

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.

- 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.



Edit: MDI 07:40:26	Program Generat	ion	
MDI N451	Editor VPS Shape Creator		
Initial Initial (Sadial Drilling); (Saferty Line BELOW); (Good S54 G18 G40 G90 G97 G99; (TOOL = 5 / OFFSET = 5); (WORK OFFSET = 54); (WORK OFFSET = 54); (UORK OFFSET = 54); (CANIS START ANGLE = 0.0); (= or EQUALY SPACED HOLES = 3); T505; (G54; G98 (FEED PER MINUTE); G08 02.0; M133 P1500; M08; G00 V0.2; G201 N2,7 X2,3 Z-0.5 F8.; C240; G240; (Canis C40, C40, C40, C40, C40, C40, C40, C40,			To Switch Boxes [F4]
G99 M135; G00 G53 V0.; G53 X0.; G53 Z0.;	Back Forward Search (TEXT) [F1 Current Directory: VPS/), or [F1]	Last Modified
(END RADIAL DRILLING CYCLE);	Live Tooling	<dir></dir>	03/01/19 09:44 >
M01;	Face Turn	35949	03/01/19 09:44
	OD Turn	54709	03/01/19 09:44
Main Spindle	ID Turn	54900	03/01/19 09:44
Spindle Speed: 0 BPM	OD Thread	48054	03/01/19 09:44
Spindle Power: 0.0 KW	ID Thread	47329	03/01/19 09:44
Surface Speed: 0 EPM	Drilling	28345	03/01/19 09:44
Overrides Chip Load: 0 00000 IPT	OD Groove	109935	03/01/19 09:44
Chip Load: 0.00000 IPI	ID Groove	110114	03/01/19 09:44
Feed: 100% Feed Rate: 0.0000 IPR	OD Radius	54998	03/01/19 09:44
Spindle: 100% Active Feed: 0.0000 IPR	ID Radius	55027	03/01/19 09:44
Rapid: 100%	OD Chamfer	67503	03/01/19 09:44
Spindlo Load(%)	ID Chamfer	67668	03/01/19 09:44
Setup Door OpenPower Save			

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



urrent Directory	Forward	Search (TE	EXT) [F1], or	(F1) to c	lear.		
unent birectory	File Name			Size	Last	Modified	
PROBING	The Hume			<dir></dir>	08/18/17	19:12	>
/PS				<dir></dir>	08/18/17	19:12	>
CUSTOM				<dir></dir>	08/18/17	19:13	>
				_			

- 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.



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

Edit: MDI		07:41:27		Program	Generation
MDI		N451	Editor VPS	Shape Creator	
IRadial Drilling): IRadial Drilling): (SAFETY UNE BELOW G00 G54 G18 G40 G G00 G54 G18 G40 G (TOOL = 5 / OFFSET = 54, IORILL RPM = 1500); (C-AXIS START ANCIE (# OF EQUALLY SPAC T505; (G40 G20, G20); G96 (FEED PER MINL G00 C0, X2.9; G00 C0, X2.9; G00 V0.; (G241 R2.7 X2.3 Z-0); G241 R2.7 X2.3 Z-0; (C240); G80; (M05;	(); 80 G97 G99; = 5); b; E = 0.0}; ED HOLES = 3); JTE); 5 F8.;		FACING CYCLE -		Run in MDI [CYCLE START] Generate Goode [F4] Reset [ORIGIN] Devices [F2] Jog Axis+[HAND JOG]
G99 M135; G00 G53 Y0.; G53 X0.; G53 Z0.; (END RADIAL DRILLIN M01;	IG CYCLE); Main Spindle		Back Variable FEEDRATE_ROUGH FINISH_PASS Z_FINISH_STOCK BETRACT X_HOME	0.01 72 0.003	Ranges [0.001 - 0.03] 70 72 [0.00 - 0.05] VN
Overrides Feed: 100% Spindle: 100% Rapid: 100%	Spindle Speed: Spindle Power: Surface Speed: Chip Load: Feed Rate: Active Feed:	0 RPM 0.0 KW 0 FPM 0.00000 IPT 0.0000 IPR 0.0000 IPR	RETRACT_Z_HOME END_M_CODE	, У З0	Y N 0 1 30
Spindle Load(%)	-	0%	No FINISH PASS v	vill be executed. To execu	te a finish pass enter (70)
			·		
Setup Door OpenPow	er Save				
Input:					

- 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.



• The program is then output to MDI.

Edit: MDI		15:47:23	MDI	
MDI		N0		
[Circle Contour Milling (EXTERNAL CONTOUR (TOOL 1 / DIA 5 IN.); (SPINDLE 1500 RPM/ (R PLANE 0.1 / DEPTH 600 G17 G40 G49 G8 G10 L12 P1 Ro.5 (THIS G00 G90 G54 X3.25 Y S1500 M03; G43 H01 Z0.1; M08; G01 Z-0.2 F5; G01	MILLING); FEED 10. IPM); -0.2); G G90; LINE SETS D1 OFFSE 2.; L F10.;	D;	Active Codes Active Tool G00 Rapid Motion Tool: 1 Offset: 0 G90 Absolute Position Tool: 1 Offset: 0 G80 Cycle Cancel Tool Group: Max Load: 0 G80 Cycle Cancel Uffs: 1000 Max Load: 0 G54 Work Offset #54 Uffs: 1000 Next Tool D00 H00 M00 T0 Pocket: 1	Coolant off 1/1 0/1
602 10.5; 603 599 X3, Y-2.5 10.5 601 G40 X3,25 Y-2; 601 20,110. M09; 600 G90 G53 20 M05 M01 (END VPS CIRCLE	;; contour); Spindle		Positions Program G54 G49 Timers	And Counters
Main Spindle STOP Overrides Feed: 100% Rapid: 100% Rapid: 100%	Spindle Speed: Spindle Load: Surface Speed: Chip Load: Feed Rate: Active Feed:	0 RPM 0.0 KW 0 FPM 0.00000 0.0000 0.0000	(IN) Load This Cycle: (IN) Load Children Cycle: (IN) Load Children Cycle: (IN) Load Children Cycle: (IN)	0:00:00 0:00:00 #1: (#2: (ning: (
Spindle Load(%)	Save	0%		_

- 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.

Edit: MDI		07:48:00		rogram Generation		
MDI		N451	Editor VP Shape Creator			
IFACING CYCLE3 (SAFETY LINE BELOW, (SAFETY LINE BELOW, (TOOL = 5/10F90) (TOOL = 5/10F90) (MAXIMU PSINDLE I (CONSTANT SURFACE) (CONSTANT SURFACE) (INASIMU PSINDLE I (CONSTANT SURFACE) (STOCK TO ARMOVE (DEPTH PER PASS = T505; (S54) (S54) (S55) (G00 20.23); (G00 20.20); (G00 20.20); (G00 20.20); (G00 20.20);); 80 G97 G99; } 51; FMA = 1800); 5 SPEED = 500); 5 SPEED = 500); 2 .5); 0.001); 0.001;		OD ROUGH AND	FINISH CYCLES	To Switch Boxes (Load (ENT	F4] ER]
N99 G00 Z0.; G01 X0.; N100 G00 Z0.03; G00 G80 Z0.23; G97 S720; M09; G00 G53 X0 -			Back Forward Current Directory: VPS/ File Name Part Off OD Profile Removal Cycles	Search (TEXT) [F1], or [F1] Size 56334 36710 40416	to clear. Last Modified 03/01/19 09:44 03/01/19 09:44 03/01/29 09:44	
STOP Overrides Feed: 100% Spindle: 100% Rapid: 100%	Main Spindle Spindle Speed: 0 Spindle Power: 0.0 Surface Speed: 0 Chip Load: 0.000 Feed Rate: 0.000 Active Feed: 0.000	RPM KW FPM 000 IPT 00 IPR 00 IPR	OD Thread Repair ID Thread Repair Center Dril Tapping	38511 41250 23501 35614	03/01/19 09:44 03/01/19 09:44 03/01/19 09:44 03/01/19 09:44	
Spindle Load(%)		0%				
Setup Door OpenPow	er Save		1	- T		
mpor. I						

- 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.

Edit: MDI	07:43:40	Program Generation	
MDI	N451	Editor VPS Shape Creator	
IFACING CYCLE): (GAFETY LINE BELOW); (G40 ESA LG 40 G80 G91; (G40 ESA LG 40 G80 G91; (WORK OFFSET = 54); (WORK OFFSET = 54); (MAMAUM SPINDLE PPM = (CONTSIDE DIAMETER 0.2); (FINSH DIAMETER 0.2); (STOCK TO REMOVE = 0.3); (STOCK = 0.2); (STOCK = 0.2);	7 G99; 1800); D = 500);); W0.003 F0.01;	File Name: (Not Saved) Y Ray Dimensions X Position 2.5000 Z Position 0.0000 Y Rapid Point 2.5000 X Position 2.6000 Z Position 0.0000 Z Position 0.0000 Z Position 0.0000 Z Position 0.0000 Z Position 0.1000 ENTERI Add Row 0.0000 Insert Row 1: Uncar Feed Motion 1: CCW Circular Feed Motion 3: CCW Circular Feed Motion 3: CCW Circular Feed Motion 4: Cancel Exit [CANCEL] 0.0000	
STOP Sr Sr Overrides Feed: 100% Spindle: 100% Spindle: 100% Rapid: 100% Spindle: 100%	n Spindle vindle Speed: 0 RPM vindle Power: 0.0 KW triace Speed: 0 FPM Chip Load: 0.00000 IPT Feed Rate: 0.0000 IPR Active Feed: 0.0000 IPR	Jog Rate: .1 F4 Save File Immer Set Value F2 Enable Zoom immer Add Motion F3 Open the calculator. Atter Motion Type once New shape; discard changes Extre Delete Motion	
Spindle Load(%)	0%	Press left arrow to collapse row.	or right arrow to expand.
Setup Door OpenPower Save		T T	

• Linear and circular moves can be input by using "insert". Each move requires a new line.

			Program Generation
	VPS	Shape Creator	
File Nom	. (Not Co	(od)	_
V Pay D	imoneione	veu)	
V NAW D	Position	2 5000	
2	Position	0.0000	
V Panid	Point	0.0000	
* napro	Position	2,6000	
2	Position	0.1000	
V Start	1031(10)	0.1000	
V Sturt	Position	1 6666	
7	Position	0.1000	
V 1 · 1 i	near Feed	0.1000	
• 1. L1	Position	1.0000	
7	Position	-1.0000	P
4	nale	180,000	
0	hamfer	0.000	
6	lound	0.0000	
LENTE	B) Add Boy		
(Child	ang Add 100		



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.

		Program Generation	
Editor VPS	Shape Creator		
File Name: (Not Sav	(ed)		
X Position	1.0000		
Z Position	-1.0000	i	
Angle	180.000		
Chamfer	0.0000		
Round	0.0000		
V 2: Linear Feed			
X Position	1.2500		
Z Position	-1.0000		
Angle	90.000		
Chamfer	0.0000		
Round	0.0000		
V 3: Linear Feed		[€]	
X Position	1.2500		
Z Position	-1.5000		
Angle	180.000		
Dound	0.0000		
V 4. CM Arc	0.0000		
V 4: UN AIC	1 7500		
7 Position	-1.7500		
Radius	0.2500		
Chamfer	0,0000		
Round	0.0000		
[ENTER] Add Roy	N		
log Rate: .1		F4 Save File	
ENTER Set Value		F2 Enable Zoom	
Add Motion		F3 Open the calculator.	
ALTER Alter Motion	n Type	ORIGIN New shape; discard changes	
Delete Mot	ion		
_		Press left arrow to collapse row, or right arrow t	o expand.
		The second se	

- 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.

Edit: MDI	07:47:39	Program Generation
MDI	N451	Editor VPS Shape Creator
[FACING CYCLE]: (SAFETY LINE BELOW): G00 G54 G18 G40 G80 G9 (TOOL = 5 / 0FFSET = 5): (WORK 0FFSET = 54): (WORK 0FFSET = 54): (WORK 0FSET = 54): (MAXIMUM SPINDLE RPM = (OUTSIDE DIAMETER: 0): (STOCK TO REMOVE = .03) (DEPTH PER ROSK = 0.001 T505): 75.01	17 G99; 1800); ED = 500);); L); Save As	File Name: 1.scp V 2: Linear Feed X Position 1.2500 Z Position -1.0000 Angle 99.000 Chanfer 0.0000 V 3: Linear Feed X Position 1.2500 -2 P
034; 650 51800; 657 5720 M03; 600 20.23; 600 20.23; 600 20.03; 600 20.03; 602 20.03; 602 20.03; 602 20.03; 601 20.03; 601 20.03; 600 20.03; 600 20.03; 600 680 20.23; 607 5720; M09; 609 58 20; 609 58 20; 609 58 20; 600 58 20;	. W0.003 F0.0	
Mair	n Spindle ENTER Save	ATTRA Select File Onom Clear Input
STOP S Stop St	pindle Speed pindle Power urface Speed: 0 FPM Chip Load: 0.00000 IPT	Add Motion F3 Open the calculator.
Feed: 100% Spindle: 100% Rapid: 100%	Active Feed: 0.0000 IPR	Anter Motion Type Cenan New shape: discard changes
Spindle Load(%)	0%	Press left arrow to collapse row, or right arrow to expand.
Setup Door OpenPower Save	2	
input		

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

MDI N451 (00) Profile Removal Cycles): (SAFETY LINE BELOW): (SAFETY LINE BED):	Edit: MDI		07:	49:38		Program Generat	
0D Profile Removal Cycles; 00 Profile Removal Cycles; 194 G54 168 DELOV; 00 Forfile Removal Cycles; 195 G54; 00 Forfile Removal Cycles; 195 G54; 00 STOCK REMOVAL CYCLE; 195 G54; 00 STOCK REMOVAL CYCLE; 196 T1 L42 014; 00 STOCK REMOVAL CYCLE; 197 OCK REMOVAL CYCLE; <td< td=""><td>MDI</td><td></td><td>1</td><td>V451</td><td>Editor VPS Shape</td><td>Creator</td><td></td></td<>	MDI		1	V451	Editor VPS Shape	Creator	
05-47; 059 (1800); 059 (1800); 050	(OD Profile Removal C (SAFETY LINE BELOW GO0 G54 G18 G40 G8 (TOOL = 5 / OFFSET = (MAXIMUM SPINDLE R (SURFACE SPEED SPEE (DEPTH PER PASS =) (WORK OFFSET = 54) (OD STOCK REMOVAL) T505:	ycles);); (0 G97 G99; =5); (PM = 1800); EED = 500); (ED = 500); (0.05); (CYCLE);			OD Profile Removal Cycle	71) IRREGULAR PATH CYCL	Run in MDI (CYCLE STAR Generate Gcode (F Reset (DRIGI Devices (F Jog Axis+(HAND JO
Main Spindle Main Spindle Back Control of the speed:	G50 51800; G96 5500 M03; G90 20.1; M08; G90 22.6 20.1; G71 P144 Q145 D0.0; N144 G42 X1. Z0.1; G01 X1. Z0.1; G01 X1. Z0.1;	5 U0.01 W0.003 F0.	.01;		Finish tool path	(0)	
Variable Variable Value Ranges N145 G01 240 X2 6 Z - 1.75; G00 X2 7 Z - 1.	G01 X1.25 Z-1.; G01 X1.25 Z-1.5;	25.			Back		
NL45 G01 G40 X2.6 2-1.75; G00 X2.6 2.0; M09; G01 X2.6 2.0; M09; SHAPE 1.scp [ENTER] to select file M09; S01 G53 X0 TOOL_NUMBER 5 [1 - 94] M09; S01 G53 X0 Spindle Speed: 0 RPM 1800 [54 - 59] Stopp Overrides Spindle Speed: 0 RPM 1800 [1 - 3400] Spindle Speed: 0 FPM Sufface Speed: 0 FPM Spindle: 100%, Spindle: Spindle Speed: 0 FPM NAX, RPM 1800 [1 - 3400] Sufface Speed: 0 FPM Sufface Speed: 0 FPM Sufface Speed: 0 FPM Spindle: 100%, Spindle: Stork El/D value Create 2.5000 10 9 Feed: 100%, Spindle: 0.0000 IPR Active Feed: 0.0000 IPR 40 42 40 42 Spindle Load(%) 0% 0% 10 0.05 10.0-0.15)	G01 X2.5 Z-1.75 R0.	20;			Variable	Value	Ranges
Mog; Gao 653 yo. TooL_NUMBER 5 [1 - 24] Main Spindle Spindle Speed: 0 RPM 1480 [54 - 59] Stopp Overrides Spindle Speed: 0 RPM 1800 [1 - 3400] Spindle Speed: 0 RPM 1800 [1 - 3400] Surface Speed: 0 FPM Spindle Speed: 0 Spindle: 100% Spindle Speed: 0 FPM Spindle: Spindle Speed: 0 FPM 1800 [1 - 3400] Surface Speed: 0 FPM Spindle Speed: 0 FPM 1800 [50 - 2000] Spindle: Spindle: Spindle: 0.0000 FP FCOD_COLANT 8 9 Stock_REMOXALCYCLE 71 0 71.73 0 70.13 100.00.01.51 Feed: 10.00% FP 40.42 40.42 00 10.00.01.51 100.00.01.51 100.00.01.51 100.00.01.51 100.00.01.51 100.00.01.51 100.00.01.51 100.00.01.	N145 G01 G40 X2.6 Z G00 X2.6 Z0.1;	1.75;			SHAPE	1.scp	[ENTER] to select file
Stop Spindle Speed: 0 RPM SurRACE_SPEED_MINUTE Sol [1 - 99] Overrides Spindle Power: 0.00000 RPM Surface Speed: 0 RPM 1800 [1 - 3400] Surface Speed: 0 RPM Surface Speed: 0 FPM 1800 [1 - 3400] Spindle Power: 0.00000 RP Surface Speed: 0.00000 PFM Surface Speed: 0.00000 PFM Chip Load: 0.00000 IPR Active Feed: 0.0000 IPR 500CK_REMOVAL_CYCLE 71 0 0 17.3 Spindle Load(%) 0% 0% 10 0.05 (0.00.01) 10.05 10.00 10.05 10.00 10.05 10.00 10.05 10.00 10.05 10.00 10.05 10.00 10.05 10.00 10.05 10.00 10.05 10.00 10.05 10.00 10.05 10.05 10.05 10.05 10.05 10.05 10.05 10.05 10.05 10.05 <	M09;				TOOL_NUMBER	5	[1 - 24]
Main Spindle Work, OFFSET 54 [54 - 59] Stop Spindle Speed: 0 RPM 1800 [1 - 3400] Surface Speed: 0 FRM Spindle Power: 0.0 KW Spindle Speed: [56 - 2000] Overrides Spindle Speed: 0 FFM 1000 [50 - 2000] Spindle Speed: Spindle Speed: 0 Spindle Speed: Spindle Spindle Spi	G00 G53 X0 ·				TOOL_OFFSET_NUMBER	5	[1 - 99]
STOP Spindle Speed: 0 RPM MAX_RPM 1800 [1 - 3400] Overrides Spindle Power: 0.0 KW Surface Speed: 0 F800 [50 - 2000] Overrides Spindle Speed: 0 FFW Spindle Speed: 0 F800 [50 - 2000] Feed: 100% Feed: 0.0000 IPR STOCK_DIAMETER 2.5000 9 Spindle: 100% Active Feed: 0.0000 IPR STOCK_REMOVAL CYCLE 71 0 0 71.73 Rapid: 100% Peerstreets 0.05 [0.0 - 0.15] [0.0 - 0.15] 100.0 - 0.15] Enter STOCK REMOVAL CYCLE Files STOCK REMOVAL CYCLE 71 0 0.5] [0.0 - 0.15]		Main Spindle			WORK_OFFSET	54	[54 - 59]
Store Spindle Power: 0.0 KW SURFACE_SPEED_MINUTE 500 [50 - 2000] Overrides Chip Load: 0.0000 IPT Feed Rate: 0.0000 IPT 8 8 9 Feed: 100% Feed Rate: 0.0000 IPT Forck_IMAFTER 2.5000 71 3 Spindle: 100% Active Feed: 0.0000 IPT Forck_REMOVAL CYCLE 71 0.71.73 Spindle: 100% Off Forck_REMOVAL CYCLE 71 0.71.73 Spindle: Load(%) 0% Force Stork_REMOVAL CYCLE 71 0.0.0.5)		Spindle Speed:	0 F	RPM	MAX_RPM	1800	[1 - 3400]
Surface Speed: 0 FPM FLOD_COOLANT 8 8.9 Feed: 100% Chip load: 0.0000 IPT Feed Rate: 0.0000 IPT STOCK_DIAMETER 2.5000 0.71.73 Rapid: 100% Active Feed: 0.0000 IPT TOCK_REMOVAL_CYCLE 71 0.71.73 Spindle: 100% Active Feed: 0.0000 IPT 40.42 40.42 Inclusion: Doc 0.05 [0.0 - 0.15] Inter STOCK REMOVAL CYCLE: Filer STOCK REMOVAL CYCLE: 10.00	STOP	Spindle Power:	0.0 k	KW	SURFACE_SPEED_MINUTE	500	[50 - 2000]
Spindle: 100% Feed: 0.0000 IPR STOCK_DIAMETER 2.5000 Spindle: 100% Feed: 0.0000 IPR STOCK_REMOVAL_CYCLE 71 0 71.73 Spindle: 100% Feed: 0.0000 IPR TOOL_NOSE_COMP 42 40.42 DoC 0.05 [0.0 - 0.15] [0.0 - 0.15] 100 (0.0 - 0.15) Pinde Load(%) Feet STOCK REMOVAL_CYCLE: Finish (0). OB Roudh (71). Tregular Path Rough (73) 100 (0.0 - 0.15)		Surface Speed:	0 F	FPM	FLOOD_COOLANT	8	8 9
Feed value 0.0000 IPR Spindle: 100% Active Feed 0.0000 IPR Rapid: 100% Conception 40 42 pondle: 0.05 (0.0-0.15) prindle: Doception 0.05 (0.0-0.15) prindle: Doception Doception 1000	Overrides	Chip Load:	0.00000 1	PI	STOCK_DIAMETER	2.5000	
Spinale: Lutw TooL_NOSE_COMP 42 40 42 maple: 100% 00% [0.0 - 0.15] [0.0 - 0.15] pindle Load(%) 0% Inter STOCK REMOVAL CYCLE: Finish (0), 0D Rough (72), irregular Path Rough (73)	Feed: 100%	Active Feed	0.0000 1	PR	STOCK_REMOVAL_CYCLE	71	0 71 73
DOC 0.05 [0.0 - 0.15] [0.0 - 0	Spindle: 100%	, lotive reed.	0.0000 1		TOOL NOSE COMP	42	* 40 42
Spindle Load(%) 0% Enter STOCK REMOVAL CYCLE: Finish (0), OD Rough (71), Irregular Path Rough (73)	Napid. 100%				DOC	0.05	[0.0 - 0.15]
	Spindle Load(%)			0%	Enter STOCK REMOVAL CY	CLE: Finish (0), OD Rough	(71), Irregular Path Rough (73)
(m)	Setup Door OpenPower	Save					

- 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.

Edit: MDI		07:48:16		Program C	Generation	
MDI		N451	Editor VPS Shape	Creator		
(FACING CYCLE); (SAFETY LINE BELOW); 600 G54 G18 G40 G80 (TOOL = 5 / OFFSET = 5 (WORK OFFSET = 54); (MAXIMUN SPINDLE FRP (CONSTANT SUPFACE 5 (OUTSIDE DIAMETER: 0.); (STOCK TO REMOVE = .) (DETTH PER PASS = 0.)	G97 G99; 5); 4 = 1800); PEED = 500); 5); 03); 061);		OD Profile Removal Cyc	les		kun in MDI (CYCLE START) Senerate Gcode (F4) Keset (ORIGIN) Devices (F2) Dg Avis+(HAND JOG)
T505; 654:	Select File					
G50 51800; G97 5720 M03; G00 20,23; G00 20,23; G00 20,29; M08; G96 5500; G00 20,33; G72 P59 Q100 D0.001 N99 G60 20,3; G00 20,03; G00 20,03; G00 20,3; G00 20,23; G97 5720; M09; G00 G53 X0 -	Insert Directory: 1.scp				L	So ROUGH AND FINISH CYCLES
			Select [ENTER] Exit	[CANCEL]		
Coverrides Feed: 100% Rapid: 100%	Spindle Power: 0 Surface Speed: 0 Chip Load: 0 Feed Rate: 0 Active Feed: 0	0 KW FPM 000000 IPT 00000 IPR 00000 IPR	SURFACE_SPEED_MINUTE FLOOD_COOLANT STOCK_DIAMETER STOCK_REMOVAL_CYCLE TOOL_NOSE_COMP	500 8 71 40	[50 - 24 8 9 0 71 73 40 42	1001
Spindle Load(%)		0%	DOC	0.05	[0.0 - 0	.15]
Setup Door OpenPower S) Gave		PRESS ENTER TO SELECT	your predefined Sha	аре ние	_

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



- 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.



• The program is then output to MDI.

N451 91;	G00 G99 G40 G80 G54 Spindle D2 Spindle D2 Spindle D2 Spindle D0 Direction:	Active C Rapid Motion Feed Per Revoluti Cancel Tool Nose Cycle Cancel Work Offset #54 D00 H00 H00 H00 H00 H00 H00 H00 H00 H00 H	odes on Compensation M00 T0 Spint Main St 0,0,0 0,0,0 0,0,0 0,0,0 0,0,0 0,0,0 0,0,0 0,0,0 0,0,0 0,0,0 0,0,0 0,0,0 1,00000000	A Tool: 5 Type: Tool Groo Max Load Life: 1009 diles aindle 0 6 6 9 9 6 9 9 6 9 9 6 9 9 6	ctive Tool Offset: 5 None pp: : 0	Coolant Off
91;	G00 G99 G40 G80 G54 Spindle C Spindle C Spindle D Spindle D Spindle D Spindle D D Spindle D Spindle Spindle D Spindle Spindle Spindle D Spindle Spindle Spindle Spindle D Spindle Spindle Spindle D Spindle Spindle Spindle Spindle D Spindle Spindle Spindle Spindle D Spindle Spindle Spindl	Rapid Motion Feed Per Revoluti Cancel Tool Nose Cycle Cancel Work Offset #54 D00 H00 H00 <t< th=""><th>on Compensation M00 T0 Spin Main Si 0,0,0 0,0,0 0,0,0 0,0,0 0,0,0 0,0,0 0,0,0 0,0,0 0,0,0 0,0,0 0,0,0 0,0,0 0,0,0 1,000 1,00000000</th><th>Tool: 5 Type: Tool Grou. Max Load Life: 1009 dles sindle 30 6 6 90 6 90 90 90 90 90 90 90 90 90 90</th><th>Offset: 5 None pp: : 0 : 0 : 0 : 0 : 0 : 0 : 0 : 0 : 0 : 0</th><th></th></t<>	on Compensation M00 T0 Spin Main Si 0,0,0 0,0,0 0,0,0 0,0,0 0,0,0 0,0,0 0,0,0 0,0,0 0,0,0 0,0,0 0,0,0 0,0,0 0,0,0 1,000 1,00000000	Tool: 5 Type: Tool Grou. Max Load Life: 1009 dles sindle 30 6 6 90 6 90 90 90 90 90 90 90 90 90 90	Offset: 5 None pp: : 0 : 0 : 0 : 0 : 0 : 0 : 0 : 0 : 0 : 0	
91;	Spindle Sp Spindle Po Spindle Lo Surface Sp Chip Load: Spindle Ov Direction:	beed: (RPM) ower: (KW) ad: (%) peed: (FPM) : verride:	Spin Main Sr 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	dles pindle 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	Live Tor 0 0,0000 0,0000 0,0000 100% Stop	ام م
91;	Spindle Sp Spindle Pc Spindle Lo Surface Sy Chip Load: Spindle Ov Direction:	beed: (RPM) ower: (KW) bad: (%) peed: (FPM) : verride:	Main Sr 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	bindle 5 6 000 900 9%	Live Too 0 0.0 0.0000 0.00000 0.00000 1.00% Stop)))
	Position	ns Pro	oram G54 T505		Timers And C	ounters
0 RPM 0.0 KW 0 FPM 0.00000 IPT 0.0000 IPR 0.0000 IPR	io x io y io z B	(IN) 0.0000 0.0000 0.0000		Load 0% 0% 0%	This Cycle: Last Cycle: Remaining M30 Counter #1: M30 Counter #2:	0:00:05 0:00:05 0:00:00 92 92
0%	c		_	0%	Loops Remaining:	0
	0 RPM 0.0 KW 0 FPM 0.00000 IPT 0.0000 IPR 0.0000 IPR	0 RPM 0 RPM 0 FPM 0 0 FPM 0.0000 IPR 0.0000 IPR 0.0000 IPR 0.0000 IPR 0.0000 C	PFM (IN) 0.0 RPM (IN) 0.0 KW Image: Constraint of the second secon	Program CS4 Program CS4	O RPM (IN) Load 0.0 RPM 0.0000 0% 0.0000 IPR 0 7 0.0000 0% 0.0000 IPR 0 Z 0.0000 0% 0.0000 IPR 0 0 0% 0%	D RPM (IN) Load Thirty-said C 0.0 RPM (IN) Load This Cycle: 0.00000 IPR Y 0.0000 0% Remaining 0.00000 IPR Y 0.0000 0% M30 Counter #1: 0.0000 IPR 0.2 0% 0% Counter #2: 0.0000 0% C 0% C 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.
- 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.

Start-up	% 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
0.111	(BEGIN TOOL BODY)
Cutting	
	(END TOOL BODY)
ut down	G00 G80 G99 Z0.1 M09 G97 S1000 G53 X0 G53 Z0 M01
	%

Sh

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 \$1,000 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
I	G00 G53 X0
I	G53 Z0
I	T0101
I	G97 S1000 M03
I	G40 G54 G80 G99 Z0.1 M08
I	X2.
I	G50 S6000
I	G96 S400
I	
l	
l	
l	
l	(END TOOL BODY)
l	
l	G00 G80 G99 Z0.1 M09
	G97 S1000
	G53 X0
	G53 Z0
	M01
I	%

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
 - G8o Cancel canned cycle
 - G99 Sets feed to IPR
 - Zo.1 Approach part in Z
 - Mo8 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 Z0
	M01
	%

Tool Path Startup

• T6o6

- 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
1	T0101
	G97 S1000 M03
	G40 G54 G80 G99 Z0.1 M08
	X2.
1	G50 S6000
	G96 S400
	(BEGIN TOOL BODY)
	(END TOOL BODY)
	C00 C80 C99 70 1 M09
	G00 G80 G99 20.1 M09
1	G53 X0
	653 70
	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.
 - Goo G8o G99 Z.1 Mo9
 - 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 Xo.
 - G53 sends the axis home
 - Xo. indicates the axis to send home
 - Note that X is homed first to avoid possible collision
 - G53 Zo.
 - G53 sends the axis home
 - Zo. indicates the axis to send home
 - Мо1
 - 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) 600 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 Go1 straight line movements there are:
- Go2 Curve clockwise
- Go3 Curve counterclockwise

- Go1, Go2, and Go3 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:
 - Go2 X1.0 Z-.625 R.125
 - Go3 X1.25 Z-.75 R.125
 - G01 Z-1.057
- Notice that after a circular move there has to be a Go1 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.00
N10 X0.
G01 Z0.
X.5
G03 X.75 Z125 R.125
G01 Z5
G02 Z625 X1.0 R.125
G03 Z75 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 ZØ. X1.6 X2. Z-0.2 Z-3.8 X2.4 Z-4. X2.6 G03 X3. Z-4.2 R0.2 GØ1 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 XØ. G53 ZØ. 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.oo3 - 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.



000017 (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 (3.35 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 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.

000017 (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 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. V200 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.o2 - 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.o3 - 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.o1 - 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.o6 X-axis distance and direction from first cut to last, radius
 - K.o6 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 GØ1 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 ZØ. 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



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.



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 l.1 K.125 F2. X1.375 – End X position Z-.5 – End Z position l.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.o4 Thread height, defines thread depth, radius measure
 - A6o- 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 ZØ. M01 N2 (TOOL AT SAFE INDEX POSITION) G00 G53 X0. G53 ZØ. 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 ZØ. 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.
 - *P*1:Single edge cutting, cutting amount constant
 - P2:Double edge cutting, cutting amount constant
 - P3: Single edge cutting, cutting depth constant
 - *P*4: 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 %



- 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.

- 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.





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

Graphics					
Tool: [F2]: E [ENTER	Work C Apply Zoom	554: + Drill Po [HOME]: Reset Zoom [END]: Center Zoom	INT: 8 [PAGE UP]: 2 [PAGE DOWN]	Zoom -	
Positions	Р	rogram G54 T1 00		Timers And Co	unters
	(IN)		Load	This Cycle:	0:00:05
🕞 X	0.0000		0%	Last Cycle:	0:00:05
A 7	0 0000		0%	Remaining M30 Counter #1	0:00:00
<u> </u>	0.0000		070	M30 Counter #2:	22
				Loops Remaining:	0
		Opt Stop			

- 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.

Operation: M	1EM	07:36
MEM	Memory/Boring Job.txt	N
X2.1 Z0.1 M08 / G71 P10 Q20 N10 X-0.04; G01 Z0.; X1.875 K-0.06 Z-1.65; N20 X2.1; G00 Z0.1; G53 X0. M09;	};) U0.02 W0.003 D0.04 F0.01 ;	