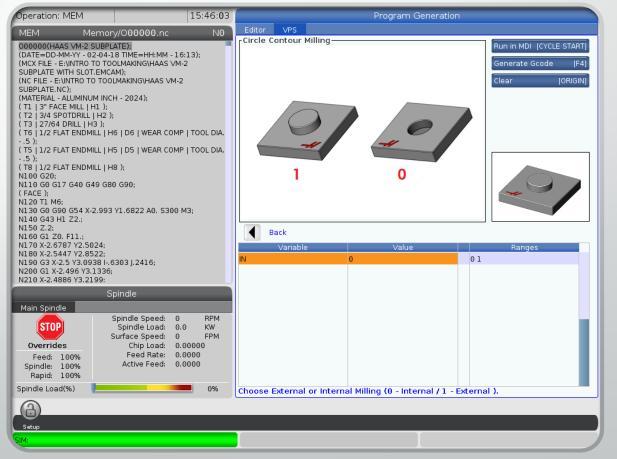
- Start filling out the variables in the form.
- The prompts will walk the operator through each inquiry. In this case, it says to insert the desired work offset number.



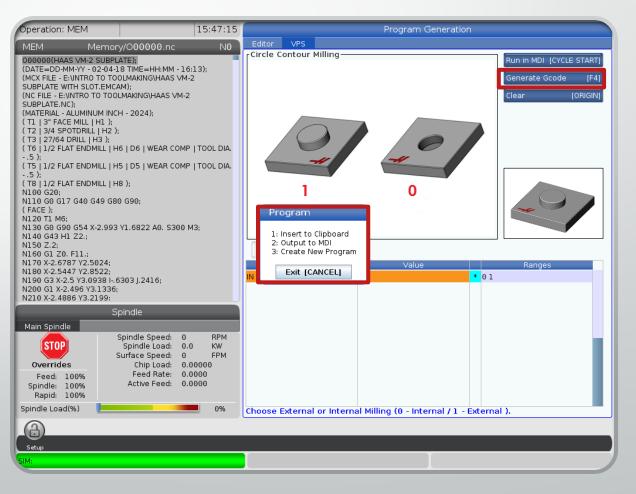
- After the variables are filled in, it should look something like this.
- This tells the machine how to machine the part in relation to the PRZ.



 The next screen asks whether the feature is a boss or pocket. In this case it is a pocket.



- Select generate code to bring up the output menu.
- Select "2", which will output the code to MDI.



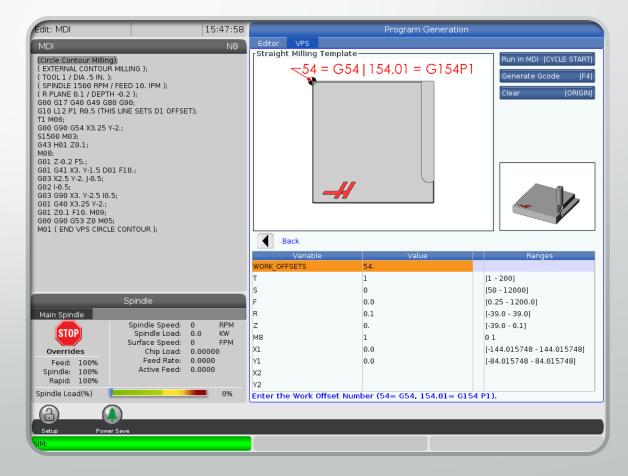
• The program is then output to MDI.

dit: MDI		15:47:23				MDI			_
MDI		N0							
(Circle Contour Milling) (EXTERNAL CONTOUR N				Active	Codes	_	Act	ive Tool	Coolant
(TOOL 1 / DIA.5 IN.); (SPINDLE 1500 RPM / I (SPINDLE 1500 RPM / I GO G17 G40 G49 G80 G10 L12 P1 R0.5 (THIS T1 M06; G40 G90 G54 X3.25 Y-3 S1500 M03; G43 H01 Z0.1; M08; G01 Z-0.2 F5.; G01 G41 X3, Y-1.5 D01 G03 X2.5 Y-2, J-0.5; G02 I-0.5; G03 G90 X3, Y-2.5 I0.5 G01 G40 X3.25 Y-2.; G01 G40 X3.25 Y-2.; G01 G01 F10. M09; G00 G90 G53 Z0 M05; M01 (END VPS CIRCLE	-0.2 ); G90; LINE SETS D1 OFFSE 2.; F10.;	:T);	G00 G90 G40 G80 G54	Rapid Motion Absolute Positii Cutter Compen: Cycle Cancel Work Offset #5: D00 H00	sation Can	rcel T0	Tool: 1 Type: Tool Group: Max Load: 0 Life: <b>100%</b> <b>Next Tool</b> Pocket: 1 Tool #: 2		0/ff
	Spindle		Positior	ns F	rogram	G54 G49		Timers And C	Counters
Main Spindle STOP Overrides Feed: 100% Spindle: 100% Spindle Load(%)	Spindle Speed: Spindle Load: Surface Speed: Chip Load: Feed Rate: Active Feed:	0.0 KW	í x í corr í corr i correction i x i x i x i x i x i x i x i x i x i x	(IN) 0.0000 0.0000 0.0000 0.0000			0% 0%	This Cycle: Last Cycle: Remaining M30 Counter #1: M30 Counter #2: Loops Remaining:	0:00:0 0:00:0 0:00:0
A (									

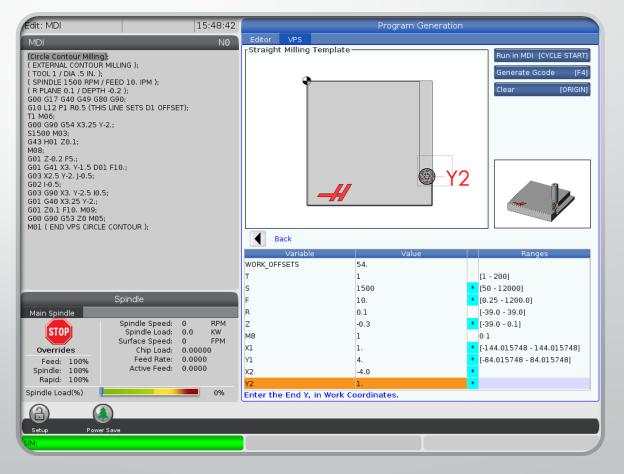
- This next VPS programming example will go over how to program a "Straight Milling toolpath".
- Start by going back to Edit/VPS, and select "Straight Milling" from the VPS menu.

Edit: MDI	15:49:12	Program Gene	eration
MDI	NØ	Editor VPS	
(Straight Milling Template): (TOOL 1); (SPINDLE 1500 RPM / FEED 10. IPM ); (R PLANE 0.1 / DEPTH -0.3); G00 G90 G54 G49 G80 G90; T1 M06; G00 G90 G54 X1. Y4. S1500 M03; G43 H01 Z0.1 M08; G01 S4. Y1. F10.; G01 Z4. Y1. F10.; G01 Z4. Y1. F10.; G01 G90 G53 Z0 M05; M01 (END VPS STRAIGHT MILLING );		Current Directory: VPS/ File Name Circle Contour Milling	) [F1], or [F1] to clear.
		Circle Pocket Milling Drilling	24126 08/18/17 19:12 159769 08/18/17 19:12
Spindle		Engraving	19322 08/18/17 19:12
Main Spindle		Face Milling	215766 08/18/17 19:12
		O-Ring Groove Milling	
Spindle Speed: 6	9 RPM		16478 08/18/17 19:12
	9 RPM 9.0 KW	Square Contour Milling	35420 08/18/17 19:12
Spindle Load: 0 Surface Speed: 0	0.0 KW 0 FPM	Square Contour Milling Square Pocket Milling	35420 08/18/17 19:12 48912 08/18/17 19:12
Spindle Load: 0 Surface Speed: 0 Overrides Chip Load: 0	0.0 KW 0 FPM 0.00000	Square Contour Milling Square Pocket Milling Straight Milling	35420      08/18/17      19:12        48912      08/18/17      19:12        10792      08/18/17      19:12
STOP Overrides Feed: 100% Spindle Load: 0 Surface Speed: 0 Surface Speed: 0 Surface Speed: 0 Spindle Load: 0 Surface Speed: 0 Seindle Load: 0 Spindle Load: 0 Surface Speed: 0 Seindle Load: 0 Surface Speed: 0 Spindle Load: 0 Spindle	0.0 KW 0 FPM 0.00000 0.0000	Square Contour Milling Square Pocket Milling	35420 08/18/17 19:12 48912 08/18/17 19:12
Stop Overrides Feed: 100% Spindle: 100% Spindle: 100% Spindle: 100%	0.0 KW 0 FPM 0.00000	Square Contour Milling Square Pocket Milling Straight Milling	35420      08/18/17      19:12        48912      08/18/17      19:12        10792      08/18/17      19:12
Stop Overrides Feed: 100% Spindle Load: 0 Surface Speed: 0 Surface Speed: 0 Surface Speed: 0 Surface Speed: 0 Surface Speed: 0 Stop	0.0 KW 0 FPM 0.00000 0.0000	Square Contour Milling Square Pocket Milling Straight Milling	35420      08/18/17      19:12        48912      08/18/17      19:12        10792      08/18/17      19:12
STOP Spindle Load: 0 Surface Speed: 0 Overrides Chip Load: 0 Feed: 100% Feed Rate: 0 Spindle: 100% Active Feed: 0	0.0 KW 0 FPM 0.00000 0.0000	Square Contour Milling Square Pocket Milling Straight Milling	35420      08/18/17      19:12        48912      08/18/17      19:12        10792      08/18/17      19:12
Spindle Load: 0 Surface Speed: 0 Surface Speed: 0 Surface Speed: 0 Surface Speed: 0 Surface Speed: 0 Surface Speed: 0 Spindle: 100% Spindle: 100% Spindle: 100%	0.0 KW 0 FPM 0.00000 0.0000 0.0000	Square Contour Milling Square Pocket Milling Straight Milling	35420      08/18/17      19:12        48912      08/18/17      19:12        10792      08/18/17      19:12
Stop      Spindle Load:      Spindle Load: <td>0.0 KW 0 FPM 0.00000 0.0000 0.0000</td> <td>Square Contour Milling Square Pocket Milling Straight Milling</td> <td>35420      08/18/17      19:12        48912      08/18/17      19:12        10792      08/18/17      19:12</td>	0.0 KW 0 FPM 0.00000 0.0000 0.0000	Square Contour Milling Square Pocket Milling Straight Milling	35420      08/18/17      19:12        48912      08/18/17      19:12        10792      08/18/17      19:12
Stop Overrides Feed: 100% Spindle: 100% Spindle Load(%)	0.0 KW 0 FPM 0.00000 0.0000 0.0000	Square Contour Milling Square Pocket Milling Straight Milling	35420      08/18/17      19:12        48912      08/18/17      19:12        10792      08/18/17      19:12
Stop Overrides Feed: 100% Spindle: 100% Rapid: 100% Spindle Load(%) Spindle Load(%)	0.0 KW 0 FPM 0.00000 0.0000 0.0000	Square Contour Milling Square Pocket Milling Straight Milling	35420      08/18/17 19:12        48912      08/18/17 19:12        10792      08/18/17 19:12

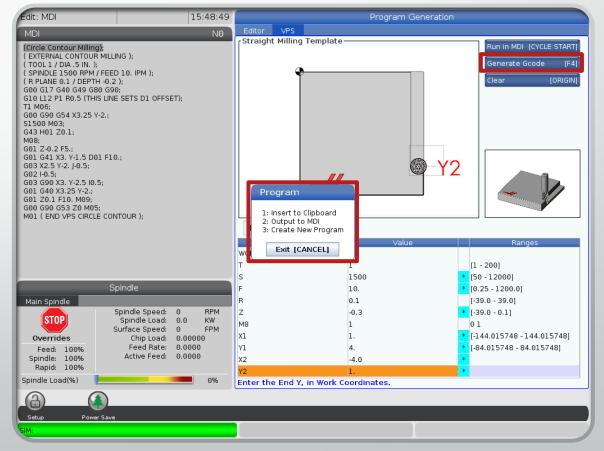
• Again, fill in the variables. The first inquiry asks which work offset to use.



 Fill out the variables like so. This tells the machine how to cut the part in relation to the PRZ.



 Select "Generate Gcode" by pressing F4 as prompted. Select "2" to output to MDI.



- The program is then output to MDI.
- Notice that both of these examples were very similar. VPS is designed to be intuitive, and should be easy to navigate using prompts.
- This code can also be posted to memory, or even saved to the clipboard for later use.

		15:48:57			MDI			
MDI		N0		_				
(Straight Milling Tem ( TOOL 1 );	nplate);		A	ctive Codes		Acti	ive Tool	Coolant
( SPINDLE 1500 RPM ( R PLANE 0.1 / DEPT						Tool: 1	Offset: 0	Off
G00 G17 G40 G49 G			G00 Rapid Mo G90 Absolute			Type:	None	
T1 M06; G00 G90 G54 X1. Y4	. S1500 M03:			mpensation Car	icel	Tool Group:		1/1
G43 H01 Z0.1 M08;			G80 Cycle Car	•		Max Load: 0	)	
G01 G90 Z-0.3 F5.; G01 X-4. Y1. F10.;			G54 Work Offs			Life: 100%		
G01 Z0.1 F5. M09;	_					Next Tool		
G00 G90 G53 Z0 M0 M01 ( END VPS STRA			D00	H00 M00	TO	Pocket: 1		
						Tool #: 2		0/1
	Spindle		Positions	Program	G54 G49		Timers And 0	Counters
Main Spindle				Program	G54 G49			Counters
	Spindle Speed:	0 RPM 0.0 KW	(IN)	Program	G54 G49		This Cycle:	0:00:0
STOP	Spindle Speed: Spindle Load: Surface Speed:	0.0 KW 0 FPM		Program	G54 G49	0% L	This Cycle: Last Cycle:	0:00:0
STOP Overrides	Spindle Speed: Spindle Load: Surface Speed: Chip Load:	0.0 KW 0 FPM 0.00000	(IN)	Program	G54 G49	0% L	This Cycle: Last Cycle: Remaining	0:00:0
Overrides Feed: 100%	Spindle Speed: Spindle Load: Surface Speed: Chip Load: Feed Rate:	0.0 KW 0 FPM	(IN) (in)	Program	G54 G49	0% L F 0% p	This Cycle: Last Cycle: Remaining M30 Counter #1:	0:00:0
STOP	Spindle Speed: Spindle Load: Surface Speed: Chip Load: Feed Rate:	0.0 KW 0 FPM 0.00000 0.0000	(IN)	Program	G54 G49	0% L 6 0% n 0% n	This Cycle: Last Cycle: Remaining M30 Counter #1: M30 Counter #2:	0:00:0
Overrides Feed: 100% Spindle: 100%	Spindle Speed: Spindle Load: Surface Speed: Chip Load: Feed Rate:	0.0 KW 0 FPM 0.00000 0.0000	(IN) (in)	Program	G54 G49	0% L 6 0% n 0% n	This Cycle: Last Cycle: Remaining M30 Counter #1:	0:00:0
Overrides Feed: 100% Spindle: 100% Rapid: 100%	Spindle Speed: Spindle Load: Surface Speed: Chip Load: Feed Rate:	0.0 KW 0 FPM 0.00000 0.0000 0.0000	(IN) (in)	Program	G54 G49	0% L 6 0% n 0% n	This Cycle: Last Cycle: Remaining M30 Counter #1: M30 Counter #2:	0:00:0 0:00:0
Stop Overrides Feed: 100% Spindle: 100% Rapid: 100%	Spindle Speed: Spindle Load: Surface Speed: Chip Load: Feed Rate:	0.0 KW 0 FPM 0.00000 0.0000 0.0000	(IN) (in)	Program	G54 G49	0% L 6 0% n 0% n	This Cycle: Last Cycle: Remaining M30 Counter #1: M30 Counter #2:	Counters 0:00:0 0:00:0 0:00:0
Overrides Feed: 100% Spindle: 100% Rapid: 100% Spindle Load(%)	Spindle Speed: Spindle Load: Surface Speed: Chip Load: Feed Rate:	0.0 KW 0 FPM 0.00000 0.0000 0.0000	(IN) (in)	Program	G54 G49	0% L 6 0% n 0% n	This Cycle: Last Cycle: Remaining M30 Counter #1: M30 Counter #2:	0:00:0 0:00:0

#### Program Structure

- When starting a new program, the material, stock size, PRZ, and tool
   list should be included in the beginning of the program as notes.
- 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.

%
O00001 (MILL FORMAT);
N1 (NOTES)
G00 G17 G20 G40 G49 G54 G64 G80 G90 G94 G98 G100;
G53 Z0.;
T1 M06;
X0. Y0. S1000 M03;
G43 H01 Z1. M08;
(BEGIN TOOL BODY);
;
;
;
;
(END TOOL BODY);
G00 G80 Z1. M09;
G49 G53 Z0.;
M01

#### Program Structure

#### • Programs are broken up into 3 basic sections:

- Start-up
  - In the example program to the right, line 4-8 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 9-14. This will vary in length and complexity depending upon the operation.
- Shut Down
  - In the example program this would be lines 15-17. These lines will have the same format for each tool that is programmed.

Start-up	% O00001 (MILL FORMAT); N1 (NOTES) G00 G17 G20 G40 G49 G54 G64 G80 G90 G94 G98 G100; G53 Z0.; T1 M06; X0. Y0. S1000 M03; G43 H01 Z1. M08; (BEGIN TOOL BODY);
Cutting	;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;
Shut down	, (END TOOL BODY); G00 G80 Z1. M09; G49 G53 Z0.; M01

#### **Tool Path Startup**

- The first 4 lines of each tool path contain the start up of the tool. This should be done for each toolpath.
- The format is as follows:
  - Safe start-up line
  - Send Z home
  - Send X & Y home, Tool change/offset
  - Turn spindle on

% OO0001 (MILL FORMAT); N1 (NOTES) G00 G17 G20 G40 G49 G54 G64 G80 G90 G94 G98 G100; G53 Z0.; T1 M06; X0. Y0. S1000 M03; G43 H01 Z1. M08; (BEGIN TOOL BODY); ; ; ; (END TOOL BODY); G00 G80 Z1. M09; G49 G53 Z0.; M01

#### **Tool Path Startup**

#### Safe Start-up

- This line is used to prepare the machine for cutting:
  - Goo Rapid
  - G17 Set work area plains to X & Y
  - G20 Inch
  - G40 Cancel tool nose radius comp.
  - G49 Cancel tool length comp.
  - G54 Active work offset
  - G64 Cancels exact stop (G61)
  - G8o cancel canned cycle
  - G90 Set dimensioning to absolute
  - G94 Deactivates Inverse Time Feed Mode Returns the control to Feed Per Minute mode.
  - G98 Set Canned Cycle Initial Point Return Canned cycles retract to the start point instead of the retract point.
  - G100- Turns off mirror imaging

#### %

O00001 (MILL FORMAT); N1 (NOTES) G00 G17 G20 G40 G49 G54 G64 G80 G90 G94 G98 G100; G53 Z0.; T1 M06; X0. Y0. S1000 M03; G43 H01 Z1. M08; (BEGIN TOOL BODY); ; ; ; (END TOOL BODY); G00 G80 Z1. M09; G49 G53 Z0.; M01

#### **Tool Path Startup**

#### • T1 M6

- This calls up tool 1.
- An M6 is required for the tool to change.
- Xo.Yo.
  - This moves both X and Y to the work PRZ.
- S1000 M3
  - This turns the spindle on clockwise at 1000 RPM.

#### % OO0001 (MILL FORMAT); N1 (NOTES) G00 G17 G20 G40 G49 G54 G64 G80 G90 G94 G98 G100; G53 Z0.; T1 M06; X0. Y0. S1000 M03; G43 H01 Z1. M08; (BEGIN TOOL BODY); ; ; ; (END TOOL BODY); G00 G80 Z1. M09; G49 G53 Z0.; M01

#### Shut Down

- After the cutting is complete, the tool path can be shut down. This is done in the final 3 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.
  - Go G8o Z1. Mo9
    - This puts the machine back in rapid, cancels canned cycles, lifts Z 1" above the PRZ, and turns off the coolant.
  - G49 G53 Zo.
    - This cancels tool length comp. while returning Z to home.
  - M1
    - This stops the program when optional stop is active. This is most commonly used at the end of a cycle.

% O00001 (MILL FORMAT); N1 (NOTES) G00 G17 G20 G40 G49 G54 G64 G80 G90 G94 G98 G100; G53 Z0.; T1 M06; X0. Y0. S1000 M03; G43 H01 Z1. M08; (BEGIN TOOL BODY); ; ; ; (END TOOL BODY); G00 G80 Z1. M09; G49 G53 Z0.; M01

# Writing Safe Programs

- Regardless of how many tools are 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 Z then X & Y 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 can not rerun the second operation without manually changing the tool.

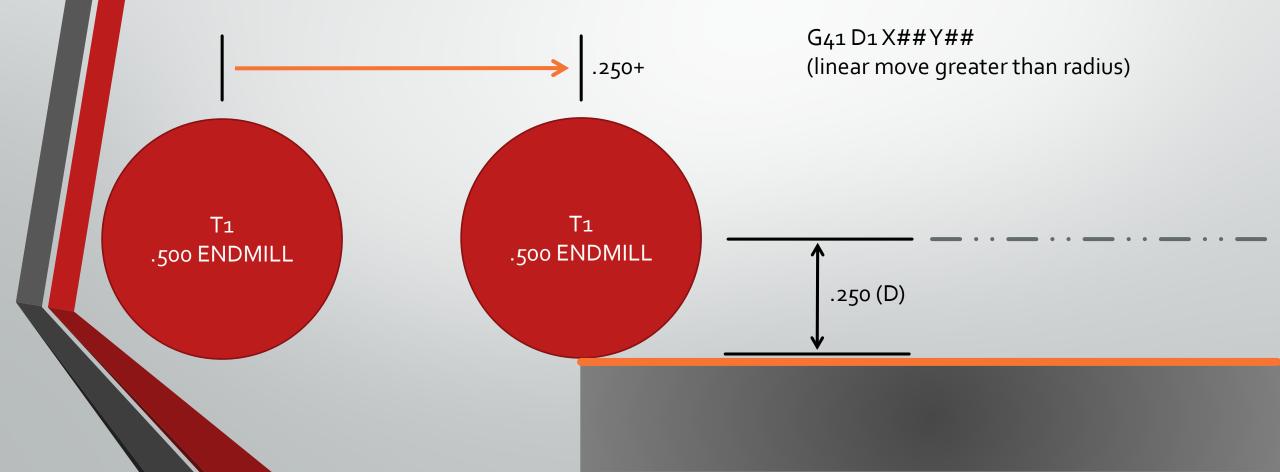
# Diameter Cutting Compensation

Start -

- Diameter comp. factors in the tools diameter and offset the tool while cutting. It does this by shifting over away from the part the distance of the tool radius.
- Cutter compensation can only be activated on linear moves.
  - Cutter compensation will not activate on Go, G2 or G3 moves.

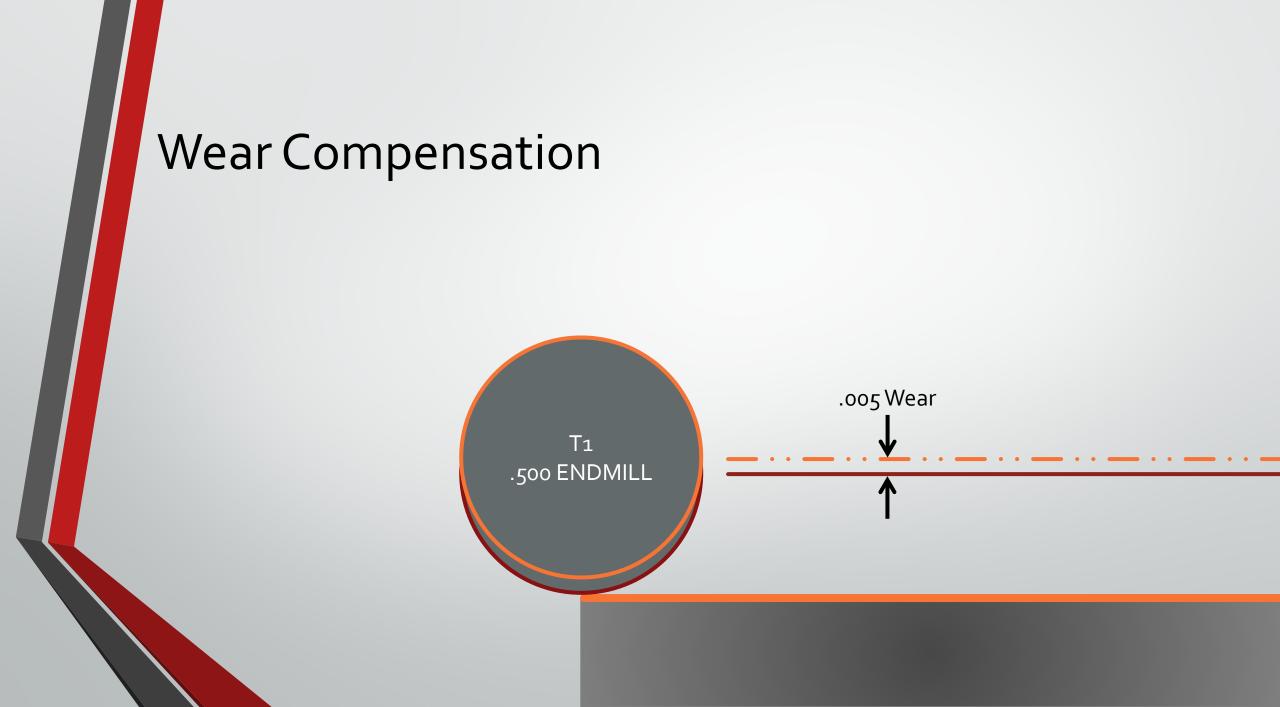
Er	nd		
7			
			L
			L
			L
			L
		PART	L
			L
			L
			L
	-		

# Diameter Cutting Compensation



# Wear Compensation

- Wear compensation is used to adjust the placement of the tool in small increments to make up for variables such as tool wear and deflection.
- Wear uses these small adjustments to fine tune the tool and cut as precisely as possible.



# Incremental Looping

- M97 combined with P and L values allow for the program to run through repeated code for a given number of times.
  - P- Program number to find in program
  - L- number of times to loop through the program
  - M99- ends the repeated code and returns to the M97 line
- This strategy is especially useful when a single toolpath needs to happen repeatedly. As the machine can be programed to run a toolpath, then move to another location before looping back to the beginning of that toolpath.

M97 P1000 L2 (L2 command will run the N1000 line twice) M30 N1000 (sub routine) ; ; ; ; M99 (end sub routine)

# Multiple Tool and Work Offsets

- There are times when using multiple tool and work offsets at the same time are beneficial. There are many reasons to use more than one work offset or tool offsets:
  - Multiple setups can be included in one program, often a first and second operation.
  - One tool can be used with multiple offsets. For example, this can be helpful when two different tolerances are needed while using the same tool.
  - For this to work, Setting 15 must be turned on in the control settings.

#### 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.

# G73 High Speed Peck Drilling

- G73 is used to drill holes quickly while pecking. This helps to evacuate chips and improve coolant flow over the tool and in the hole.
- A G73 Cycle looks like this:
  - G73 X5. Y2. Z-.75 Q.25 R.1 F3.

% O00001 (MILL FORMAT); N1 (NOTES) G00 G17 G20 G40 G49 G54 G64 G80 G90 G94 G98 G100; G53 Z0.; T1 M06; X0. Y0. S1000 M03; G43 H01 Z1. M08; (BEGIN TOOL BODY); G73 X5. Y2. Z-.75 Q.25 R.1 F3. (END TOOL BODY); G00 G80 Z1. M09; G49 G53 Z0.; M01

# G73 High Speed Peck Drilling

- G73 X5. Y2. Z-.75 Q.25 R.1 F3.
  - X5. & Y2. Is the hole location
  - Z-.75 Is the hole depth
  - Q.25 Is the peck depth
  - R.1 Is the the retract plain
  - F3. Is the feedrate

# G73 High Speed Peck Drilling

- G73 X5. Y2. Z-.75 R.1 Q.25 F3.
- Optional G73 codes:
  - I First peck depth
  - J Amount to reduce each peck
  - K Minimum peck depth
  - L Number of holes to drill if in incremental mode (G91)
  - P Pause at hole bottom (seconds)

# G81 Drilling Canned Cycle

- G81 is a drilling canned cycle. Once the Z travel starts, it does not stop until the depth is reached. After the depth is achieved, the tool then rapids out of the hole.
- A G81 cycle looks like this:
  - G81 X3. Y2. Z-1. R.1 F3.

O00001 (MILL FORMAT); N1 (NOTES) G00 G17 G20 G40 G49 G54 G64 G80 G90 G94 G98 G100; G53 Z0.; T1 M06; X0. Y0. S1000 M03; G43 H01 Z1. M08; (BEGIN TOOL BODY); G81 X3. Y2. Z-1. R.1 F3. , , (END TOOL BODY); G00 G80 Z1. M09; G49 G53 Z0.; M01

%

# G81 Drilling Canned Cycle

- G81 X3. Y2. Z-1. R.1 F3.
  - X3. & Y2. Is the hole location
  - Z-1. Is the hole depth
  - R.1 Is the retract plain
  - F3. Is the feedrate

#### G82 Drilling Canned Cycle w/ Dwell

- G82 is the same cycle as G81 but adds a dwell when the depth is achieved. This allows time for the tool to rotate and cut, relieving any possible cutting pressure.
  - This is often done with spot drills
- A G82 cycle looks like this:
  - G82 X3. Y2. Z-1. P1 R.1 F3.

% O00001 (MILL FORMAT); N1 (NOTES) G00 G17 G20 G40 G49 G54 G64 G80 G90 G94 G98 G100; G53 Z0.; T1 M06; X0. Y0. S1000 M03; G43 H01 Z1. M08; (BEGIN TOOL BODY); G82 X3. Y2. Z-1. P.1 R.1 F3. , (END TOOL BODY); G00 G80 Z1. M09; G49 G53 Z0.; M01

# G82 Drilling Canned Cycle W/ Dwell

- G82 X3. Y2. Z-1. P.1 R.1 F3.
  - X3. & Y2. Is the hole location
  - Z-1. Is the hole depth
  - P.1- dwell 1 second
  - R.1 Is the retract plain
  - F3. Is the feedrate

### G83 Normal Peck Drilling

- G8<sub>3</sub> is used to drill while pecking. The drill retracts completely out of the hole at a set increment.
- A G8<sub>3</sub> cycle looks like this:
  - G83 X3. Y2. Z-1. R.1 Q.25 F3.

% O00001 (MILL FORMAT); N1 (NOTES) G00 G17 G20 G40 G49 G54 G64 G80 G90 G94 G98 G100; G53 Z0.; T1 M06; X0. Y0. S1000 M03; G43 H01 Z1. M08; (BEGIN TOOL BODY); G83 X3. Y2. Z-1. R.1 Q.25 F3. (END TOOL BODY); G00 G80 Z1. M09; G49 G53 Z0.; M01

# G83 Normal Peck Drilling

- G83 X3. Y2. Z-1. R.1 Q.25 F3.
  - X3. & Y2. Is the hole location
  - Z-1. Is the hole depth
  - R.1 Is the retract plain
  - Q.25 Is the pecking increments
  - F3. Is the feedrate

# G83 Normal Peck Drilling

- G83 X3. Y2. Z-1. R.1 Q.25 F3.
- Optional G83 codes:
  - I First peck depth
  - J Amount to reduce each peck
  - K Minimum peck depth
  - L Number of holes to drill if in incremental mode (G91)
  - P Pause at hole bottom (seconds)

## G84 Tapping

G84 is a tapping cycle. Tapping cycles orient the tool before starting a repeatable plunge into the part at the desired feed. The speed and feed need to correspond to the pitch of the thread for this toolpath to work. Otherwise, the tool or part will break and the thread will be ruined.

A G84 cycle looks like this:

• G84 X3. Y2. Z-.625 R.1 F10.

% O00001 (MILL FORMAT); N1 (NOTES) G00 G17 G20 G40 G49 G54 G64 G80 G90 G94 G98 G100; G53 Z0.; T1 M06; X0. Y0. S250 M03; G43 H01 Z1. M08; (BEGIN TOOL BODY); G84 X3. Y2. Z-.625 R.1 F19.23 (END TOOL BODY); G00 G80 Z1. M09; G49 G53 Z0.; M01

#### G84 Tapping

#### • G84 X3. Y2. Z-.625 R.1 F19.23

- X3. & Y2. Is the hole location
- Z-.625 Is the hole depth
- R.1 Is the retract plain
- F19.23 Is the feed rate
  - Feed rate is found with the following;
    - RPM/TPI= IPM
    - 250/13= 19.23
- Setting 130 on the control can be used to control the tap retract speed. The range is 0-4, 2 being 2 times the feed rate, 3 for 3 times the feed rate and 4 for 4 times the feed rate.
- Setting 133 on the control can be used during tapping. This ensures the spindle is oriented during tapping. This is done so threads can be tapped a second time or peck tapping can be used.

# G84 Tapping

- G84 X3. Y2. Z-.625 R.1 F10.
- Optional G84 codes:
  - J Retract multiple (How much faster exit is than entry)
  - L Number of holes to drill if in incremental mode (G91)
  - S Spindle speed

#### G187 Smoothing

- *G187* is an accuracy command that can set and control both the smoothness and max corner rounding value when cutting a part. The format for using *G187* is *G187 Pn Ennnn*.
- P Controls the smoothness level, P1(rough), P2(medium), or P3(finish). Temporarily overrides Setting 191.
  E Sets the max corner rounding value. Temporarily overrides Setting 85.

Setting 191 sets the default smoothness to the user specified ROUGH, MEDIUM, or FINISH when *G187* is not active. The Medium setting is the factory default setting.

Think of this as a toolpath tolerance. How far the tool can deviate from the programmed line while cutting. In the example, the straight line is the programmed path, while the curved line is the actual machined path. Rough would be faster but allow for greater deviation, while finish would be slower but stay closer to the programmed line.



G12, G13 Circular Pocket Milling

- Circular Pocket Milling CW (G12) & Circular Pocket Milling CCW (G13) mill circular pockets.
- They can both be used for roughing or finishing operations. Radial cutter comp. is built in, so no G41 or G42 is needed.
- The only difference between G12 & G13 is the cutting direction. G12 is CW and G13 is CCW.

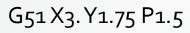
#### G12, G13

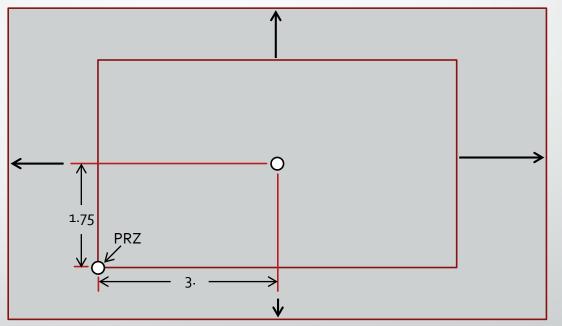
- G12 I0.3 K1.5 Q1. F10. Z-1.2 D01
  - D Tool radius or diameter selection
  - F Feedrate
  - I Radius of first circle (or finish if no K). I value must be greater than Tool Radius, but less than K value.
  - K Radius of finished circle (if specified)
  - L Loop count for repeating deeper cuts
  - Q Radius increment, or stepover (must be used with K)
  - Z Depth of cut or increment

#### G51 Scaling

- G51 is used to scale toolpaths. It uses a set reference point to scale from with the desired scale factor.
- G51 example:
  - G51 X3. Y1.75 P1.5
    - X, Y, and Z The scale reference point
    - P Scale factor
- Features can be scaled by a factor of 0.001 to 999.999.
- G51 doesn't need a reference location provided, but if one isn't included, the last location will be used instead.

# G51 Scaling





## G52 Set Work Coordinate Shift

- G52 can be used as a global work shift.
  - The X,Y,Z values that are input in the control shift all work offsets by the specified amount. G52 does not change the work offset values, rather when the offset is activated it adds the values to the specified work offset.
  - G52 can be used on nested parts in a fixture or on identical features on a part.

#### G68 Rotational Shift

- G68 rotates the toolpath in relation to a point at a set angle.
- G68 example:
  - G68 Xo. Yo. R-90.
    - X & Y Point of rotation
    - R Degree of rotation
- G68 rotates about the active plain, which is X & Y (G17) in this case. G18 or G19 can be activated as well to rotate the toolpath about X & Z or Y & Z.

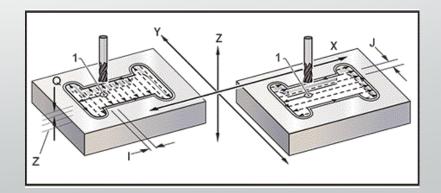
# G68 Rotational Shift

#### G68 Xo.Yo. R-90.

Before	After	

G150 General Purpose Pocket Milling

- General Purpose Pocket Milling cuts out a defined profile defined by a P value and a local sub-program.
- This toolpath leads into the pocket in Z, Roughs the profile, then runs a finish pass.



G150 General Purpose Pocket Milling

#### • G150 example:

- G150 X2.25 Y3.5 Z-.5 G41 J0.35 K.01 Q0.25 R.1 P1002 D01 F15.
  - X & Y X and Y start position
  - Z Final depth
  - G41 Radial tool comp.
  - I X axis cut increment (positive value)
  - J Y axis cut increment (positive value)
  - K Finishing pass amount (positive value)
  - Q Incremental Z axis cut depth/pass (positive value)
  - R Retract plain position
  - P Sub-program number
  - D Tool radius/diameter offset
  - F Feed

### M97, M98, M99 Subprograms

- M97 and M98 both call up sub-programs, which are mini programs that can be called upon throughout a program.
  - M97 calls up a sub routine that is stored within the main program, after the M30.
  - M98 calls up a sub program that is stored as a separate program in the machine memory.
- Both options use two values:
  - P Line No. (M97) and Program number (M98)
  - L Repeat sub-program (1-99)

### M97, M98, M99 Subprograms

- M99 has three purposes:
- **1.** It can return a program to its beginning and continue running it.
- 2. It is used at the end of a sub-program to return to the main program.
- 3. It can act as a GOTO command.
- Therefore, M99 is essential when using M97 & M98, otherwise the program would be stuck in the sub program.

### M97, M98, M99 Subprograms

 There are many possibilities once M97, M98, and M99 are put together. Here are some examples of how sub programs are called up to and from the main program.

%

#### %

Ooooo1; M97 P100 L2 (Calls N100 sub 2 times); M30; N100 (Subprogram); G00 G91 X1.; G83 Z-1.5 R.1 Q.375 F4.; M99 (Returns to main program); % Ooooo2; M98 P200 L4 (Calls O200 sub 4 times) ; M30 ; % ✓ Separate program in memory ↓ % Ooo200 (Subprogram); Moo ; M99 (Return to main program) ;

#### **Block Delete**

 If Block Delete is activated, all program lines with a "/" in the front will be skipped.



% O00001 (MILL FORMAT); N1 (NOTES) G00 G17 G20 G40 G49 G54 G64 G80 G90 G94 G98 G100; G53 Z0.; T1 M06; X0. Y0. S1000 M03; G43 H01 Z1. M08; (BEGIN TOOL BODY); /G83 X3. Y2. Z-1. R.1 Q.25 F3. (END TOOL BODY); G00 G80 Z1. M09; G49 G53 Z0.; M01

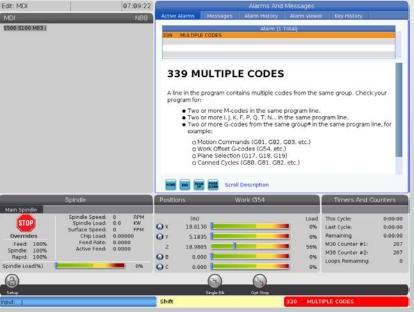
#### **Program Trouble Shooting**

- 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.
- If a decimal point was missing in a line (G1 X2. Y2) the graphics would show the Y axis not moving to the proper position.

