



Basic Lathe Training

Training course

Table Of Contents

Pages

Types of HAAS Lathes	3-5
Types of Turrets	6-9
Pendant Overview	10-11
Start-up Procedure	11
Pendant Display	12-13
Numerical/ Alphabetical Keys	14
Override Keys	15
Jog Keys	16
Mode Keys	17
List Programs	17-19
Zero Return	20
Handle Jog	20
MDI	21
Memory	22
Edit	23-27
Display	28
Program	29
Position	30-33
Offset	34-37
Current Commands	38-43
Alarms	44-46
Diagnostic	47-48
Setting	49-51
Help	52
Function Keys	53-54
Types of Tooling	55-57
Programming	58-59
G-Codes	60
M Codes	60
Absolute vs. Incremental	61
Visual Programming System (VPS)	62
Tool Setting	63-67
Shape Creator	68-69
VPS	70-74

TYPES OF LATHES

ST SERIES



- ST-10 → 55
- ST-40 → 45L
- ST-10 → 35Y

The Haas ST Series high-performance turning centers were designed from the ground up to provide setup flexibility, extreme rigidity, and high thermal stability.

TL SERIES



- TL-1
- TL-1-EDU
- TL-2

The Haas TL Series Toolroom Lathes are affordable, easy to use, and offer the precision control and flexibility of the Haas CNC system. Because they are very easy to learn and operate – even without knowing G-code – they are perfect for start-up shops, or as a first step into CNC machining.

DS SERIES



- DS-30Y

The DS-30Y Y-axis turning center combines dual-spindle turning with Y axis, C axis, and live tooling to create a powerful "done-in-one" machining solution for any shop. Off-center milling, drilling, and tapping operations are possible for increased machining capabilities.

CL SERIES



- CL-1

The innovative Haas CL-1 is an ultra-compact CNC chucker lathe that comes standard with an 8-station automatic turret, making it perfect for high-volume production of small, precision parts, such as those found in the communications, aerospace, medical, and dental industries. The CL-1 can be moved easily with a pallet jack or equipment dolly, and is small enough to fit into most freight elevators.

TYPES OF TURRETS



BOLT-ON TURRET

THE HAAS 12-STATION BOLT-ON TOOL TURRET PROVIDES SUPER-RIGID MOUNTING OF TURNING TOOLS AND BORING BARS. THE TURRET ACCEPTS INDUSTRY-STANDARD BOLT-ON TOOLHOLDERS AROUND THE PERIMETER, AND HAS RADIAL SLOTS ON THE FACE FOR MOUNTING TURNING TOOLS IN EITHER THE RIGHT- OR LEFT-HAND DIRECTION.



VDI TURRET

THE HAAS 12-STATION VDI TURRET ACCEPTS INDUSTRY-STANDARD LIVE-TOOLING HEADS (WHEN COMBINED WITH THE LIVE TOOLING OPTION) AND STATIONARY TOOLHOLDERS WITH VDI40 CONNECTIONS. A SINGLE LOCKING WEDGE SECURES THE TOOLHOLDERS IN THE TURRET, ALLOWING QUICK, EASY SETUPS.



HYBRID TURRET

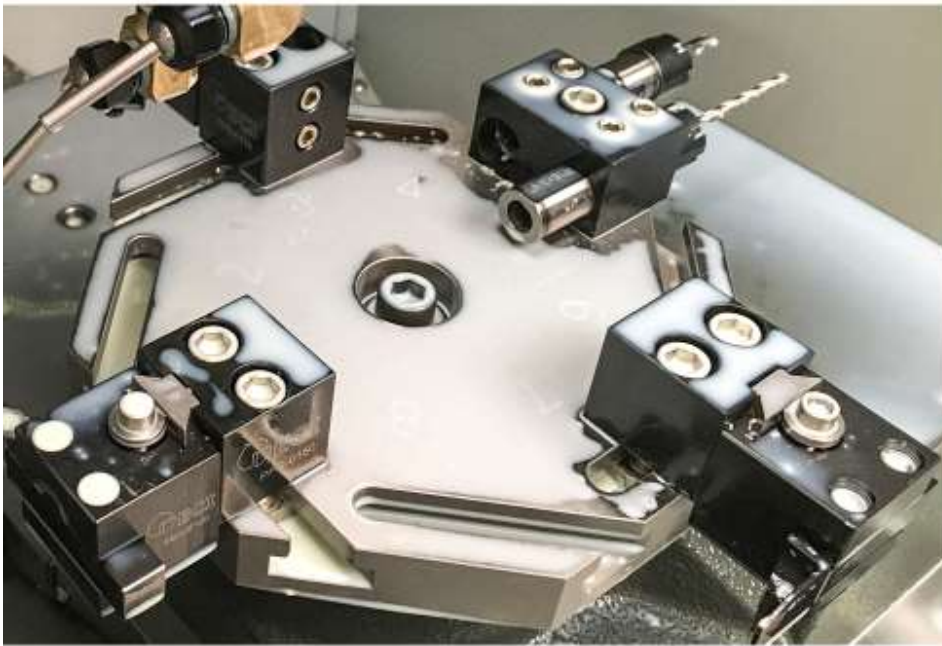
THE HAAS 12-STATION HYBRID TURRET ACCEPTS VDI40 TOOLS (INCLUDING LIVE TOOLS), AS WELL AS INDUSTRY-STANDARD BOLT-ON TOOLS. THE TURRET HAS 6 VDI STATIONS AND 6 BOLT-ON STATIONS (RADIAL SLOTS FOR TURNING TOOLS AND PERIMETER STATIONS FOR ID TOOLS).



BMT65

THE HAAS 12-STATION BMT65 TURRET PROVIDES EXTRA-RIGID MOUNTING FOR TURNING AND BORING TOOLS TO IMPROVE CUTTING PERFORMANCE, AND OFFERS ADDITIONAL TOOL CLEARANCE WHEN WORKING WITH A TAILSTOCK. INDUSTRY-STANDARD TOOLHOLDERS INSTALL EASILY, WITHOUT ADDITIONAL ALIGNMENT, TO REDUCE SETUP TIMES. THE TURRET IS AVAILABLE IN BOTH STATIC AND DRIVEN-TOOL VERSIONS.

THE BMT65 TURRET WITH LIVE TOOLING ALLOWS SECONDARY OPERATIONS – MILLING, DRILLING, SLOTTING, AND TAPPING – ON THE FACE OF THE PART AND AROUND THE DIAMETER. TO BOOST PRODUCTIVITY EVEN MORE, A HALF-INDEX OPTION IS AVAILABLE FOR BOTH VERSIONS THAT DOUBLES YOUR AVAILABLE TOOL STATIONS TO 24.



8 STATION AUTOMATIC TURRET

4 OR 8 STATION AUTOMATIC TOOL TURRET.
THIS TURRET ACCOMMODATES GANG-STYLE
TOOLING WITH 1/2".

PENDANT OVERVIEW

PENDANT OVERVIEW

- STARTING YOUR MACHINE
- NUMERICAL/ ALPHABETICAL NUMBER KEYS
- OVERRIDES KEYS
- JOG KEYS
- MODE KEYS
- DISPLAY KEYS
- FUNCTION KEYS



STARTING YOUR MACHINE

1. PRESS THE "POWER ON" BUTTON
 2. TWIST THE EMERGENCY STOP NOB TO THE RIGHT
 3. PRESS "RESET" TO CLEAR ALL ACTIVE ALARMS
 4. PRESS "POWER UP/RESTART" TO HOME THE MACHINE
- a) BE SURE THAT NOTHING WILL HIT BEFORE BEGINNING THIS PROCESS



PENDENT SCREEN

ACTIVE MODE AND DISPLAY BAR

ACTIVE PROGRAM DISPLAY

SPINDLE STATUS

ICON BAR

Operation: MEM 13:11:47

MEM Memory/LTI TEST.nc N0

054321 (LTI TEST.OP1X)

(OD TURN CYCLE);
(SAFETY LINE BELOW);
G00 G54 G18 G40 G80 G97 G99;
(ITool = 1 / OFFSET = 1);
(WORK OFFSET = 54);
(MAXIMUM SPINDLE RPM = 1800);
(CONSTANT SURFACE SPEED = 500);
(OUTSIDE DIAMETER = 2.);
(FINISH DIAMETER = 1.);
(FILLET RADIUS = .1);
(DEPTH PER PASS = 0.05);
T101;
O54;
G50 S1800;
G97 S720 M03;
G00 Z0.2;
G00 X2.4 M08;
G96 S500;
G71 P37 Q38 D0.05 U0.01 W0.003 F0.01;
N37 G00 X1.;
G01 Z-0.5 R0.0840;
N38 X2.4;
G00 Z0.2;
G97 S720 M09;
G00 X1.2 (SAFE TC POSITION);
G00 Z36. (SAFE TC POSITION);
M30;

Active Program

Active Codes	Active Tool	Coolant
G00 Rapid Motion	Tool 1: Offset: 1	Off
G99 Feed Per Revolution	Type: None	1/1
G40 Cancel Tool Nose Compensation	Tool Group: -----	0/1
G80 Cycle Cancel	Max Load: 0	
G54 Work Offset #54	Life: 100%	

Spindles

Spindle Speed: (RPM)	Main Spindle
0	0
Spindle Power: (KW)	0.0
Spindle Load: (%)	0%
Surface Speed: (FPM)	500.0000
Chip Load:	0.00000
Spindle Override:	100%
Direction:	Stop

Main Spindle

STOP	Spindle Speed: 0 RPM
	Spindle Power: 0.0 KW
	Surface Speed: 500 FPM
Overrides	Chip Load: 0.00000 PT
Feed: 100%	Feed Rate: 0.0000 PR
Spindle: 100%	Active Feed: 0.0000 PR
Rapid: 100%	
Spindle Load(%)	0%

Positions Program G54 T101

	(IN)	Load
X	1.3100	0%
Z	-0.0732	0%
B	0.0	0%

Timers And Counters

This Cycle:	0:01:02
Last Cycle:	0:01:02
Remaining:	0:00:00
M30 Counter #1:	0
M30 Counter #2:	0
Loops Remaining:	0

Setup Power Save

PENDENT SCREEN

MAIN DISPLAY

ACTIVE CODES

COOLANT LEVEL

TIMERS, COUNTERS AND ADVANCED TOOL MANAGEMENT

ACTIVE ALARMS

Operation: MEM 13:11:47

MEM Memory/LTI TEST.nc N0

054321 (LTI TEST.OP1X)

(OD TURN CYCLE);
(SAFETY LINE BELOW);
G00 G54 G18 G40 G80 G97 G99;
(ITool = 1 / OFFSET = 1);
(WORK OFFSET = 54);
(MAXIMUM SPINDLE RPM = 1800);
(CONSTANT SURFACE SPEED = 500);
(OUTSIDE DIAMETER = 2.);
(FINISH DIAMETER = 1.);
(FILLET RADIUS = .1);
(DEPTH PER PASS = 0.05);
T101;
O54;
G50 S1800;
G97 S720 M03;
G00 Z0.2;
G00 X2.4 M08;
G96 S500;
G71 P37 Q38 D0.05 U0.01 W0.003 F0.01;
N37 G00 X1.;
G01 Z-0.5 R0.0840;
N38 X2.4;
G00 Z0.2;
G97 S720 M09;
G00 X1.2 (SAFE TC POSITION);
G00 Z36. (SAFE TC POSITION);
M30;

Active Program

Active Codes	Active Tool	Coolant
G00 Rapid Motion	Tool 1: Offset: 1	Off
G99 Feed Per Revolution	Type: None	1/1
G40 Cancel Tool Nose Compensation	Tool Group: -----	0/1
G80 Cycle Cancel	Max Load: 0	
G54 Work Offset #54	Life: 100%	

Spindles

Spindle Speed: (RPM)	Main Spindle
0	0
Spindle Power: (KW)	0.0
Spindle Load: (%)	0%
Surface Speed: (FPM)	500.0000
Chip Load:	0.00000
Spindle Override:	100%
Direction:	Stop

Main Spindle

STOP	Spindle Speed: 0 RPM
	Spindle Power: 0.0 KW
	Surface Speed: 500 FPM
Overrides	Chip Load: 0.00000 PT
Feed: 100%	Feed Rate: 0.0000 PR
Spindle: 100%	Active Feed: 0.0000 PR
Rapid: 100%	
Spindle Load(%)	0%

Positions Program G54 T101

	(IN)	Load
X	1.3100	0%
Z	-0.0732	0%
B	0.0	0%

Timers And Counters

This Cycle:	0:01:02
Last Cycle:	0:01:02
Remaining:	0:00:00
M30 Counter #1:	0
M30 Counter #2:	0
Loops Remaining:	0

Setup Power Save

PENDENT SCREEN

ACTIVE TOOL

SPINDLE INFORMATION

POSITION DISPLAY / AXIS LOAD

SYSTEM STATUS BAR

INPUT BAR

The screenshot shows a CNC control interface with the following components:

- Operation:** MEM, 13:11:47
- Active Program:** MEM Memory/LTI TEST.nc N0
- Active Codes:**
 - G00 Rapid Motion
 - G99 Feed Per Revolution
 - G40 Cancel Tool Nose Compensation
 - G80 Cycle Cancel
 - G54 Work Offset #54
- Active Tool:**
 - Tool 1 Offset: 1
 - Type: None
 - Tool Group: -----
 - Max Load: 0
 - Life: 100%
- Coolant:** OFF
- Spindles:**

	Main Spindle
Spindle Speed: (RPM)	0
Spindle Power: (KW)	0.0
Spindle Load: (%)	0%
Surface Speed: (FPM)	500.0000
Chip Load:	0.00000
Spindle Override:	100%
Direction:	Stop
- Main Spindle:**
 - Spindle Speed: 0 RPM
 - Spindle Power: 0.0 KW
 - Surface Speed: 500 FPM
 - Chip Load: 0.00000 FT
 - Feed Rate: 0.0000 PR
 - Active Feed: 0.0000 PR
 - Spindle Load(%): 0%
- Positions:**

	(IN)	Load
X	1.3100	0%
Z	-0.9732	0%
B	0.0	0%
- Timers And Counters:**
 - This Cycle: 0:01:02
 - Last Cycle: 0:01:02
 - Remaining: 0:00:00
 - M30 Counter #1: 0
 - M30 Counter #2: 0
 - Loops Remaining: 0
- Input Bar:** M0

NUMERICAL / ALPHABETICAL KEYS

- ALPHABETIC KEYS
 - THE SHIFT KEY ALLOWS YOU TO ACCESS BUTTONS WITH ALTERNATIVE FUNCTIONS, DISPLAYED IN YELLOW
 - "EOB" STANDS FOR END OF BLOCK
- NUMERIC KEYS



OVERRIDES KEYS

THE OVERRIDE KEYS ALLOW YOU TO CHANGE THE FEED RATE, SPINDLE SPEED, SPINDLE DIRECTION AND RAPID MOVEMENT RATE



OVERRIDES KEYS

- WHILE RUNNING A PROGRAM YOU CAN:
- ADJUST THE FEED RATE OF THE MACHINE BY INCREMENTS OF 10%
- THE FEED RATE CAN BE RESET STRAIGHT TO 100%
- THE FEED RATE CAN BE CONTROLLED BY THE "HANDLE JOG" WHEEL A PERCENT AT A TIME
- THE SAME CAN BE DONE WITH THE SPINDLE SPEED
- THE DIRECTION OF THE SPINDLE CAN BE CHANGED
- THE RAPID MOVEMENT CAN BE DECREASED TO SPECIFIC INCREMENTS



JOG KEYS

THE JOG KEYS ALLOW YOU TO MOVE THE VARIOUS AXIS WITHIN THE MACHINE



JOG KEYS

- "TS" ← MOVES THE TAILSTOCK TOWARDS THE WORKING AREA
- "TS RAPID" MOVES THE TAILSTOCK IN THE A SELECTED DIRECTION AT THE RAPID FEED RATE
- "TS" → MOVES THE TAILSTOCK AWAY FROM THE WORKING AREA
- THESE ALLOW YOU TO MOVE THE MACHINE IN THE SPECIFIED DIRECTION
- "CHIP FWD" MOVES THE CHIP AUGER FORWARD, PUSHING CHIPS OUT
- "CHIP STOP" STOPS THE CHIP AUGER
- "CHIP REV" MOVES THE CHIP AUGER BACKWARDS, BRINGS CHIPS BACK IN



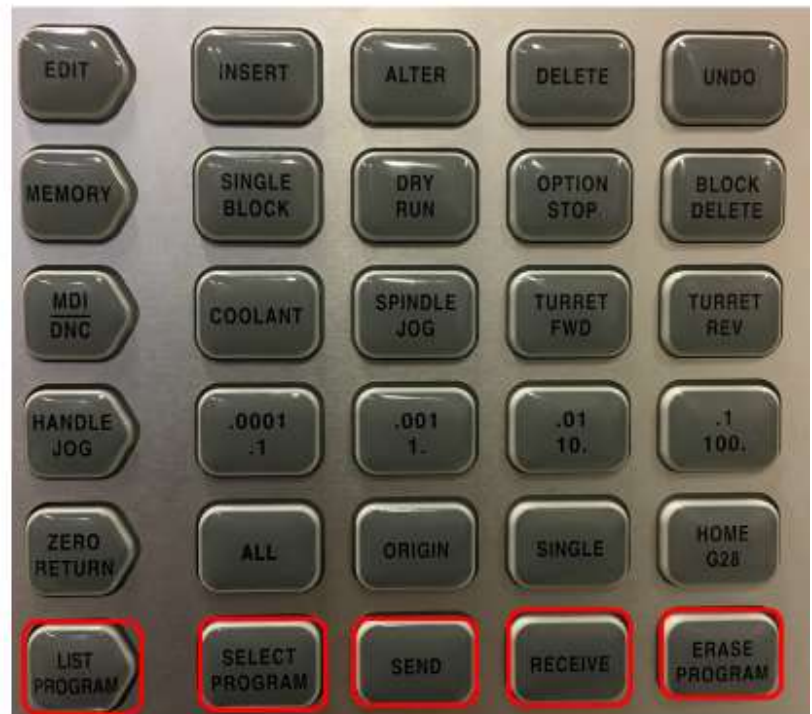
MODE KEYS

THE MODE KEYS ALLOW YOU TO USE VARIOUS FUNCTIONS THROUGHOUT YOUR MACHINE



MODE KEYS

- "LIST PROGRAM" WILL DISPLAY THE CURRENT PROGRAMS IN THE MACHINES MEMORY, A CONNECTED USB OR ANY OTHER MEMORY DRIVE CONNECTED TO YOUR MACHINE
- "SELECT PROGRAM" TAKES THE HIGHLIGHTED PROGRAM AND MAKES IT THE ACTIVE PROGRAM
- "SEND" ALLOWS YOU TO SEND A PROGRAM FROM THE MACHINE TO COMPUTER
- "RECEIVE" ALLOWS YOU TO RECEIVE A PROGRAM FROM A COMPUTER
- "ERASE PROGRAM" WILL ERASE THE HIGHLIGHTED PROGRAM. IF THE PROGRAM IS ACTIVE IT CANNOT BE ERASED



MODE KEYS

- WHEN PRESSING "LIST PROGRAM" THIS SCREEN :
 - "INSERT" WILL ALLOW YOU TO CREATE A NEW PROGRAM
 - "ENTER" WILL ALLOW YOU TO SELECT MULTIPLE PROGRAMS
 - "F2" WILL COPY ANY SELECTED PROGRAMS
 - "F3" WILL TO

Current Directory: Memory/

O #	Comment	File Name	Size	Last Modified
		09000	<DIR>	10-01-2018 13:49 >
		B	<DIR>	02-24-2017 07:59 >
		G17	<DIR>	06-29-2017 13:18 >
		JIM	<DIR>	07-23-2018 13:27 >
		PRECISION...	<DIR>	08-27-2018 16:31 >
		T10	<DIR>	08-07-2017 10:17 >
		TI 12mm _...	6 MB	04-26-2018 11:24
	(PRECISION PR...	precision ...	6 KB	06-19-2018 13:48
		radio1.nc	66 KB	12-01-2016 11:54
		radioactive...	68 KB	12-01-2016 11:33
00000	(Setup 1)	Setup_1.nc	17 KB	07-18-2017 12:22
00001	(04-00-00310 F...	O00001.nc	4 KB	10-12-2018 12:08
00002	(04-00-00310 F...	O00002.nc	4 KB	10-01-2018 16:17
00003	(CLAMP MULTI ...	O00003.txt	799 B	09-28-2017 07:19
00004	(SLUG)	SLUG.nc	4 KB	10-01-2018 13:44
00005	(1.25 SHAFT TE...	1.25 SHAF...	3 KB	08-20-2018 16:21

File Name:

Folder Has: 47 Items Disk Space: 731 MB Free (71%) Selected Items: 0

MODE KEYS

- WHEN PRESSING "INSERT" YOU WILL GET THIS PROMPT WHERE YOU WILL INPUT
 - PROGRAM NUMBER
 - THE FILE NAME
 - COMMENTS ON THE FILE

Current Directory: Memory/

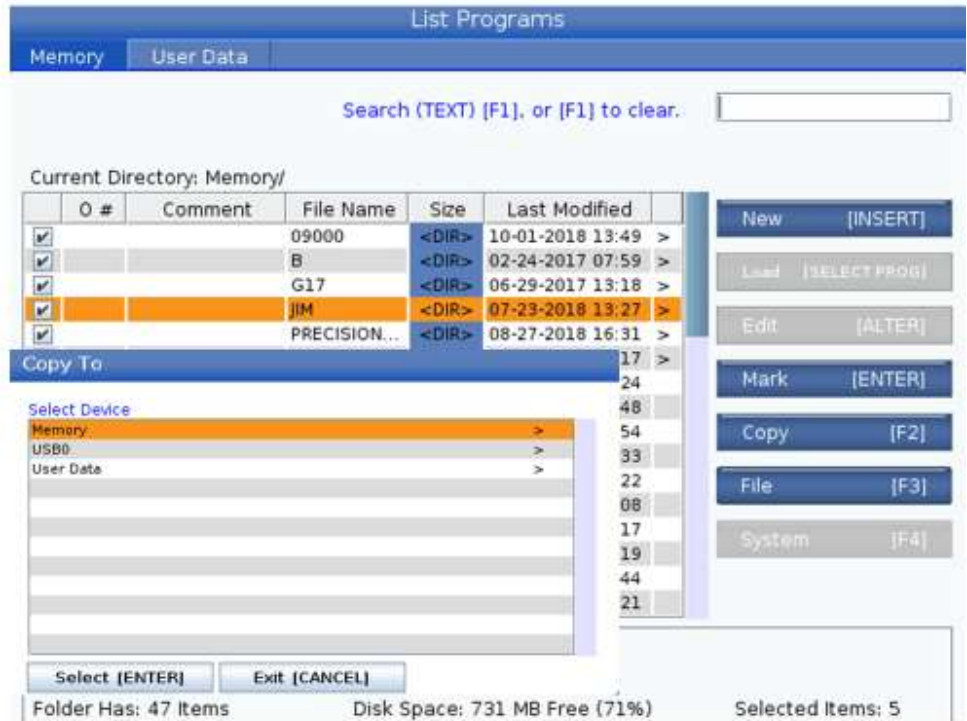
O #	Comment	File Name	Size	Last Modified
		09000	<DIR>	10-01-2018 13:49 >
		B	<DIR>	02-24-2017 07:59 >
		G17	<DIR>	06-29-2017 13:18 >
		JIM	<DIR>	07-23-2018 13:27 >
		PRECISION...	<DIR>	08-27-2018 16:31 >
		T10	<DIR>	08-07-2017 10:17 >
		TI 12mm _...	6 MB	04-26-2018 11:24
	(PF	precision ...	6 KB	06-19-2018 13:48
		radio1.nc	66 KB	12-01-2016 11:54
		radioactive...	68 KB	12-01-2016 11:33
00000	(S	Setup_1.nc	17 KB	07-18-2017 12:22
00001	(0	O00001.nc	4 KB	10-12-2018 12:08
00002	(0	O00002.nc	4 KB	10-01-2018 16:17
00003	(C	O00003.txt	799 B	09-28-2017 07:19
00004	(S	SLUG.nc	4 KB	10-01-2018 13:44
00005	(L	1.25 SHAF...	3 KB	08-20-2018 16:21

File Name:

Folder Has: 47 Items Disk Space: 731 MB Free (71%) Selected Items: 0

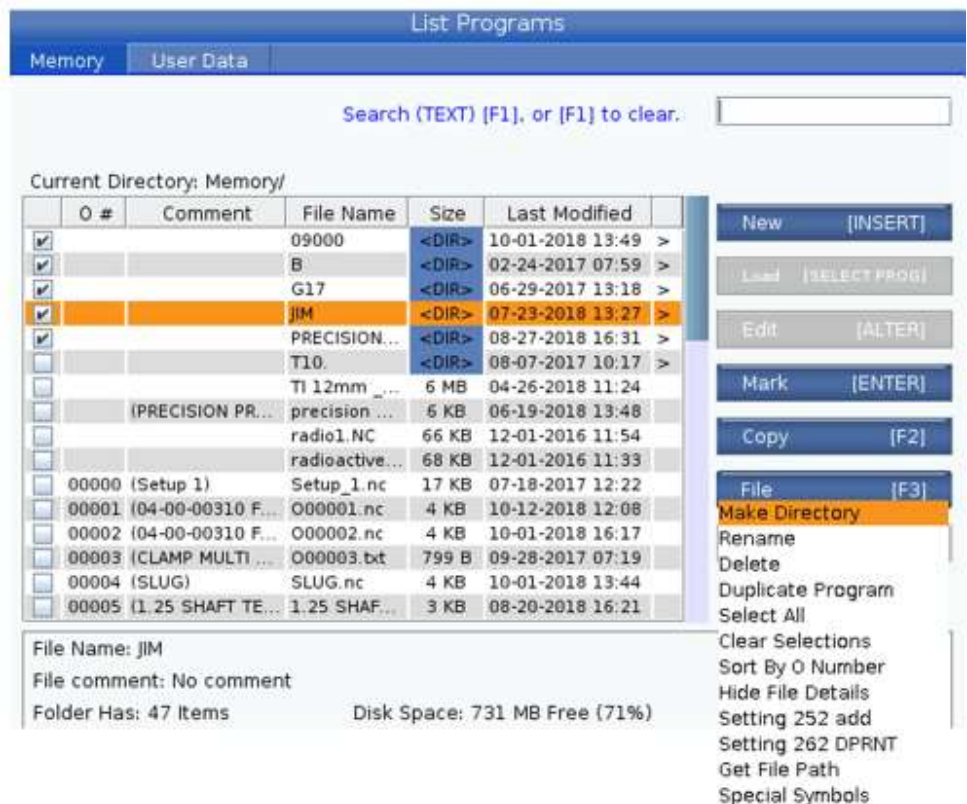
MODE KEYS

- WHEN PRESSING "F2" YOU WILL BE PROMPTED TO SELECT A LOCATION TO COPY THE PROGRAMS TO
 - YOU CAN SELECT TO SAVE TO THE MEMORY, A USB OR A NETWORK DRIVE THAT YOU MAY HAVE SET UP
 - FILES CAN BE SAVED INTO ANY FOLDERS WITHIN THESE STORAGE LOCATIONS



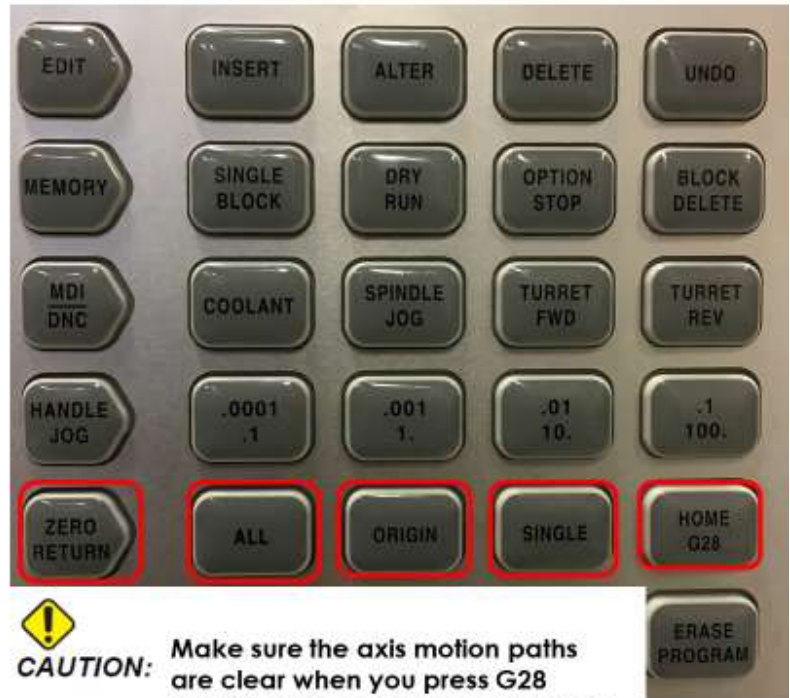
MODE KEYS

- WHEN PRESSING "F3" YOU WILL BE PROMPTED TO SELECT :
 - MAKE A DIRECTORY (MAKE A FOLDER)
 - RENAME (A PROGRAM)
 - DUPLICATE PROGRAM
 - SELECT ALL
 - CLEAR SELECTIONS
 - SORT BY O NUMBER
 - HIDE FILE DETAILS
 - SETTING 252 ADD (SUBPROGRAM LOCATION)
 - SETTING 262 DPRNT (ALLOWS MACHINE TO COMMUNICATE WITH EXTERNAL DEVICES)
 - GET FILE PATH
 - SPECIAL SYMBOLS



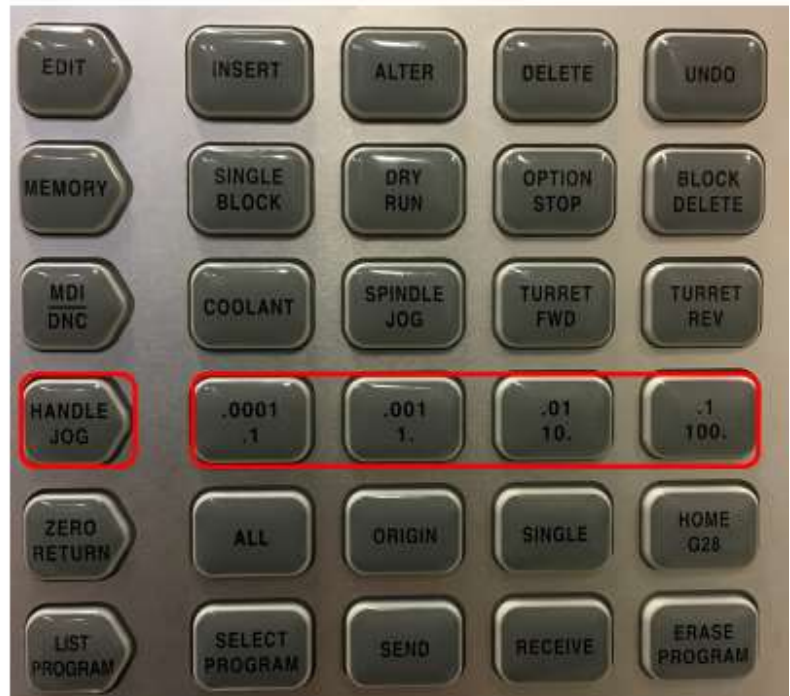
MODE KEYS

- "ZERO RETURN" ALLOWS YOU TO RETURN THE AXIS TO MACHINE ZERO
- "ALL" PRESSED AFTER "ZERO RETURN" WILL RETURN ALL AXIS TO THEIR MACHINE HOME POSITION
- "ORGIN" SETS SELECTED VALUES TO ZERO IN DIFFERENT DISPLAYS
- "SINGLE" WHEN PRESSED AFTER SELECTING AN AXIS USING THE ALPHA KEYS AND "ZERO RETURN" WILL RETURN ONLY THE SELECTED AXIS TO MACHINE HOME
- "HOME G28" WILL RETURN ALL THE AXIS TO HOME IN A RAPID MOTION. BY SELECTING AN AXIS BEFORE PRESSING THIS YOU CAN RETURN A SINGLE AXIS AT A TIME



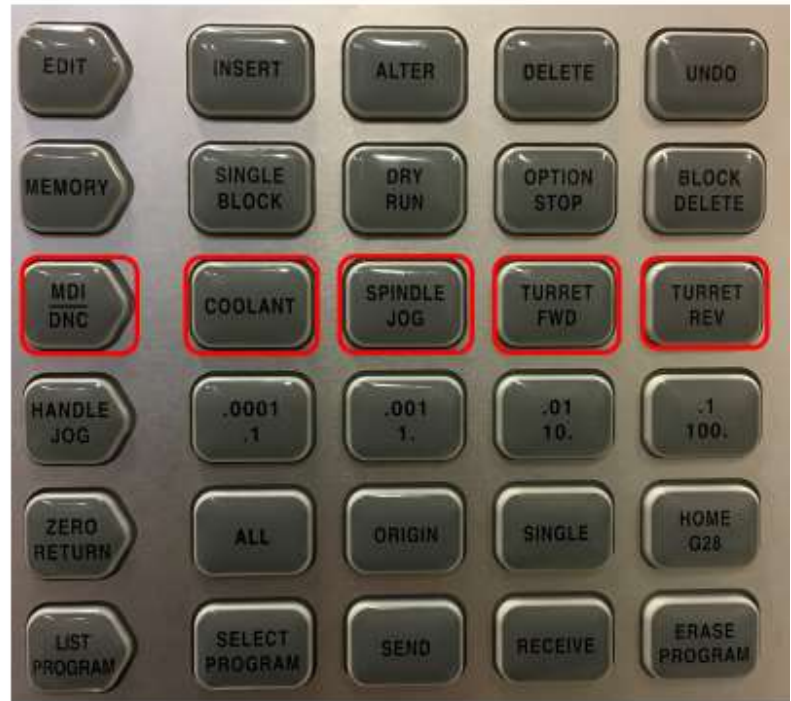
MODE KEYS

- "HANDLE JOG" ALLOWS YOU TO SELECT AND AXIS USING THE JOG KEYS AND MOVE THAT AXIS USING THE HAND WHEEL
- THE TOP NUMBER IS THE RATE THAT THE MACHINE WILL MOVE PER CLICK OF THE HAND WHEEL. WHEN IN MM THE DECIMAL TO RIGHT ONE SPACE (I.E. .0001 →.001)
- THE BOTTOM NUMBER SETS THE MOVEMENT SPEED AFTER PRESSING "JOG LOCK" AND SELECTING THE AXIS YOU WOULD LIKE TO MOVE



MODE KEYS

- "MDI/DNC" ALLOWS YOU TO GO INTO MDI MODE WHICH IS A PLACE TO WRITE TEMPORARY PROGRAMS OR CODES
- "COOLANT" TURNS THE COOLANT ON
- "SPINDLE JOG" ALLOWS YOU TO USE THE HAND WHEEL TO MOVE THROUGH A PROGRAM OR OTHER MENUS
- "TURRET FWD" ROTATES THE TURRET TO THE NEXT TOOL
- "TURRET REV" ROTATES THE TURRET TO THE PREVIOUS TOOL
- BY INPUTTING THE TOOL NUMBER BEFORE PRESSING "TURRET FWD" OR "TURRET REV" YOU CAN GO DIRECTLY TO THE TOOL NUMBER AND THE MACHINE WILL GO IN THE MOST EFFICIENT DIRECTION



MODE KEYS

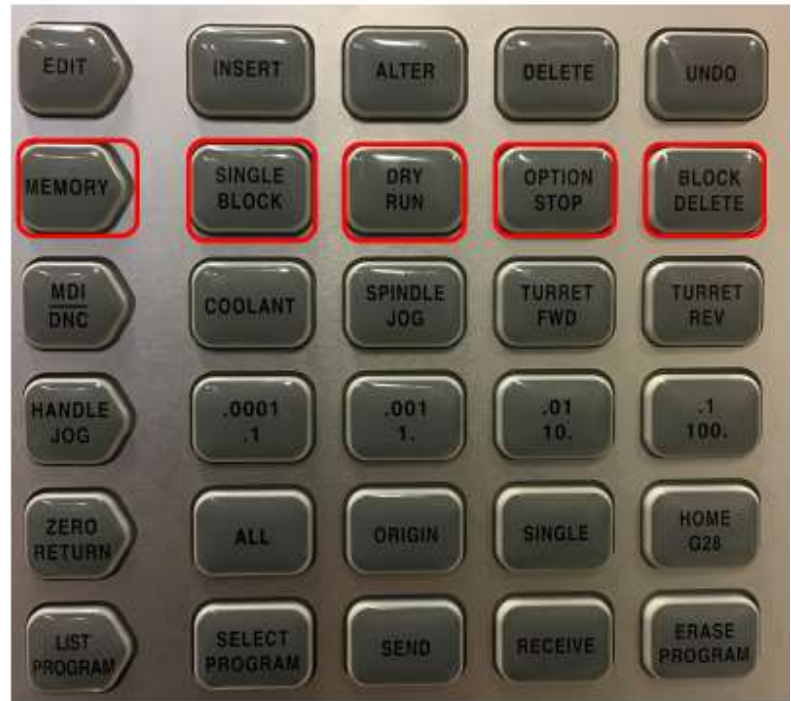
- PRESSING "MDI" WILL BRING YOU TO THE MDI MODE WHERE YOU CAN CREATE TEMPORARY PROGRAMS

```

MDI
NO
(OD Profile Removal Cycles);
( SAFETY LINE BELOW );
G00 G54 G18 G40 G80 G97 G99;
( TOOL = 1 / OFFSET = 1 );
( MAXIMUM SPINDLE RPM = 1800 );
( SURFACE SPEED SPEED = 500 );
( DEPTH PER PASS = 0.05 );
( WORK OFFSET = 54);
(OD STOCK REMOVAL CYCLE);
T101;
G54;
G50 S1800;
G96 S500 M03;
G00 Z0.1;
M08;
G00 X1.1 Z0.1;
G71 P1720 Q1721 D0.05 U0.01 W0.003 F0.01;
N1720 G40 X0. Z0.1;
G01 X0. Z0.025;
G01 X0. Z0.025;
G01 X0. Z0.;
G01 X0.25 Z0.;
G01 X0.25 Z-1.;
G01 X0.5 Z-1.;
G01 X0.5 Z-2.;
G01 X0.75 Z-2.;
G01 X0.75 Z-2.;
G01 X0.75 Z-3.
    
```

MODE KEYS

- “MEMORY” WILL BRING YOU TO YOUR SELECTED PROGRAM AND BRING YOU INTO A STATE TO RUN THE PROGRAM
- “SINGLE BLOCK” WILL ALLOW YOU TO RUN THROUGH A PROGRAM ONE LINE AT A TIME
- “DRY RUN” IS USED TO CHECK YOUR PROGRAM BY CUTTING AIR AND WITH LOWER FEEDS AND RAPID MOVEMENTS. USUALLY RUN WITHOUT A PART IN THE MACHINE
- “OPTION STOP” WHEN TURNED ON, THE MACHINE WILL STOP WHEN IT REACHES AN M01 COMMAND
- “BLOCK DELETE” WILL TELL YOUR PROGRAM TO IGNORE ANY LINES WITH A FORWARD SLASH (“/”) AT THE BEGINNING OF THE LINE



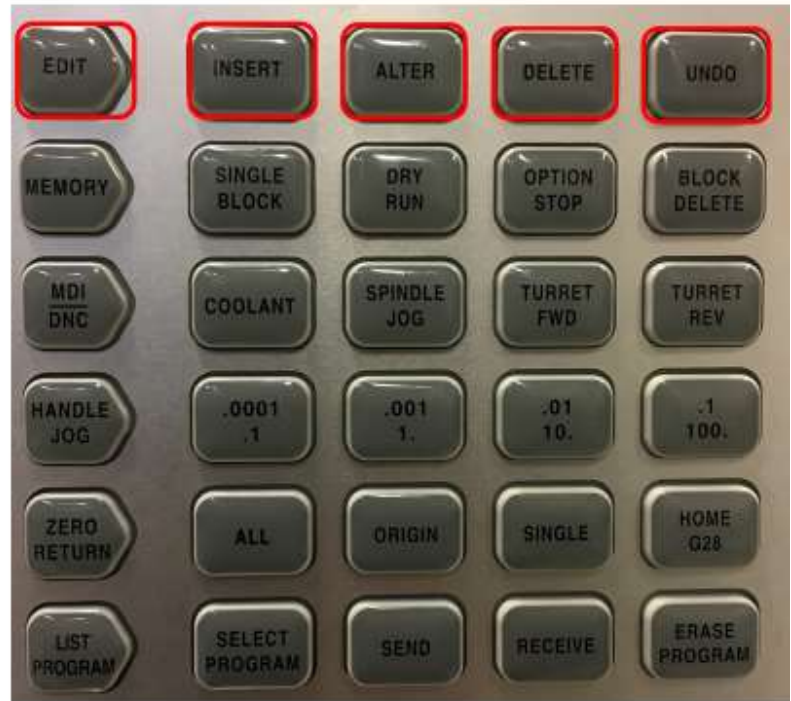
MODE KEYS

- PRESSING MEMORY WILL BRING UP YOUR SELECTED PROGRAM INTO A STATE BE RUN BY PRESSING “CYCLE START”

```
MEM          Memory/LTI TEST.nc          NO
054321 (LTI_TEST_OP1);
(OD TURN CYCLE);
(SAFETY LINE BELOW);
G00 G54 G18 G40 G80 G97 G99;
(TOOL = 1 / OFFSET = 1);
(WORK OFFSET = 54);
(MAXIMUM SPINDLE RPM = 1800);
(CONSTANT SURFACE SPEED = 500);
(OUTSIDE DIAMETER: 2.);
(FINISH DIAMETER: 1.);
(FILLET RADIUS = .1);
(DEPTH PER PASS = 0.05);
T101;
G54;
G50 S1800;
G97 S720 M03;
G00 Z0.2;
G00 X2.4 M08;
G96 S500;
G71 P37 Q38 D0.05 U0.01 W0.003 F0.01;
N37 G00 X1.;
G01 Z-0.5 ,R0.0840;
N38 X2.4;
G00 Z0.2;
G97 S720 M09;
G00 X12. (SAFE TC POSITION);
G00 Z36. (SAFE TC POSITION);
M05;
```

MODE KEYS

- "EDIT" ALLOWS YOU TO EDIT YOUR SELECTED PROGRAM AND REACH VPS (VISUAL PROGRAMMING SYSTEM) IN YOUR MACHINE
- "INSERT" WILL INPUT ANY INFORMATION FROM YOUR INPUT BAR OR THE CLIPBOARD AFTER THE CURSOR POSITION
- "ALTER" REPLACES THE HIGHLIGHTED TEXT WITH THE TEXT FROM THE INPUT BAR OR THE CLIPBOARD
- "DELETE" WILL DELETE THE HIGHLIGHTED TEXT
- "UNDO" WILL UNDO THE LAST 40 CHANGES MADE TO A PROGRAM



MODE KEYS

- PRESSING "EDIT" WILL ALLOW YOU TO EDIT YOUR PROGRAM. BY PRESSING:
 - "MEMORY" YOU WILL SAVE ANY CHANGED AND SET IT AS THE ACTIVE PROGRAM. IT WILL BRING YOU TO A POSITION TO RUN THE PROGRAM
 - "F1" WILL ALLOW YOU TO ACCESS THE MENU ON THE TOP TAB
 - "F2" WILL ALLOW YOU TO SELECT MULTIPLE LINES OF G-CODE
 - "F4" WILL PASTE ANYTHING SAVED TO THE CLIPBOARD INTO THE PROGRAM



MODE KEYS

- ONCE YOU PRESS "F1" YOU WILL GO TO THE MENU WHERE YOU CAN:
 - "NEW" IS USED TO CREATE A NEW PROGRAM
 - "SET TO RUN" SETS THE PROGRAM TO BE THE ACTIVE PROGRAM
 - "SAVE" WILL SAVE THE PROGRAM AND OVERWRITES THE OLDER VERSION
 - "SAVE AS" ALLOWS YOU SAVE THE PROGRAM AS A NEW PROGRAM
 - "DISCARD CHANGES" WILL ALLOW THE USER TO DISCARD AND CHANGES TO THE PROGRAM THAT THEY HAVE MADE



MODE KEYS

- ONCE YOU PRESS "F1" YOU WILL GO TO THE MENU WHERE YOU CAN:
 - "UNDO" WILL UNDO ANY CHANGES MADE UP TO 40 CHANGES
 - "REDO" WILL REDO ANY CHANGES MADE UP TO 40 CHANGES
 - "CUT SELECTION TO CLIPBOARD" REMOVES THE SELECTED LINES AND PUT THEM INTO THE CLIPBOARD
 - "COPY SELECTION TO CLIPBOARD" COPIES THE SELECTED LINES AND PUTS THEM INTO THE CLIPBOARD



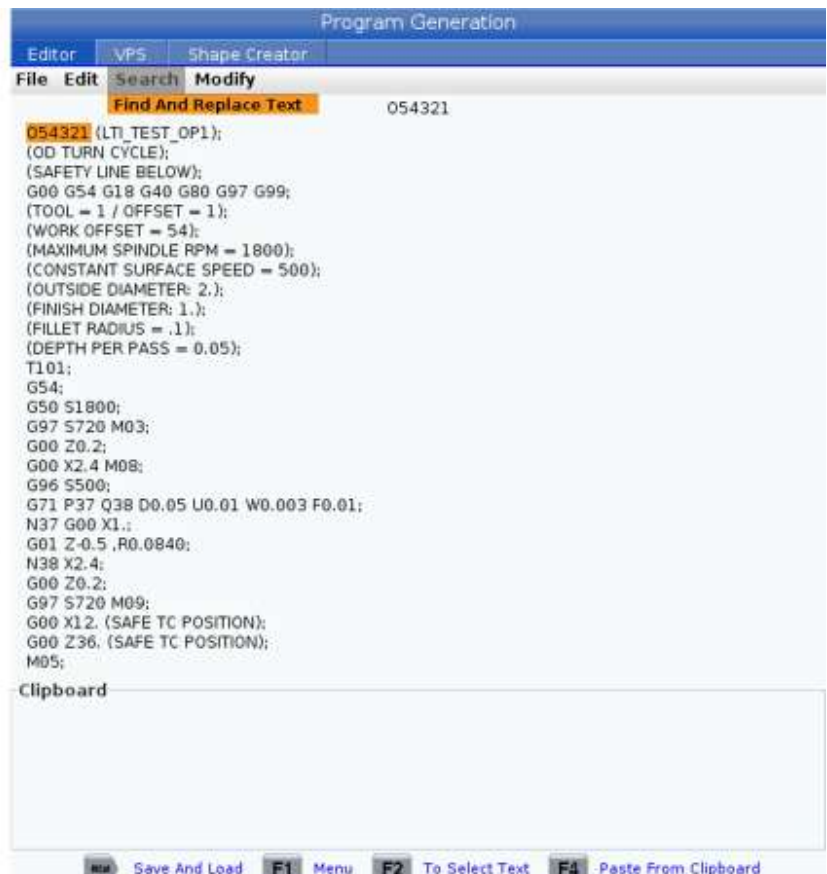
MODE KEYS

- ONCE YOU PRESS "F1" YOU WILL GO TO THE MENU WHERE YOU CAN:
 - "PASTE FROM CLIPBOARD" WILL PASTE ANYTHING SAVED IN THE CLIPBOARD ON THE LINE BELOW THE LINE HIGHLIGHTED
 - "INSERT FILE PATH (M98)" TO PUT THE FILE PATH INFORMATION
 - "INSERT MEDIA FILE (M130) WILL INPUT AN M130 WHICH WILL CAUSE A SPECIFIED MEDIA FILE TO APPEAR ON SCREEN
 - "SPECIAL SYMBOLS" ALLOWS YOU TO ACCESS AND INPUT SPECIAL SYMBOLS



MODE KEYS

- ONCE YOU PRESS "F1" YOU WILL GO TO THE MENU WHERE YOU CAN:
 - "FIND AND REPLACE TEXT" WILL ALLOW YOU TO SEARCH THE PROGRAM FOR SPECIFIC LINES OF CODE AND CHANGE THESE LINE(S)



MODE KEYS

- ONCE SELECTING "FIND AND REPLACE TEXT" YOU WILL BE PROMPTED TO:
 - INPUT WHAT YOU WOULD LIKE TO SEARCH FOR
 - INPUT WHAT YOU'D LIKE TO REPLACE IT WITH IF YOU CHOOSE SO
- BY PRESSING:
 - "F1" YOU WILL BE ABLE TO SEARCH THE PROGRAM WITH WHAT YOU INPUT INTO THE FIND LINE
 - "F2" WILL REPLACE THE LINE ONCE IT WAS FOUND WITH WHAT IS IN THE REPLACE LINE
 - "F3" WILL FIND AND REPLACE THE NEXT INSTANCE OF THE LINE YOU ARE SEARCHING FOR. NOTE THAT BOTH FIND AND REPLACE SHOULD BE FILLED OUT BEFORE DOING THIS TASK
 - "F4" WILL FIND AND REPLACE ALL THE INSTANCES OF THE LINE YOU SEARCHED FOR WITH WHAT IS IN THE REPLACE LINE. THIS CANNOT BE UNDONE



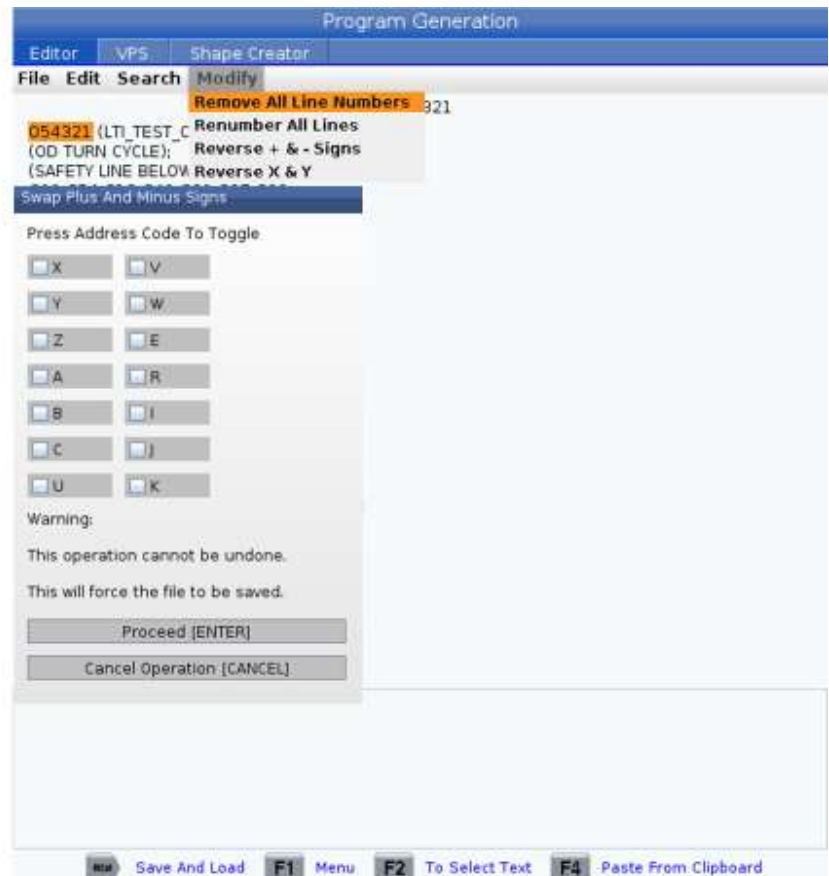
MODE KEYS

- ONCE YOU PRESS "F1" YOU WILL GO TO THE MENU WHERE YOU CAN:
 - "REMOVE ALL LINE NUMBERS" WILL REMOVE ANY N### THAT ARE IN THE PROGRAM
 - "RENUMBER ALL LINES" WILL INPUT LINE NUMBERS
 - LINE NUMBERS ARE A GOOD WAY TO REFERENCE DIFFERENT PARTS OF THE PROGRAM
 - "REVERSE + & -" WILL CHANGE ALL OF THE + WITH - ON THE SELECTED AXIS
 - "REVERSE X & Y" WILL SWITCH ALL THE XS AND Ys



MODE KEYS

- WHEN SELECTING "REVERSE + & -" YOU WILL BE PROMPTED TO CHOOSE WHICH AXIS OR AXES YOU WOULD LIKE TO CHANGE.
 - NOTE THIS CANNOT BE UNDONE AND WILL SAVE THE FILE AFTER MAKING THE CHANGE
 - PRESS "ENTER" TO CONFIRM AND "CANCEL" TO CANCEL



DISPLAY KEYS

- DISPLAY KEYS ALLOW YOU TO VIEW DIFFERENT MACHINE DISPLAYS, OPTIONAL INFORMATION AND HELP PAGES



DISPLAY KEYS

"PROGRAM" WILL BRING UP YOUR CURRENT PROGRAM, WHETHER IT IS MDI OR MEMORY



DISPLAY KEYS

• WHEN SELECTING PROGRAM YOU WILL BE GIVEN THIS SCREEN, HERE YOU CAN:

- RUN THE CURRENT PROGRAM
- VIEW THE ACTIVE CODES
- VIEW THE ACTIVE TOOL
- VIEW THE COOLANT LEVEL
- VIEW INFORMATION ON THE SPINDLES

Operation: MEM | 11:25:56

MEM Memory/LTI TEST.nc N0

054321 (LTI TEST_DP1):
 (OD TURN CYCLE);
 (SAFETY LINE BELOW);
 G00 G54 G18 G40 G80 G97 G99;
 (TOOL = 1 / OFFSET = 1);
 (WORK OFFSET = 54);
 (MAXIMUM SPINDLE RPM = 1800);
 (CONSTANT SURFACE SPEED = 500);
 (OUTSIDE DIAMETER: 2.);
 (FINISH DIAMETER: 1.);
 (FILLET RADIUS = .1);
 (DEPTH PER PASS = 0.05);
 T101;
 G54;
 G50 S1800;
 G97 S720 M03;
 G00 Z0.2;
 G00 X2.4 M08;
 G96 S500;
 G71 P37 Q38 D0.05 U0.01 W0.003 F0.01;
 N37 G00 X1.;
 G01 Z-0.5 R0.0840;
 N38 X2.4;
 G00 Z0.2;
 G97 S720 M09;
 G00 X1.2 (SAFE TC POSITION);
 G00 Z36 (SAFE TC POSITION);
 M05.

Active Program

Active Codes		Active Tool		Coolant
G00	Rapid Motion	Tool: 1	Offset: 1	Off
G99	Feed Per Revolution	Type: OD Turn		
G40	Cancel Tool Nose Compensation	Tool Group: ---		1/1
G80	Cycle Cancel	Max Load: 0		0/1
G54	Work Offset #54	Life: 100%		

D00 H00 M00 T0

Spindles

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

Main Spindle

STOP
Overrides

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

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

Spindle Load(%) 0%

Positions Program: G54 T101

Axis	(IN)	Load
X	0.0000	0%
Z	0.0000	0%
B	0.0	0%

Timers And Counters

This Cycle:	0:00:01
Last Cycle:	0:00:01
Remaining:	0:00:00
M30 Counter #1:	0
M30 Counter #2:	0
Loops Remaining:	0

Setup Power Save
SIM

DISPLAY KEYS

"POSITION" WILL BRING UP THE MACHINE POSITIONS. HERE IS THE PROGRAM POSITION, DISTANCE TO GO, MACHINE POSITION AND OPERATOR POSITION CAN BE VIEWED



POSITION

- WHEN SELECTING POSITION YOU WILL BE ABLE TO DISPLAY DIFFERENT POSITIONS USED BY THE MACHINE INCLUDING:

"PROGRAM" WHICH IS THE MACHINE'S POSITION IN REFERENCE TO YOUR CURRENT WORK OFFSET

Operation: MEM | 11:31:16

MEM | Memory/LTI TEST.nc | NO

054321 (LTI TEST_DP1);
 (OD TURN CYCLE);
 (SAFETY LINE BELOW);
 G00 G54 G18 G40 G80 G97 G99;
 (TOOL = 1 / OFFSET = 1);
 (WORK OFFSET = 54);
 (MAXIMUM SPINDLE RPM = 1800);
 (CONSTANT SURFACE SPEED = 500);
 (OUTSIDE DIAMETER: 2.);
 (FINISH DIAMETER: 1.);
 (FILLET RADIUS = .1);
 (DEPTH PER PASS = 0.05);
 T101;
 G54;
 G50 S1800;
 G97 S720 M03;
 G00 Z0.2;
 G00 X2.4 M08;
 G96 S500;
 G71 P37 Q38 D0.05 U0.01 W0.003 F0.01;
 M37 G09 X1.;
 G01 Z-0.5 R0.0040;
 M38 X2.4;
 G00 Z0.2;
 G97 S720 M09;
 G00 X12. (SAFE TC POSITION);
 G00 Z36. (SAFE TC POSITION);
 M05;

Axis	Position: (IN)	Load
X	0.0000	0%
Z	0.0000	0%
B	0.0	0%

ALARM To view options.

Main Spindle

STOP

Overrides

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

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

Spindle Load(%) 0%

Positions

Program G54 T101

Axis	(IN)	Load
X	0.0000	0%
Z	0.0000	0%
B	0.0	0%

Timers And Counters

This Cycle: 0:00:01
 Last Cycle: 0:00:01
 Remaining: 0:00:00
 M30 Counter #1: 0
 M30 Counter #2: 0
 Loops Remaining: 0

Setup Power Save

SPN

POSITION

"DISTANCE TO GO" WILL DISPLAY THE DISTANCE THE MACHINE MUST TRAVEL IN ORDER TO COMPLETE THE NEXT LINE OF CODE

Positions					
Program	Distance To Go	Machine	Operator	All	
Axis	Position: (IN)		Load		
X	0.0000		46%		
Z	-0.4370		0%		
B	0.0		0%		

To view options.

POSITION

"MACHINE" IS THE MACHINES POSITION IN REFERENCE TO THE MACHINE'S HOME POSITION

Positions					
Program	Distance To Go	Machine	Operator	All	
Axis	Position: (IN)		Load		
X	-8.6900		46%		
Z	-10.2009		0%		
B	0.0		0%		

To view options.

POSITION

"OPERATOR" CAN BE USED TO GET A MEASUREMENT FROM A SPECIFIC LOCATION. TO DO THIS YOU MUST BE IN "HANDLE JOG" MODE WITH THIS DISPLAY UP

Positions				
Program	Distance To Go	Machine	Operator	All
Axis	Position: (IN)		Load	
X	0.0000		46%	
Z	-10.2009		0%	
B	0.0		0%	

[Axis Letter] + **ORIGIN** Reset axis position

[Axis Letter] + **ENTER** Set axis position **ALTER** To view options.

POSITION

"ALL" SHOWS ALL OF THE PREVIOUS 4 DISPLAYS ON ONE SCREEN

Positions				
Program	Distance To Go	Machine	Operator	All
Position: (IN)				
Axis	G54 T101	Axis	Dist To Go	
X	1.3100	X	0.0000	
Z	-0.2009	Z	0.0000	
B	0.0	B	0.0	
Axis	Machine	Axis	Operator	
X	-8.6900	X	0.0000	
Z	-10.2009	Z	-10.2009	
B	0.0	B	0.0	

ALTER To view options.

POSITION

THESE POSITIONS CAN BE DISPLAYED ON THE BOTTOM CENTER IN THE "POSITION" DISPLAY. THE ONLY ONE THAT CAN'T BE DISPLAYED IS ALL 4 AT ONCE

Operation: MEM | 11:31:16

MEM Memory/LTI TEST.nc N0

```

054321 (LTI_TEST_OP1);
(OD TURN CYCLE);
(SAFETY LINE BELOW);
G00 G54 G18 G40 G80 G97 G99;
(TOOL = 1 / OFFSET = 1);
(WORK OFFSET = 54);
(MAXIMUM SPINDLE RPM = 1000);
(CONSTANT SURFACE SPEED = 500);
(OUTSIDE DIAMETER: 2.);
(FINISH DIAMETER: 1.);
(FILLET RADIUS = .1);
(DEPTH PER PASS = 0.05);
T101;
G54;
G50 S1000;
G97 S720 M03;
G00 Z0.2;
G00 X2.4 M08;
G96 S500;
G71 P37 Q38 D0.05 U0.01 W0.003 F0.01;
M37 G00 X1.;
G01 Z-0.5 ,R0.0040;
M38 X2.4;
G00 Z0.2;
G97 S720 M09;
G00 X12. (SAFE TC POSITION);
G00 Z36. (SAFE TC POSITION);
M05;

```

Axis	Position: (IN)	Load
X	0.0000	0%
Z	0.0000	0%
B	0.0	0%

ALARM To view options.

Main Spindle

STOP

Overrides

Feed: 100%

Spindle: 100%

Rapid: 100%

Spindle Load(%) 0%

Positions Program: G54 T101

	(IN)	Load
X	0.0000 	0%
Z	0.0000 	0%
B	0.0 	0%

Timers And Counters

This Cycle: 0:00:01

Last Cycle: 0:00:01

Remaining: 0:00:00

M30 Counter #1: 0

M30 Counter #2: 0

Loops Remaining: 0

Setup Power Save

LTI

DISPLAY KEYS

"OFFSET" WILL BRING UP YOUR OFFSETS PAGE WITH A TAB FOR TOOL OFFSETS AND A TABLE FOR WORK OFFSETS



DISPLAY KEYS

• IN THE OFFSETS PAGE YOU CAN INPUT YOUR "TOOL" OFFSET INFORMATION

▪ IN MOST SECTIONS:

- "F2" WILL SET THE INPUT TO THE VDI CENTER LINE
- "F3" WILL SET THE INPUT TO THE BOT CENTER LINE
- "F1" WILL INSERT THE NUMBER FROM YOUR INPUT BAR AND REPLACE THE NUMBER CURRENTLY DISPLAYED
- "ENTER" WILL ADD/SUBTRACT THE INPUT LINE WITH THE DISPLAYED NUMBER
- "F4" WILL BRING TO THE "WORK" OFFSET PAGE

▪ OTHER OPTIONS ARE DISPLAYED AT THE BOTTOM OF THIS SCREEN

Offsets					
Tool	Work				
Active Tool: 1					
Tool Offset	Turret Location	X Geometry	Z Geometry	Radius Geometry	Tip Direction
1 Spindle	1	-10.0000	-10.0000	0.0320	3: X- Z-
2	0	0.	0.	0.	0: None
3	0	0.	0.	0.	0: None
4	0	0.	0.	0.	0: None
5	0	0.	0.	0.	0: None
6	0	0.	0.	0.	0: None
7	0	0.	0.	0.	0: None
8	0	0.	0.	0.	0: None
9	0	0.	0.	0.	0: None
10	0	0.	0.	0.	0: None
11	0	0.	0.	0.	0: None
12	0	0.	0.	0.	0: None
13	0	0.	0.	0.	0: None
14	0	0.	0.	0.	0: None
15	0	0.	0.	0.	0: None
16	0	0.	0.	0.	0: None
17	0	0.	0.	0.	0: None
18	0	0.	0.	0.	0: None

Enter A Value

F2 Set to VDI center line F3 Set to BOT center line
X DIAMETER MEASURE X Diameter Measure F1 Set Value ENTER Add To Value F4 Work Offset

DISPLAY KEYS

"TURRET LOCATION" IS THE POSITION YOUR TOOL IS IN ON THE TURRET

"X GEOMETRY" IS THE LENGTH IN THE X DIRECTION

"Z GEOMETRY" IS THE LENGTH IN THE Z AXIS

"RADIUS GEOMETRY" IS THE RADIUS ON YOUR TOOL'S INSERT

"TIP DIRECTION" IS THE DIRECTION YOU TOOL IS FACING WHEN IT WILL BE CUTTING (PRESSING "F1" WHEN IN THIS BOX WILL GIVE YOU A LIST OF OPTIONS)

Offsets

Tool Work

Active Tool: 1

Tool Offset	Turret Location	X Geometry	Z Geometry	Radius Geometry	Tip Direction
1 Spindle	1	-10.0000	-10.0000	0.0320	3: X- Z-
2	0	0.	0.	0.	0: None
3	0	0.	0.	0.	0: None
4	0	0.	0.	0.	0: None
5	0	0.	0.	0.	0: None
6	0	0.	0.	0.	0: None
7	0	0.	0.	0.	0: None
8	0	0.	0.	0.	0: None
9	0	0.	0.	0.	0: None
10	0	0.	0.	0.	0: None
11	0	0.	0.	0.	0: None
12	0	0.	0.	0.	0: None
13	0	0.	0.	0.	0: None
14	0	0.	0.	0.	0: None
15	0	0.	0.	0.	0: None
16	0	0.	0.	0.	0: None
17	0	0.	0.	0.	0: None
18	0	0.	0.	0.	0: None

Enter A Value

X DIAMETER MEASURE X Diameter Measure

F2 Set to VDI center line F3 Set to BOT center line

F1 Set Value ENTER Add To Value F4 Work Offset

DISPLAY KEYS

"X GEOMETRY WEAR" IS THE AMOUNT OF WEAR ON THE TOOL IN THE X AXIS

"Z GEOMETRY WEAR" IS THE AMOUNT OF WEAR ON THE TOOL IN THE Z AXIS

"RADIUS WEAR" IS THE WEAR ON THE RADIUS OF THE TOOL

Offsets

Tool Work

Active Tool: 1

Tool Offset	X Geometry Wear	Z Geometry Wear	Radius Wear
1 Spindle	0.	0.	0.
2	0.	0.	0.
3	0.	0.	0.
4	0.	0.	0.
5	0.	0.	0.
6	0.	0.	0.
7	0.	0.	0.
8	0.	0.	0.
9	0.	0.	0.
10	0.	0.	0.
11	0.	0.	0.
12	0.	0.	0.
13	0.	0.	0.
14	0.	0.	0.
15	0.	0.	0.
16	0.	0.	0.
17	0.	0.	0.
18	0.	0.	0.

Enter A Value

X DIAMETER MEASURE X Diameter Measure

F1 Set Value ENTER Add To Value F4 Work Offset

DISPLAY KEYS

“LIVE TOOL RADIUS” IS THE RADIUS OF THE LIVE TOOL

“LIVE TOOL WEAR” IS THE WEAR THAT IS ON THE LIVE TOOL

“FLUTES” IS THE NUMBER OF FLUTES ON THE LIVE TOOL

“ACTUAL DIAMETER” IS THE DIAMETER OF THE TOOL

Offsets				
Tool	Work			
Active Tool: 1				
Tool Offset	Live Tool Radius	Live Tool Wear	Flutes	Actual Diameter
1 Spindle	0.	0.	0	0.
2	0.	0.	0	0.
3	0.	0.	0	0.
4	0.	0.	0	0.
5	0.	0.	0	0.
6	0.	0.	0	0.
7	0.	0.	0	0.
8	0.	0.	0	0.
9	0.	0.	0	0.
10	0.	0.	0	0.
11	0.	0.	0	0.
12	0.	0.	0	0.
13	0.	0.	0	0.
14	0.	0.	0	0.
15	0.	0.	0	0.
16	0.	0.	0	0.
17	0.	0.	0	0.
18	0.	0.	0	0.

Enter A Value

F2 Set to VDI center line **F3** Set to BOT center line

X DIAMETER MEASURE **F1** Set Value **ENTER** Add To Value **F4** Work Offset

DISPLAY KEYS

“APPROXIMATE X” IS A CLOSE MEASUREMENT OF THE TOOL IN THE X AXIS

“APPROXIMATE Z” IS A CLOSE MEASUREMENT OF THE TOOL IN THE Z AXIS

“APPROXIMATE RADIUS” IS A CLOSE MEASUREMENT OF THE RADIUS OF YOUR TOOL

“EDGE MEASURE HEIGHT” IS HOW FAR IT IS TO GET THE MAXIMUM DIAMETER OF THE TOOL

“TOOL TOLERANCE” IS THE AMOUNT OF WEAR A TOOL CAN HAVE BEFORE IT IS NO LONGER USEABLE

“PROBE TYPE” IS HOW YOU WOULD LIKE TO PROBE THE TOOL

Offsets						
Tool	Work					
Active Tool: 1						
Tool Offset	Approximate X	Approximate Z	Approximate Radius	Edge Meas... Height	Tool Tolerance	Probe Type
1 Spindle	0.	0.	0.	0.	0.	None
2	0.	0.	0.	0.	0.	None
3	0.	0.	0.	0.	0.	None
4	0.	0.	0.	0.	0.	None
5	0.	0.	0.	0.	0.	None
6	0.	0.	0.	0.	0.	None
7	0.	0.	0.	0.	0.	None
8	0.	0.	0.	0.	0.	None
9	0.	0.	0.	0.	0.	None
10	0.	0.	0.	0.	0.	None
11	0.	0.	0.	0.	0.	None
12	0.	0.	0.	0.	0.	None
13	0.	0.	0.	0.	0.	None
14	0.	0.	0.	0.	0.	None
15	0.	0.	0.	0.	0.	None
16	0.	0.	0.	0.	0.	None
17	0.	0.	0.	0.	0.	None
18	0.	0.	0.	0.	0.	None

Enter A Value

X DIAMETER MEASURE Automatic Probe Options **F1** Set Value **ENTER** Add To Value **F4** Work Offset

DISPLAY KEYS

- IN THE OFFSETS PAGE YOU CAN INPUT YOU "WORK" OFFSET INFORMATION

- IN MOST SECTIONS:

- "F1" WILL INSERT THE NUMBER FROM YOUR INPUT BAR AND REPLACE THE NUMBER CURRENTLY DISPLAYED
- "ENTER" WILL ADD/SUBTRACT THE INPUT LINE WITH THE DISPLAYED NUMBER
- "F4" WILL BRING TO THE "WORK" OFFSET PAGE

Offsets			
Tool	Work	Axes Info	
G Code	X Axis	Z Axis	Work Material
G52	0.	0.	No Material Selected
G54	0.	0.	No Material Selected
G55	0.	0.	No Material Selected
G56	0.	0.	No Material Selected
G57	0.	0.	No Material Selected
G58	0.	0.	No Material Selected
G59	0.	0.	No Material Selected
G154 P1	0.	0.	No Material Selected
G154 P2	0.	0.	No Material Selected
G154 P3	0.	0.	No Material Selected
G154 P4	0.	0.	No Material Selected
G154 P5	0.	0.	No Material Selected
G154 P6	0.	0.	No Material Selected
G154 P7	0.	0.	No Material Selected
G154 P8	0.	0.	No Material Selected
G154 P9	0.	0.	No Material Selected
G154 P10	0.	0.	No Material Selected
G154 P11	0.	0.	No Material Selected
G154 P12	0.	0.	No Material Selected

F1 To view options. Enter A Value **ENTER** Add To Value **F4** Tool Offsets

DISPLAY KEYS

- "G CODE" IS YOUR WORK OFFSET (THERE S NO INPUT NEEDED FOR THIS)
- "X AXIS" IS YOUR WORK ZERO LOCATION IN THE X AXIS AND IN REFERENCE TO THE MACHINE COORDINATE SYSTEM
- "Z AXIS" IS YOU WORK ZERO LOCATION IN THE Z AXIS AND IN REFERENCE TO THE MACHINE COORDINATE SYSTEM
- "WORK MATERIAL" IS THE MATERIAL THAT YOU WILL BUT CUTTING (PRESSING "F1" WILL DISPLAY THE MATERIAL OPTIONS)

Offsets			
Tool	Work	Axes Info	
G Code	X Axis	Z Axis	Work Material
G52	0.	0.	No Material Selected
G54	0.	0.	No Material Selected
G55	0.	0.	No Material Selected
G56	0.	0.	No Material Selected
G57	0.	0.	No Material Selected
G58	0.	0.	No Material Selected
G59	0.	0.	No Material Selected
G154 P1	0.	0.	No Material Selected
G154 P2	0.	0.	No Material Selected
G154 P3	0.	0.	No Material Selected
G154 P4	0.	0.	No Material Selected
G154 P5	0.	0.	No Material Selected
G154 P6	0.	0.	No Material Selected
G154 P7	0.	0.	No Material Selected
G154 P8	0.	0.	No Material Selected
G154 P9	0.	0.	No Material Selected
G154 P10	0.	0.	No Material Selected
G154 P11	0.	0.	No Material Selected
G154 P12	0.	0.	No Material Selected

F1 To view options. Enter A Value **ENTER** Add To Value **F4** Tool Offsets

DISPLAY KEYS

- "CURRENT COMMANDS" WILL BRING UP INFORMATION ABOUT YOUR MACHINE. ALONG WITH THAT, THE ADVANCED TOOL MANAGER (ATM), A CALCULATOR) AND ANY MEDIA FILES CAN BE FOUND HERE



CURRENT COMMANDS

- "DEVICES" IS WHERE YOU CAN SEE ALL OF THE DEVICES CONTROLLED THAT CAN BE CONTROLLED BY THE MACHINE

Current Commands

Devices Timers Macro Vars Active Codes ATM Calculator Media

Mechanisms

Device	State
Probe Arm	Retracted
Main Spindle Chuck	Clamped
Main Spindle Jog	RPM: 100
Main Spindle Orient	278.1862
Main Spindle Brake	Disengaged
Jet Air Blast	off

Probe Arm

F2 Extend

WARNING: Check that the probe arm has room to extend or damage may occur
Press [F2] to lower the probe arm, or to raise it to continue operation

CURRENT COMMANDS

- "TIMERS" IS WHERE YOU CAN VIEW:
 - "DATE"
 - "TIME"
 - "TIME ZONE"
 - "POWER ON TIME" IS THE AMOUNT OF TIME THE MACHINE HAS BEEN TURNED ON
 - "CYCLE START TIME" IS THE AMOUNT OF TIME THE MACHINE WAS RUNNING A PROGRAM
 - "FEED CUTTING TIME" IS THE AMOUNT OF TIME THE MACHINE WAS FEEDING
 - "THIS CYCLE" THE AMOUNT OF TIME THIS CYCLE IS TAKING TO RUN
 - "LAST CYCLE" IS THE AMOUNT OF TIME THE LAST PROGRAM TOOK

Current Commands						
Devices	Timers	Macro Vars	Active Codes	ATM	Calculator	Media
Date:	10-18-2018	Loops Remaining:	0			
Time:	13:17:01	M30 Counter #1:	0			
Time Zone:	PST	M30 Counter #2:	0			
Power On Time:	10:30:19	Macro Label #1:	LABEL 1			
Cycle Start Time:	0:01:36	Macro Assign #1:	0.0			
Feed Cutting Time:	0:01:05	Macro Label #2:	LABEL 2			
This Cycle:	0:01:02	Macro Assign #2:	0.0			
Last Cycle:	0:01:02					

Enter Date In The Format MM-DD-YYYY

Set Value

CURRENT COMMANDS

- "TIMERS" IS WHERE YOU CAN VIEW:
 - "LOOPS REMAINING"
 - "M30 COUNTER" IS THE AMOUNT OF TIMES AN M30 HAS BEEN REACHED
 - "MACRO LABEL ##" IS THE LABEL GIVEN IF YOU WERE TO ASSIGN A MACRO TO DISPLAY A VARIABLE
 - "MACRO ASSIGN ##" IS THE MACRO ASSOCIATED WITH THE LABEL VALUE

Current Commands						
Devices	Timers	Macro Vars	Active Codes	ATM	Calculator	Media
Date:	10-18-2018	Loops Remaining:	0			
Time:	13:17:01	M30 Counter #1:	0			
Time Zone:	PST	M30 Counter #2:	0			
Power On Time:	10:30:19	Macro Label #1:	LABEL 1			
Cycle Start Time:	0:01:36	Macro Assign #1:	0.0			
Feed Cutting Time:	0:01:05	Macro Label #2:	LABEL 2			
This Cycle:	0:01:02	Macro Assign #2:	0.0			
Last Cycle:	0:01:02					

Enter Date In The Format MM-DD-YYYY

Set Value

CURRENT COMMANDS

- "MACRO VARS" DISPLAYS THE LOCAL AND GLOBAL MACRO VARIABLES AVAILABLE

Current Commands						
Timers	Macro Vars	Active Codes	ATM	Tool Table	Calculator	Media
Macro Variables			10400 - 10999 (Global)			
(Local) 1 - 33		(Global) 10000 - 10199		(Global) 10200 - 10399		
Var	Value	Var	Value	Var	Value	
1		10000	0.000000	10200	0.000000	
2		10001	0.000000	10201	0.000000	
3		10002	0.000000	10202	0.000000	
4		10003	0.000000	10203	0.000000	
5		10004	0.000000	10204	0.000000	
6		10005	0.000000	10205	0.000000	
7		10006	0.000000	10206	0.000000	
8		10007	0.000000	10207	0.000000	
9		10008	0.000000	10208	0.000000	
10		10009	0.000000	10209	0.000000	
11		10010	0.000000	10210	0.000000	
12		10011	0.000000	10211	0.000000	
13		10012	0.000000	10212	0.000000	
14		10013	0.000000	10213	0.000000	

*Legacy 3 digit macros begin at 10000 Range. i.e. Macro 100 will be displayed as 10100.

CURRENT COMMANDS

- "ACTIVE CODES" ARE ALL THE CODES THAT ARE CURRENTLY BEING USED ALONG WITH SPEEDS AND FEEDS

Current Commands						
Devices	Timers	Macro Vars	Active Codes	ATM	Calculator	Media
G-Codes		Address Codes		DHMT Codes		Speeds & Feeds
G00	N	0	D	01	Programmed Feed Rate	0. IPR
G18	X	0.	H	00	Actual Feed Rate	0. IPR
G90	Y	0.	M	08	Main Spindle	
G113	Z	0.	T	101	Programmed Speed	1457 RPM
G20	I	0.			Commanded Speed	1457 RPM
G40	J	0.			Actual Speed	0 RPM
G49	K	0.			Direction	Stop
G80	P	0				
G99	Q	0.				
G50	R	0.				
G54	O	054321				
G97	A	0.				
G64	B	0.				
G69	C	0.				
G15	U	0.				
	V	0.				
	W	0.				
	E	0.				

CURRENT COMMANDS

"ATM" (ADVANCED TOOL MANAGER) ALLOWS OPERATOR TO SET UP REDUNDANT TOOLS THAT WILL EXPIRE BASED ON TOOL USAGE PARAMETERS SETUP BY THE OPERATOR

WHEN TOOL LIFE EXPIRES FOR ONE TOOL IN THE GROUP THE CONTROL WILL AUTOMATICALLY GRAB THE NEXT AVAILABLE TOOL IN THE GROUP

Current Commands

Devices Timers Macro Vars Active Codes **ATM** Calculator Media

F4 To Switch Boxes Allowed Limits Active Tool: 1

Group	Expired Count	Tool Order	Holes Limit	Usage Limit	Life Warn %	Expired Action	Feed Limit	Total Time Limit
All	-	-	-	-	-	-	-	-
Expired	0	-	-	-	-	-	-	-
No Group	-	-	-	-	-	-	-	-
Add Group	-	-	-	-	-	-	-	-

Tool Data For Group: All

Tool	Offset	Life	Holes Count	Usage Count	Usage Limit	Feed Time	Total Time
1	0	100%	0	2	0	0:01:03	0:03:00
2	0	100%	0	0	0	0:00:00	0:00:00
3	0	100%	0	0	0	0:00:00	0:00:00
4	0	100%	0	0	0	0:00:00	0:00:00
5	0	100%	0	0	0	0:00:00	0:00:00
6	0	100%	0	0	0	0:00:00	0:00:00

INSERT Add Group

CURRENT COMMANDS

"CALCULATOR" IS YOUR STANDARD CALCULATOR BUILT INTO THE MACHINE

Current Commands

Devices Timers Macro Vars Active Codes ATM **Calculator** Media

Standard Milling/Turning Tapping Arc

7	8	9	+ [D]	+/- [E]	MS [S]
4	5	6	- [J]	sqrt [K]	MR [R]
1	2	3	* [P]	% [Q]	MC [C]
(0)	/ [V]	Enter	
Clear [ORIGIN]		.			

F2 Switch Entry To Input Line

INSERT To append to INPUT line.

ALTER To replace INPUT line.

ORIGIN Reset Calculators

F3 Open Shape Creator

CURRENT COMMANDS

"MILLING/TURNING" IN THE CALCULATOR ALLOWS THE USER TO CALCULATE SPEEDS AND FEEDS, ALONG WITH GIVING SUGGESTED SPEEDS AND FEEDS WHEN SUPPLYING MATERIAL INFORMATION

The screenshot shows the 'Current Commands' window with the 'Calculator' tab selected. The 'Milling/Turning' sub-tab is active. The interface includes several input fields for parameters: Cutter/Part Diameter (in), Surface Speed (ft/min), RPM, Flutes, Feed (in/min), and Chip Load (in/tth). There are also dropdown menus for Work Material (currently 'No Material Selected') and Tool Material (currently 'Please Select Work Material'). On the right side, there are function keys: F2 (Switch Entry To Input Line), INSERT (To append to INPUT line), ALTER (To replace INPUT line), DELETE (Clear current input), and ORIGIN (Reset Calculators). At the bottom, there are F3 (Copy Value From Standard Calculator) and F4 (Paste Current Value To Standard Calculator) keys. A note at the bottom states '* Next to Field Name Denotes Calculated Value'.

CURRENT COMMANDS

"TAPPING" IS A CALCULATOR FOR SPEEDS AND FEEDS ON TAPPING CYCLES

The screenshot shows the 'Current Commands' window with the 'Calculator' tab selected. The 'Tapping' sub-tab is active. The interface includes input fields for TPI (rev/in), Metric Lead (mm/rev), RPM, and Feed (in/min). On the right side, there are function keys: F2 (Switch Entry To Input Line), INSERT (To append to INPUT line), ALTER (To replace INPUT line), DELETE (Clear current input), and ORIGIN (Reset Calculators). At the bottom, there are F3 (Copy Value From Standard Calculator) and F4 (Paste Current Value To Standard Calculator) keys. A note at the bottom states '* Next to Field Name Denotes Calculated Value'.

CURRENT COMMANDS

"RADIUS" ALLOWS YOU TO CALCULATE THE PATH NEEDED TO TAKE TO MAKE A SPECIFIED ARC

Current Commands

Devices Timers Macro Vars Active Codes ATM Calculator Media

Standard Milling/Turning Tapping **Arc**

F2 Switch Entry To Input Line
INSERT To append to INPUT line.
ALTER To replace INPUT line.
DELETE Clear current input
ORIGIN Reset Calculators

F3 Open Shape Creator
F4 Paste Value From Input Line

Radius in
End X in
Arc Center Z in
Arc Center X in
Z1 in
Z2 in

* Next to Field Name Denotes Calculated Value

CURRENT COMMANDS

"MEDIA" IS WHERE YOU CAN LOAD IMAGES AND VIDEOS THAT CAN BE DISPLAYED TO HELP WITH CHANGING PARTS. THIS CAN BE DISPLAYED DURING A PROGRAM WITH AN M130(FILE.XXX) WITH THE FILE'S PATH

THE FILE TYPES THAT CAN BE USED ARE:

- MPEG-2
- MPEG-4
- H.263
- DivX
- BASELINE
- PNG
- JPEG

NOTE: USING A PIXEL DIMENSION DIVISIBLE BY 8 LOAD FASTER

Current Commands

Devices Timers Macro Vars Active Codes ATM Calculator **Media**

Prog Comment:

No Media Loaded

DISPLAY KEYS

- "ALARMS" SHOWS ANY ACTIVE ALARMS WITH A SHORT DESCRIPTION AND POSSIBLE SOLUTION. MOST ALARMS CAN BE FOUND HERE WITH A DESCRIPTION AND POSSIBLE SOLUTION



ALARMS

"ACTIVE ALARMS" WILL TELL YOU ANY ALARMS THAT ARE CURRENTLY ACTIVE ON YOUR MACHINE. THIS PAGE WILL TELL YOU WHAT THE ALARM IS, WHAT MAY BE CAUSING THE ALARM AND HOW THE ALARM COULD BE RESOLVED

A screenshot of a software interface titled 'Alarms And Messages'. It features a navigation bar with tabs for 'Active Alarms', 'Messages', 'Alarm History', 'Alarm Viewer', and 'Key History'. Below the tabs, a table shows one active alarm: '107 EMERGENCY STOP'. Below the table, the alarm is detailed with the title '107 EMERGENCY STOP' and a description: 'An Emergency Stop button was pressed. The spindle and all axes have been decelerated to a stop and power has been removed from the motors. To restore power to the motors and spindle, release all Emergency Stop buttons and press reset.' At the bottom, there are four navigation buttons: 'HOME', 'END', 'PAGE UP', and 'PAGE DOWN', followed by the text 'Scroll Description'.

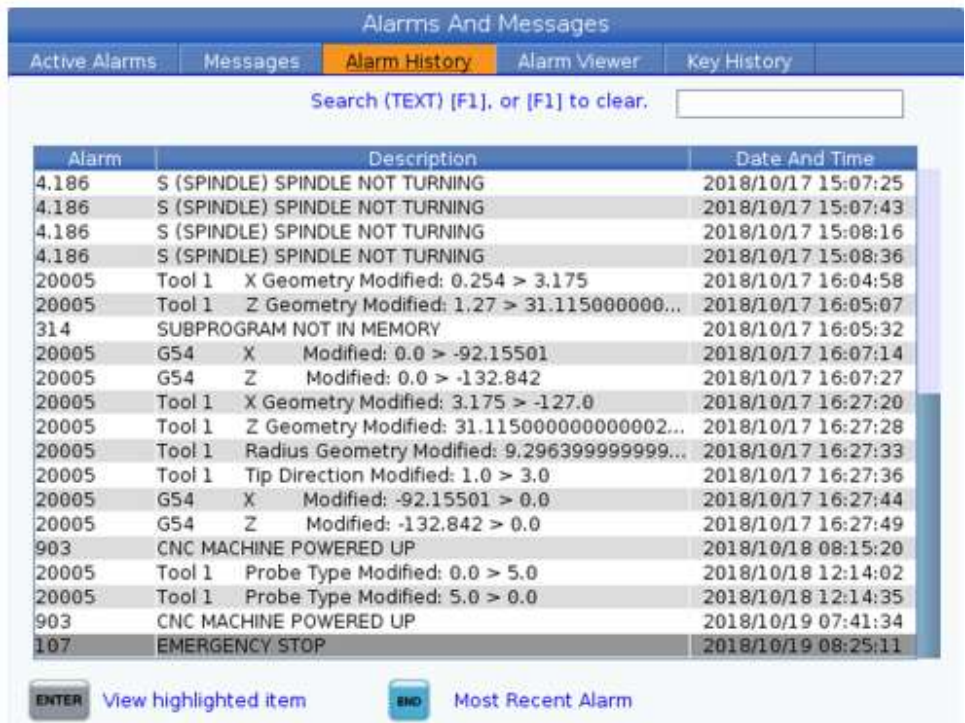
ALARMS

"MESSAGES" CAN BE USED TO LEAVE MESSAGES TO OTHER PEOPLE THAT MAY BE RUNNING THE MACHINE. A MESSAGE CAN BE INPUT BY USING THE ALPHA AND NUMERIC KEYS. ANY MESSAGES PUT IN THIS LOCATION WILL REMAIN HERE UNTIL CLEARED, EVEN IF THE MACHINE IS SHUT OFF



ALARMS

"ALARM HISTORY" WILL SHOW YOU THE ALARMS THAT HAVE RECENTLY AFFECTED YOUR MACHINE



ALARMS

"ALARM VIEWER" WILL ALLOW YOU TO VIEW MOST OF THE POSSIBLE ALARMS, YOU CAN ALSO SEARCH FOR ALARMS HERE AS WELL. ON THIS PAGE YOU CAN GET A DESCRIPTION OF THE ALARM ALONG WITH POSSIBLE SOLUTIONS FOR THAT ALARM

Alarms And Messages

Active Alarms Messages Alarm History **Alarm Viewer** Key History

Search (TEXT) [F1], or [F1] to clear:

Alarm	Name
102	SERVOS TURNED OFF
103	AXIS SERVO ERROR TOO LARGE
107	EMERGENCY STOP
108	AXIS SERVO OVERLOAD
113	SHUTTLE IN FAULT
114	SHUTTLE OUT FAULT
115	CAROUSEL POSITION FAULT
116	SPINDLE ORIENTATION FAULT
119	INPUT AC LINE OVERVOLTAGE
120	LOW AIR PRESSURE OR FLOW
122	REGEN OVERHEAT
123	SPINDLE DRIVE FAULT
125	SHUTTLE FAULT

[HELP](#) To read full description

ALARMS

"KEY HISTORY" IS YOUR LAST 2000 KEY STROKES. THIS CAN BE USED TO HELP FIND ANY ISSUES THAT MAY HAVE OCCURRED IN YOUR MACHINE

Alarms And Messages

Active Alarms Messages Alarm History Alarm Viewer **Key History**

VK_DOWN	2018/10/19	08:41:43
VK_DOWN	2018/10/19	08:41:46
VK_DOWN	2018/10/19	08:41:47
VK_UP	2018/10/19	08:41:48
VK_UP	2018/10/19	08:41:48
VK_DOWN	2018/10/19	08:41:49
VK_DOWN	2018/10/19	08:41:49
VK_DOWN	2018/10/19	08:41:50
VK_DOWN	2018/10/19	08:41:50
VK_DOWN	2018/10/19	08:41:50
VK_DOWN	2018/10/19	08:41:50
VK_DOWN	2018/10/19	08:41:50
VK_DOWN	2018/10/19	08:41:50
VK_HAASHELP	2018/10/19	08:41:52
VK_HAASHELP RELEASED	2018/10/19	08:41:52
VK_ALARM	2018/10/19	08:42:10
VK_ALARM RELEASED	2018/10/19	08:42:10
VK_ALARM	2018/10/19	08:42:11
VK_ALARM RELEASED	2018/10/19	08:42:11
VK_RIGHT	2018/10/19	08:42:12

DISPLAY KEYS

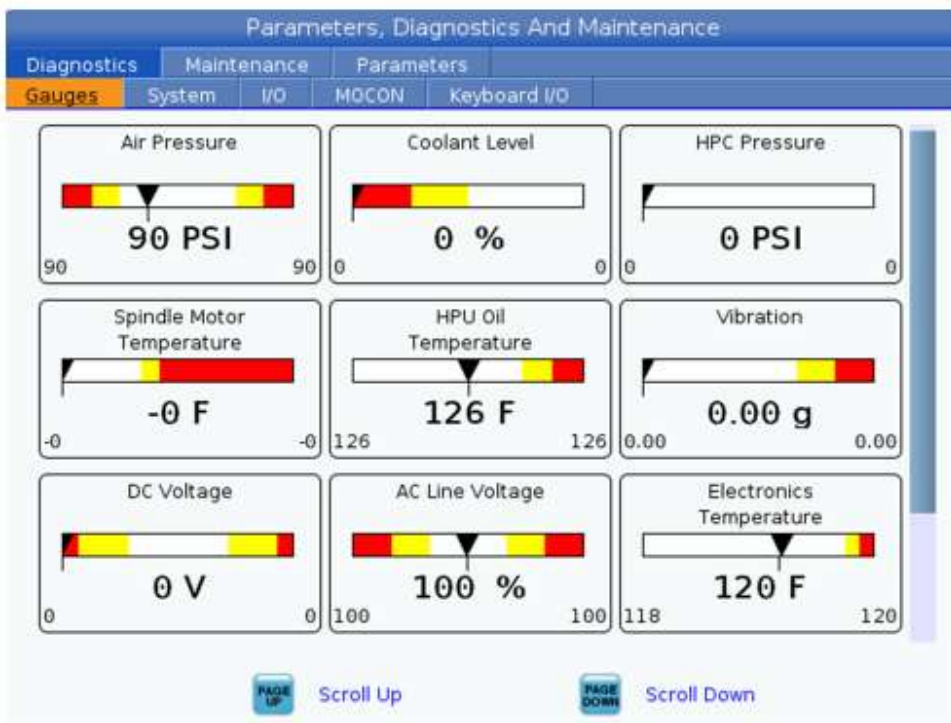
- "DIAGNOSTIC" SHOWS INFORMATION ON YOUR MACHINE'S SYSTEM. FEATURES, COMPENSATION, DIAGNOSTICS AND MAINTENANCE



DIAGNOSTICS

"DIAGNOSTICS" WILL SUPPLY YOU WITH INFORMATION ABOUT YOUR MACHINE SUCH AS:

- "GAUGES" WHICH GIVE YOU INFORMATION INVOLVING YOUR MACHINES AIR AND ELECTRIC
- "SYSTEM" WHICH GIVES YOU INFORMATION ON YOUR MACHINE'S SERIAL NUMBER AND WHAT VERSION OF SOFTWARE IT RUNS. IT ALSO GIVES YOU INFORMATION ON WHAT THE MACHINE HAS DONE WHILE POWERED ON
- "I/O", "MOCON" AND "KEYBOARD I/O" ARE USED TO DISPLAY TO SEE IF COMPONENTS ARE WORKING PROPERLY ON THE CONTROL



DIAGNOSTICS

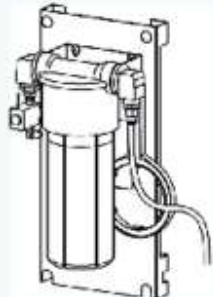
"MAINTENANCE" WILL SUPPLY YOU WITH INFORMATION ABOUT YOU MACHINE SUCH AS:

- "LUBE" WILL ALLOW YOU TO TEST YOUR LUBRICATION SYSTEM
- "COOLANT REFILL" WILL ALLOW YOU INPUT INFORMATION ABOUT REFILLING YOUR COOLANT (USED FOR AUTOMATIC COOLANT REFILL SYSTEMS)
- "SOFTWARE UPDATE" WILL ALLOW YOU TO GET SOFTWARE UPDATES WHEN CONNECTED TO THE INTERNET. IT IS AUTOMATICALLY SET TO DISABLED AND CAN BE SET HOW OFTEN THE MACHINE WILL CHECK

Parameters, Diagnostics And Maintenance

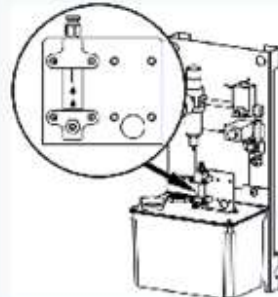
Diagnostics	Maintenance	Parameters
Lube	Coolant Refill	Software Update

AXIS LUBRICATION
Next Lube Cycle In 05:58 (HH:MM)



F2 Axis Lubrication Test

SPINDLE LUBRICATION
Next Lube Cycle In 14 Minutes



F3 Spindle Lubrication Test

DIAGNOSTICS

"PARAMETERS" WILL SUPPLY YOU WITH INFORMATION ABOUT YOU MACHINE SUCH AS:

- "FEATURES" GIVES YOU INFORMATION ON WHAT FEATURES ARE ENABLED ON YOUR HAAS MACHINE, ALONG WITH POSSIBLE FEATURES YOU CAN ADD
- "COMPENSATION" CAN BE USED TO ADJUST THE MACHINE COMPENSATION AS IT TRAVELS ALONG AN AXIS
- "ACTIVATION" IS WHERE YOU CAN FIND YOUR MACHINE SERIAL NUMBER, MAC ADDRESS, SOFTWARE VERSION, TIME TO THE NEXT BLL CYCLE AND YOUR MACHINE GENERATED CODE

Parameters, Diagnostics And Maintenance

Diagnostics	Maintenance	Parameters
Features	Compensation	Activation

Search (TEXT) [F1], or [F1] to clear.

	Feature	Status	Date:
<input checked="" type="checkbox"/>	Machine	Purchased	Acquired 10-17-18
<input checked="" type="checkbox"/>	Macros	Purchased	Acquired 10-19-18
<input type="checkbox"/>	Rotation And Scaling	Tryout Available	
<input checked="" type="checkbox"/>	Rigid Tapping	Purchased	Acquired 10-19-18
<input type="checkbox"/>	TCPC and DWO	Tryout Available	
<input checked="" type="checkbox"/>	M19 Spindle Orient	Purchased	Acquired 10-19-18
<input checked="" type="checkbox"/>	VPS Editing	Purchased	Acquired 10-19-18
<input checked="" type="checkbox"/>	Media Display	Purchased	Acquired 10-19-18
<input checked="" type="checkbox"/>	Max Memory: 1GB	Purchased	Acquired 10-19-18
<input checked="" type="checkbox"/>	Wireless Networking	Purchased	Acquired 10-19-18
<input type="checkbox"/>	Compensation Tables	Feature Disabled	Purchase Required
<input checked="" type="checkbox"/>	High Pressure Coolant	Purchased	Acquired 10-19-18
<input type="checkbox"/>	C Axis	Feature Disabled	Purchase Required
<input checked="" type="checkbox"/>	Max Spindle Speed: 5000 RPM	Purchased	Acquired 10-19-18
<input checked="" type="checkbox"/>	Max Sub Spindle Speed: 0 RPM	Feature Disabled	Purchase Required
<input checked="" type="checkbox"/>	Max LT Spindle Speed: 0 RPM	Feature Disabled	Purchase Required

*Tryout time is only updated while Feature is enabled.

ENTER Turn On/Off Feature

F4 Purchase Feature With Entered Activation Code.

DISPLAY KEYS

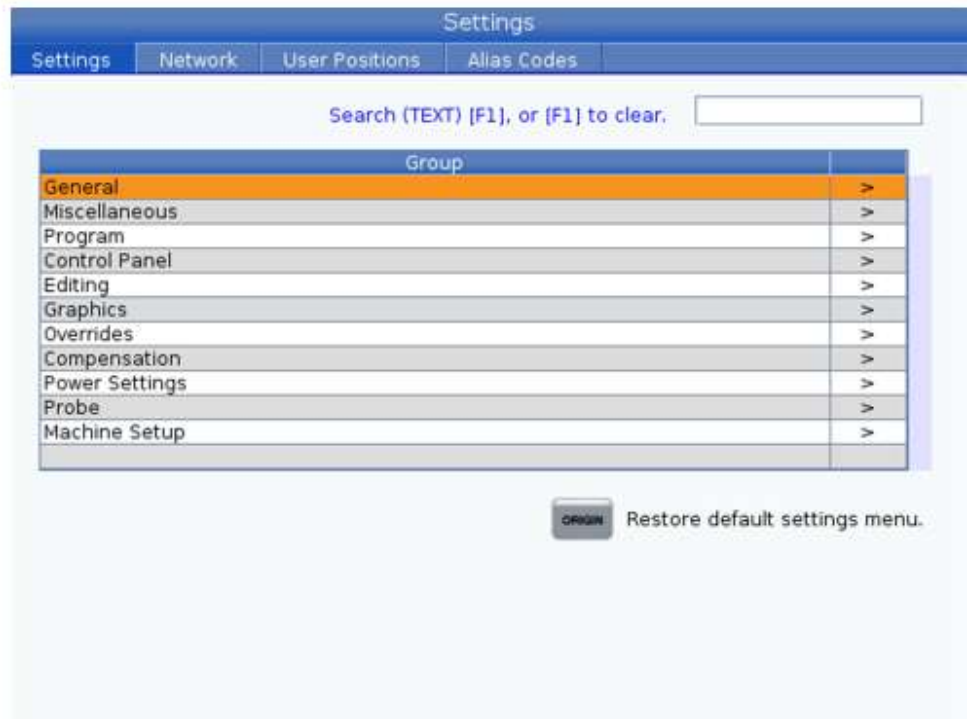
- "SETTING" ALLOWS YOU ADJUST AND VIEW CURRENT SETTINGS IN THE MACHINE



SETTINGS

"SETTINGS" ALLOWS YOU TO ADJUST DIFFERENT SETTINGS IN YOUR MACHINE

- THE SETTINGS ARE ORGANIZED INTO GROUPS TO MAKE THEM EASIER TO FIND
- BY USING TYPING IN A KEY WORD THEN PRESSING "F1" YOU WILL BE ABLE TO SEARCH SETTINGS FOR ANY SETTING WITH THAT WORD IN IT
- BY PRESSING "ORGIN" YOU CAN RESET ALL SETTINGS BACK TO THE FACTORY DEFAULTS



SETTINGS

"NETWORK" ALLOWS YOU TO CONNECT YOUR HAAS MACHINE TO THE INTERNET

- "WIRED CONNECTION" GIVES YOUR INFORMATION FOR CONNECTING YOU MACHINE USING AN ETHERNET CORD
- "WIRELESS CONNECTION" GIVES YOU INFORMATION ON CONNECTING YOUR MACHINE WIRELESSLY
- "NET SHARE" GIVES YOU INFORMATION, IF SET UP, TO CONNECT YOUR MACHINE TO COMPUTERS TO SEND AND RECEIVE FILES
- "HAAS CONNECT" IS A WEB-BASED APPLICATION THAT ALLOWS YOU TO ACCESS YOUR MACHINE'S STATUS REMOTELY THROUGH YOUR PHONE OR COMPUTER

Settings

Settings Network User Positions Alias Codes

Wired Connection Wireless Connection Net Share Haas Connect

Wired Network Information

Host Name	HaasCNC899996	DHCP Server	192.168.10.210
Domain		IP Address	192.168.10.56
DNS Server	192.168.10.210	Subnet Mask	255.255.255.0
Mac Address	00:C0:08:88:50:A0	Gateway	192.168.10.1
DHCP Enabled	Enabled	Status	UP

Name	Value
Wired Network Enabled	On
Obtain Address Automatically	On
IP Address	
Subnet Mask	
Default Gateway	
DNS Server	

Warning: Changes will not be saved if page is left without pressing **F4**

F3 Discard Changes **F4** Apply Changes

SETTINGS

"USER POSITIONS" ALLOWS YOU TO DEFINE DIFFERENT POSITIONS IN YOUR MACHINE SUCH AS WHERE A SAFE PLACE TO CHANGE TOOLS. THIS WOULD BE USED IF YOU HAD A CHANCE OF HITTING A PART OR THE TAILSTOCK

Settings

Settings Network User Positions Alias Codes

Search (TEXT) [F1], or [F1] to clear.

Group	
Safe Tool Change Location	>
Second Home Position	>
Spindle Center Line	>
Tailstock	>
User Travel Limit	>

SETTINGS

"ALIAS CODES" ARE USER DEFINED CODES THAT CAN BE USED TO REFERENCE A PROGRAM FROM A SUB-PROGRAM

The screenshot shows a software interface with a blue header bar labeled 'Settings'. Below the header are four tabs: 'Settings', 'Network', 'User Positions', and 'Alias Codes' (which is highlighted in orange). The main content area contains a table with two columns: 'M-Codes & G-Codes Program Aliases' and 'Value'. The table lists 19 rows of macro call codes, each with a value of 0. At the bottom of the table area, there is a 'Clear all' button.

M-Codes & G-Codes Program Aliases	Value
M MACRO CALL 09000	0
M MACRO CALL 09001	0
M MACRO CALL 09002	0
M MACRO CALL 09003	0
M MACRO CALL 09004	0
M MACRO CALL 09005	0
M MACRO CALL 09006	0
M MACRO CALL 09007	0
G MACRO CALL 09010	0
G MACRO CALL 09013	0
G MACRO CALL 09014	0
G MACRO CALL 09015	0
G MACRO CALL 09016	0
G MACRO CALL 09017	0
G MACRO CALL 09018	0
G MACRO CALL 09019	0

Clear all

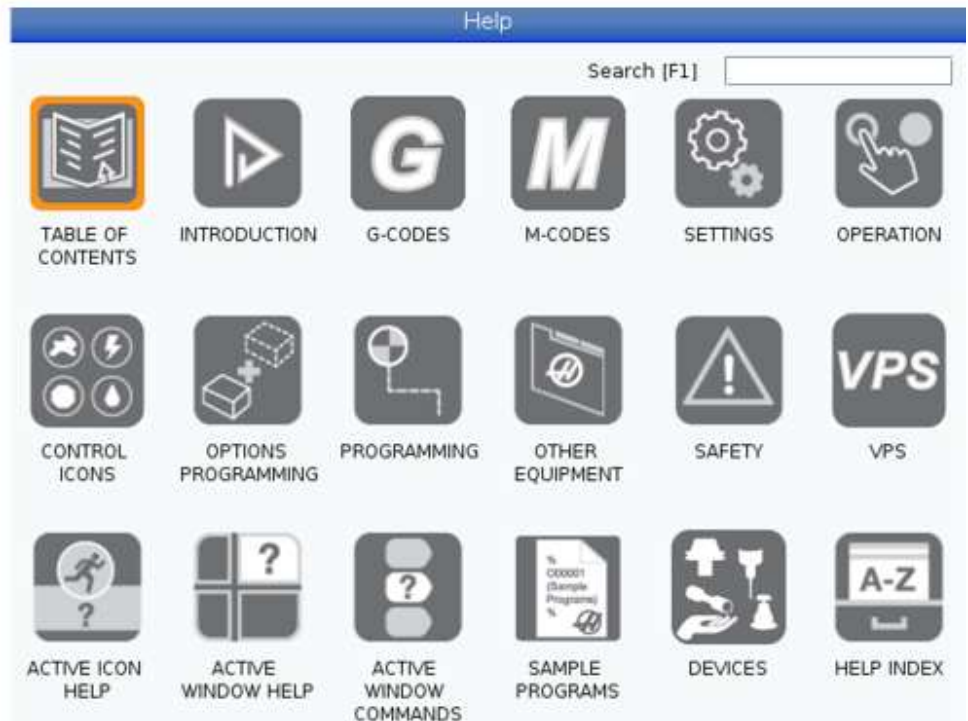
DISPLAY KEYS

- "HELP" IS YOUR DIGITAL USER MANUAL FOR YOUR MACHINE



HELP

- "HELP" IS YOUR DIGITAL MACHINE USER MANUAL. BY INPUTTING TEXT AND PRESSING "F1" YOU CAN SEARCH THROUGH THE MANUAL TO FIND THINGS WITH RELATED PHRASES. "HELP" IS BROKEN DOWN INTO SECTIONS TO MAKE IT EASIER TO FIND INFORMATION AS WELL



FUNCTION KEYS

- FUNCTION KEYS ARE USED THROUGHOUT ALL OF THE MODES AND DISPLAYS ON THE MACHINE. THEY WILL ALLOW YOU TO DO THINGS SUCH AS INSERT INFORMATION OR COPY PROGRAMS DEPENDING ON WHAT SCREEN YOU ARE ON



FUNCTION KEYS

- "RESET" CLEARS ANY ALARMS AND BRINGS YOU BACK TO THE TOP OF A PROGRAM
- "POWER UP" WILL HOME YOUR MACHINE AT START-UP
- "RECOVERY" HELPS RECOVER FROM AN INCOMPLETE MOVE
- "F1-F4" HAVE VARIOUS FUNCTION BASED ON THE SCREEN YOU ARE ON



FUNCTION KEYS

- "X DIAMETER MEASURE" USES YOUR CURRENT POSITION TO MEASURE AND SET YOUR OFFSET IN THE X-AXIS
- "NEXT TOOL" SELECTS THE NEXT TOOL IN THE TURRET
- "X/Z" TOGGLES BETWEEN THE X AND Z-AXIS JOG MODES
- "Z FACE MEASURE" USES YOUR CURRENT POSITION TO MEASURE AND SET THE OFFSET IN THE Z-AXIS



TYPES OF TOOLING

EXTERNAL TURNING

ROUGHING, PROFILING AND FACING

FINISHING TOOL



GROOVING, CUT OFF AND THREADING

GROOVING TOOL

CUT OFF TOOL

THREADING TOOL



INTERNAL TURNING

INTERNAL TURNING



INTERNAL GROOVING



INTERNAL THREADING



FACE TURNING

FACE GROOVING



FACE TURNING



PROGRAMMING

PROGRAMMING LETTER GUIDE

O	Program Number	O00001-O99999	O9000-O9999 are generally reserved for hidden programs like Macros
N	Sequence/Line Number	N0001-N99999	This is only used to help identify program lines. Used with special codes as well
G	Preparatory Codes	G00-G99	Tells machine how to get from point A to point B
X/Z	Destination Coordinates	X -999.9999 - X999.9999	Tells the machine the destination coordinates of the tool
R	Radius of an Arc	R000.0001-R999.9999	Tells the machine to follow an ARC Radius when cutting partial arcs
M	Miscellaneous/ Machine Codes	M00-M99	Tells machine to turn on or off certain machine functions
S	Spindle Speed	S(Min RPM)-S(Max RPM)	Tells machine was RPM to spin at
T	Tool Number	T0101 - T9999	T calls up a tool, the first two numerical digits call the tool number and the last two digits call the tool offset
F	Feed Rate in Inches per revolution	F0.0001-F(Max feed rate)	Tells machine how fast to move based on the spindle's rotation speed
	Feed Rate in Inches per minute	F0.01-F(Max feed rate)	Tells machine how fast to move

G-CODES

G00	Rapid traverse motion, used for moving from position to position quickly (Moves at machines fastest rapid motion)
G01	A linear motion from one point to another. The speed of this motion can be dictated by feed rate
G02	Clockwise circular interpolation
G03	Counterclockwise circular interpolation
G28	Machine Home (Moves at rapid feed rate) *ENSURE THAT THERE IS NOTHING THAT CAN BE HIT BEFORE USING THIS
G40	Cancel tool nose compensation
G41	Tool nose compensation LEFT of the programmed path
G42	Tool nose compensation RIGHT of the programmed path
G50	Sets limit for the spindle speed max RMP
G70	Finishing Cycle
G71	O.D./I.D. stock removal cycle
G72	End face stock removal cycle
G76	O.D./I.D. thread cutting cycle
G80	Cancel Canned Cycle
G81	Drill canned cycle
G82	Spot drill canned cycle
G83	Peck drill canned cycle
G84	Tapping canned cycle
G96	Constant surface speed on
G97	Cancel constant surface speed
G98	Feed per Minute
G99	Feed per Revolution

*A more extensive list can be found on, behind or in your HAAS Pendant

M-CODES

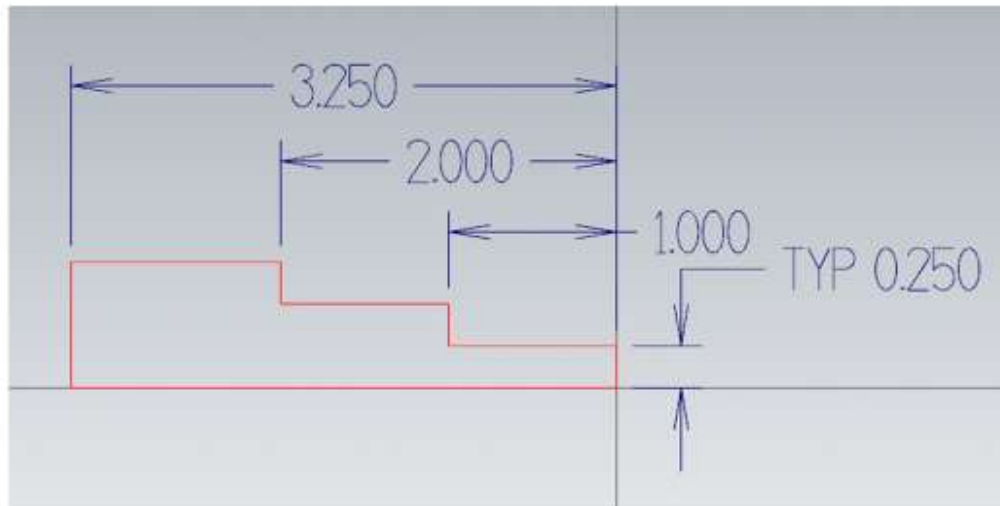
M00	This command will stop the program, machine's spindle, turns off coolant and the lockhead processing, You can continue the program by pressing "Cycle Start"
M01	Allows the operator to use the Optional Program stop function. When put into a program with optional stop on will cause the machine to stop the same as M00
M03	Starts the spindle FORWARD when a spindle speed has been defined
M04	Starts the spindle BACKWARD when a spindle speed has been defined
M05	STOPS the spindle
M08	Coolant ON
M09	Coolant OFF
M30	Program end and resets program to the beginning
M97	Local Subroutine Call
M98	Subroutine Call
M99	Subprogram Return (M98 or M97) or a program loop

- One M code can be used per line
- They will be read as the last code no matter where they are on in the line

*A more extensive list can be found on, behind or in your HAAS Pendant

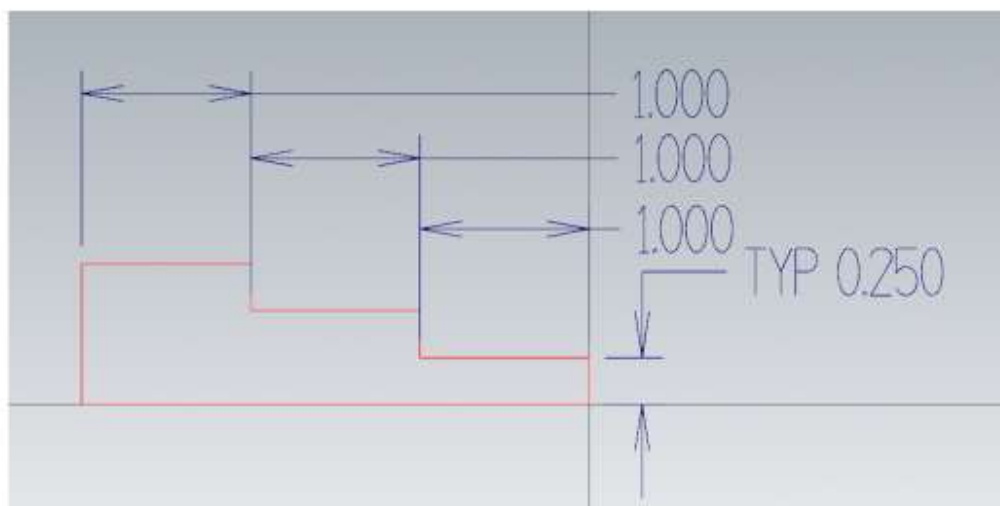
ABSOLUTE VS. INCREMENTAL PROGRAMMING

- ABSOLUTE PROGRAMMING USES A SPECIFIC DATUM TO REFERENCE FROM
 - THIS DATUM IS YOU WORK OFFSETS ZERO



ABSOLUTE VS. INCREMENTAL PROGRAMMING

- INCREMENTAL PROGRAMMING USES THAT LAST MOVE AS THE REFERENCE POINT



VISUAL PROGRAMMING SYSTEM (VPS)

- VPS CAN BE ACCESSED BY HITTING THE "EDIT" KEY AND TABBING OVER TO VPS
- PROBING AND VPS CAN BE FOUND IN THE VPS TAB

TOOL SETTING

- TOOL SETTING ALLOWS YOU TO QUICKLY TOUCH OFF YOUR TOOLS
- BEFORE DOING AN AUTOMATIC TOOL SET THE TOOL SHOULD MANUAL SET THE TO TOOL SETTER
- WHEN USING THE TOOL SETTER YOU MUST INPUT THE OFFSET FROM THE PROBE TO THE FACE OF YOUR PART

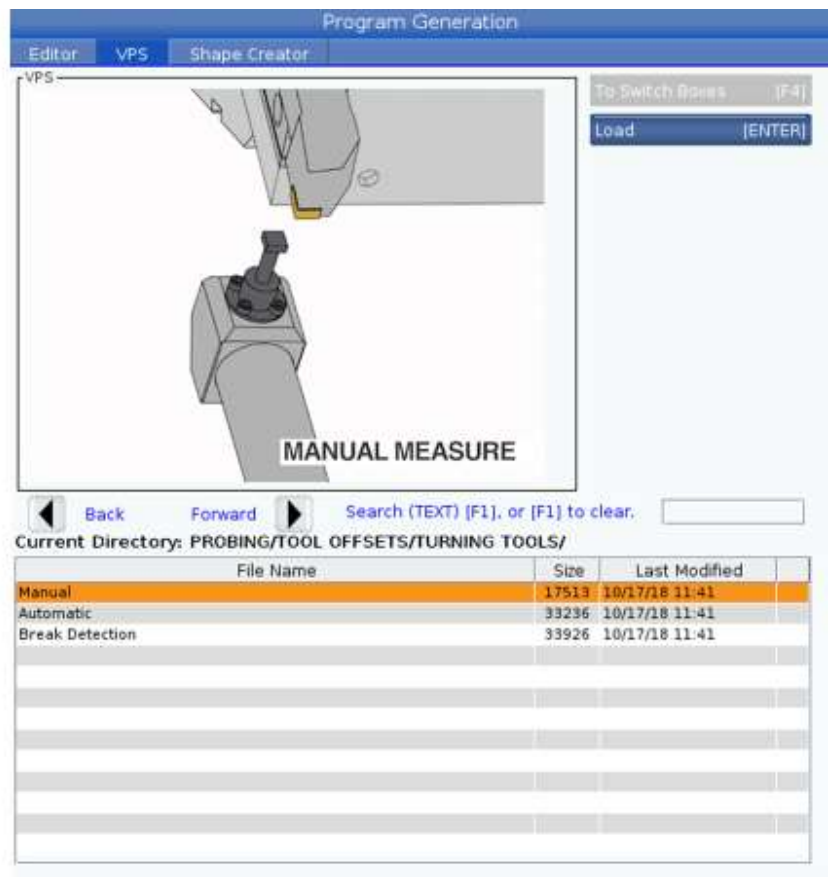
PROBING

TO USE THE AUTOMATIC TOOL SETTER TO SET TOOL OFFSETS YOU CAN PRESS "EDIT" THEN GO OVER TO THE "VPS" TAB. FROM HERE YOU WILL GO TO "PROBING" AND CURSER TO THE RIGHT. THEN YOU WILL CURSE TO THE RIGHT AGAIN ON "TOOL OFFSETS". HERE YOU CAN SELECT WHAT KIND OF TOOL YOU WILL BE PROBING



PROBING

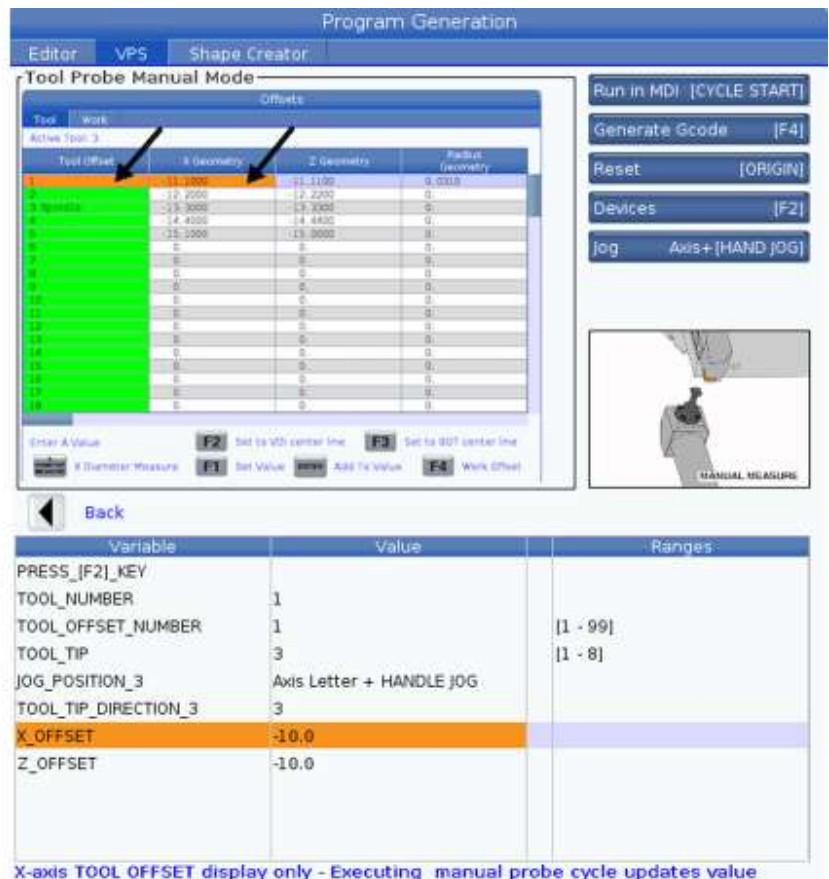
IN "TURNING TOOLS" YOU WILL HAVE THE OPTION FOR "MANUAL", "AUTOMATIC" AND "BREAK DETECTION"



PROBING

"MANUAL" WILL ALLOW YOU TOUCH OFF YOUR TOOL AFTER YOU HAVE JOGGED IT CLOSE TO THE TOOL SETTER. ONCE YOU SELECT "MANUAL" YOU SHOULD MAKE SURE THERE IS NO OBSTRUCTIONS AND PRESS "F2" TO LOWER THE TOOL PROBE. THEN YOU WILL HAVE TO INPUT:

- "TOOL_NUMBER" IS THE TOOL NUMBER
- "TOOL_OFFSET_NUMBER" IS THE OFFSET NUMBER
- "TOOL_TIP" IS THE DIRECTION OF THE TOOL TIP
- JOG THE TOOL TO THE DESIRED POSITION NEXT TO THE TOOL SETTER
- "TOOL_TIP-DIRECTION_#" IS THE DIRECTION THE TIP WILL BE PROBED
- "X_OFFSET" THE DISTANCE OF THE TIP OF YOUR TOOL, FROM HOME, TO CENTERLINE (THIS IS SET IN THE TOOL OFFSETS PAGE)
- "Z_OFFSET" THE DISTANCE FROM THE TIP OF YOUR TOOL, FROM HOME TO THE TOOL SETTER (THIS IS SET IN THE TOOL OFFSETS PAGE)



X-axis TOOL OFFSET display only - Executing manual probe cycle updates value

PROBING

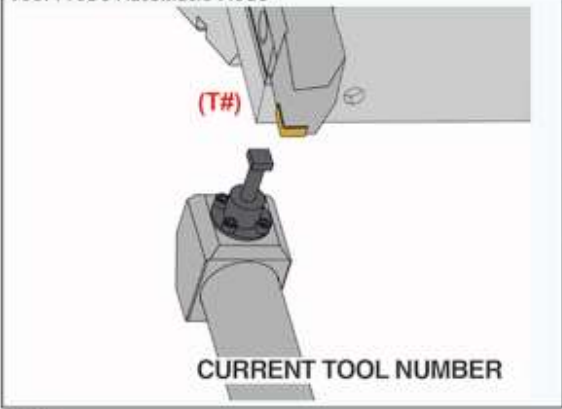
"AUTOMATIC" WILL ALLOW YOU TO TOUCH OFF TOOLS AUTOMATICALLY WHEN YOU INPUT:

- "TOOL_NUMBER" IS THE TOOL NUMBER
- "TOOL_OFFSET_NUMBER" IS THE OFFSET NUMBER
- "CALIBRATION_DIRECTION" IS THE SIDE OF THE PROBED YOU'D LIKE TO USE
- "TOOL_TYPE_...." IS THE TYPE OF TOOL (THIS IS SET IN THE TOOL OFFSETS PAGE)
- "X_OFFSET" THE DISTANCE OF THE TIP OF YOUR TOOL, FROM HOME, TO CENTERLINE (THIS IS SET IN THE TOOL OFFSETS PAGE)
- "Z_OFFSET" THE DISTANCE FROM THE TIP OF YOUR TOOL, FROM HOME TO THE TOOL SETTER (THIS IS SET IN THE TOOL OFFSETS PAGE)


Program Generation

Editor VPS Shape Creator

Tool Probe Automatic Mode



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



Variable	Value	Ranges
TOOL_NUMBER	1	[1 - 12]
TOOL_OFFSET_NUMBER	1	[1 - 99]
CALIBRATION_DIRECTION_3	3	
TOOL_TYPE_OD_TURN	OD_TURN	
X_OFFSET	-10.0	
Z_OFFSET	-10.0	

DEFAULT is current active TOOL NUMBER. Index turret to desired to measure

PROBING

"BREAK DETECTION" WILL ALLOW YOU TO CHECK A PREVIOUSLY CHECKED TOOL FOR WEAR AND BREAKAGE. YOU WILL HAVE TO INPUT:

- "TOOL_NUMBER" IS THE TOOL YOU WILL BE PROBING
- "TOOL_OFFSET_NUMBER" IS WHAT TOOL OFFSET YOU USED FOR THIS TOOL
- "CALIRATION_DIRECTION" IS THE DIRECTION YOU WILL BE PROBLED ON
- "TOOL_TYPE_..." IS THE TYPE OF TOOL (THIS IS SET IN THE TOOL OFFSETS PAGE)

Variable	Value	Ranges
TOOL_NUMBER	1	[1 - 12]
TOOL_OFFSET_NUMBER	1	[1 - 99]
CALIBRATION_DIRECTION_3	3	
TOOL_TYPE_NONE	NONE	
WEAR_TOLERANCE	0.005	[0.0 - 0.1]
X_OFFSET	-10.0	
Z_OFFSET	-10.0	

Enter a wear tolerance to determine if the tool is broken

PROBING

"BREAK DETECTION" WILL ALLOW YOU TO CHECK A PREVIOUSLY CHECKED TOOL FOR WEAR AND BREAKAGE. YOU WILL HAVE TO INPUT:

- "WEAR_TOLERANCE" IS THE DIFFERENCE THE MEASUREMENT CAN HAVE FROM THE SAVED OFFSET AND THE MEASUREMENT THAT IS BEING TAKEN
- "X_OFFSET" THE DISTANCE OF THE TIP OF YOUR TOOL, FROM HOME, TO CENTERLINE (THIS IS SET IN THE TOOL OFFSETS PAGE)
- "Z_OFFSET" THE DISTANCE FROM THE TIP OF YOUR TOOL, FROM HOME TO THE TOOL SETTER (THIS IS SET IN THE TOOL OFFSETS PAGE)

Variable	Value	Ranges
TOOL_NUMBER	1	[1 - 12]
TOOL_OFFSET_NUMBER	1	[1 - 99]
CALIBRATION_DIRECTION_3	3	
TOOL_TYPE_NONE	NONE	
WEAR_TOLERANCE	0.005	[0.0 - 0.1]
X_OFFSET	-10.0	
Z_OFFSET	-10.0	

Enter a wear tolerance to determine if the tool is broken

PROBING

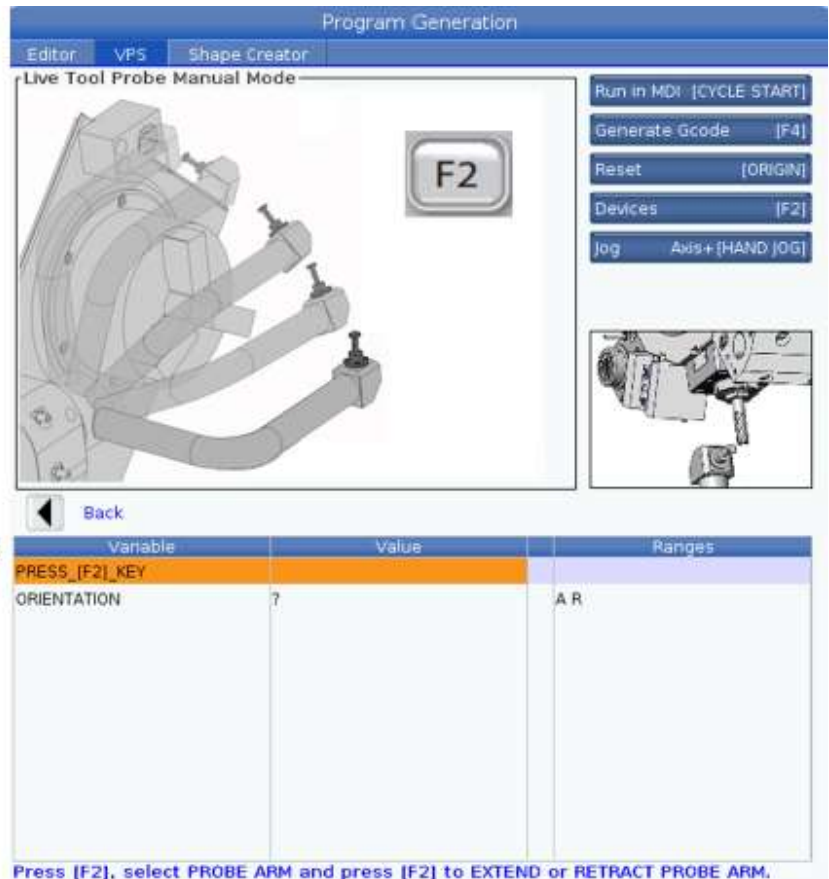
IF YOU RETURN TO THIS PAGE YOU CAN GO DOWN A SELECTION AND CHOOSE TO PROBE LIVE TOOLS. ONCE HERE YOU HAVE THE OPTION TO DO A "MANUAL" TOUCH OFF



PROBING

IN "LIVE TOOL PROBE MANUAL MODE" YOU CAN TOUCH OFF YOUR LIVE TOOLS. TO BEGIN THIS PROCESS YOU NEED TO MAKE SURE THE SPACE IS CLEAR AND THEN PRESS "F2" TO LOWER THE TOOL SETTER. AFTER YOU WILL HAVE TO INPUT:

- "ORIENTATION" IS THE DIRECTION THE LIVE TOOL IS FACING (AXIAL = A OR RADIAL = R)



Press [F2], select PROBE ARM and press [F2] to EXTEND or RETRACT PROBE ARM.

SHAPE CREATOR

- THE SHAPE CREATOR CAN BE USED TO CAN BE USED TO MAKE QUICK O.D. PROGRAM
- BY INPUTTING EACH POINT YOU WANT TO GO TO YOU CAN CREATE A SHAPE USING LINEAR MOVES, CW ARCS AND CCW ARCS WITH FILLETS AND CHAMFERS

SHAPE CREATOR

IN THE "SHAPE CREATOR" YOU CAN CREATE A 2D MODEL USING X AND Z POSITIONS AND ANGLES. YOU WILL ALSO HAVE THE OPTION TO PUT RADII AND CHAMFERS ON EACH POSITION

1. THE FIRST THING THAT NEEDS TO INPUT IS THE "RAW DIMENSIONS" OF THE STOCK MATERIAL
2. AFTER YOU WILL HAVE TO INPUT THE RAPID POINTS
3. AFTER CHOOSING A START POSITION YOU CAN INPUT POINTS THAT CAN MAKE A LINEAR MOVEMENT, A CLOCKWISE CIRCULAR MOTION AND A COUNTER CLOCK-WISE CIRCULAR MOTION

Program Generation

Editor VPS Shape Creator

File Name: TESTTER.scp

V Raw Dimensions	X Position	1.0000
	Z Position	0.0000
V Rapid Point	X Position	1.1000
	Z Position	0.1000
V Start	X Position	0.0000
	Z Position	0.0250
V 1: Linear Feed	X Position	0.0000
	Z Position	0.0250
	Angle	0.000
	Chamfer	0.0000
	Round	0.0000
V 2: Linear Feed	X Position	0.0000
	Z Position	0.0000
	Angle	180.000
	Chamfer	0.0000
	Round	0.0000
V 3: Linear Feed	X Position	0.2500
	Z Position	0.0000
	Angle	90.000

Jog Rate: .1

ENTER Set Value

INSERT Add Motion

ALTER Alter Motion Type

DELETE Delete Motion

F4 Save File

F2 Enable Zoom

F3 Open the calculator.

ORIGIN New shape; discard changes

Press left arrow to collapse row, or right arrow to expand.

SHAPE CREATOR

SOME OF THE THINGS THAT YOU CAN DO WITH THIS IS BY PRESSING:

- "ENTER" WILL SET THE VALUE FROM YOUR INPUT BAR
- "INSERT" WILL ADD A NEW MOTION
- "ALTER" WILL ALLOW YOU TO CHANGE A MOTION THAT HAS ALREADY BEEN CREATED
- "DELETE" WILL DELETE A MOTION
- "F4" WILL SAVE THE SHAPE YOU HAVE CREATED
- "F2" WILL ALLOW YOU TO CHANGE THE ZOOM ON THE PART
- "F3" WILL BRING YOU TO THE CALCULATOR
- "ORIGIN" WILL ALLOW YOU TO CREATE A NEW SHAPE (NOTE: IF THE SHAPE ISN'T SAVED IT WILL BE DELETED WHEN PRESSING ORIGIN)

Program Generation

Editor VPS Shape Creator

File Name: TESTTER.scp

V Raw Dimensions	X Position	1.0000
	Z Position	0.0000
V Rapid Point	X Position	1.1000
	Z Position	0.1000
V Start	X Position	0.0000
	Z Position	0.0250
V 1: Linear Feed	X Position	0.0000
	Z Position	0.0250
	Angle	0.000
	Chamfer	0.0000
	Round	0.0000
V 2: Linear Feed	X Position	0.0000
	Z Position	0.0000
	Angle	180.000
	Chamfer	0.0000
	Round	0.0000
V 3: Linear Feed	X Position	0.2500
	Z Position	0.0000
	Angle	90.000

Jog Rate: .1

ENTER Set Value

INSERT Add Motion

ALTER Alter Motion Type

DELETE Delete Motion

F4 Save File

F2 Enable Zoom

F3 Open the calculator.

ORIGIN New shape; discard changes

Press left arrow to collapse row, or right arrow to expand.

VPS

- IS USED TO CREATE QUICK PROGRAMS WHILE AT THE MACHINE WITHOUT AN EXTERNAL CAD/CAM PROGRAM
- THIS CAN ALSO BE USED IN ADJACENT WITH THE "SHAPE CREATOR"

VPS

- "FACE TURN" WILL ALLOW YOU CREATE A FACING OPERATION ON YOUR PART
- "OD TURN" WILL ALLOW YOU CREATE AN OUTER DIAMETER OPERATIONS, YOU CAN USE THE SHAPE CREATOR TO QUICKLY USE GENERATE THE CODE WITH THE INPUT OF DESIRED SPEEDS, FEEDS AND TOOL NUMBER
- "ID TURN" WILL ALLOW YOU TO CREATE AN INNER DIAMETER OPERATIONS
- "OD THREADING" WILL ALLOW YOU TO CREATE AN OUTER DIAMETER THREADING OPERATIONS
- "ID THREADING" WILL ALLOW YOU TO CREATE INNER DIAMETER THREADING OPERATIONS
- "DRILLING" WILL ALLOW YOU TO CREATE A DRILLING OPERATIONS

File Name	Size	Last Modified
Face Turn	35949	10/17/18 11:41
OD Turn	54709	10/17/18 11:41
ID Turn	54900	10/17/18 11:41
OD Thread	48054	10/17/18 11:41
ID Thread	47329	10/17/18 11:41
Drilling	28345	10/17/18 11:41
OD Groove	109935	10/17/18 11:41
ID Groove	110114	10/17/18 11:41
OD Radius	54998	10/17/18 11:41
ID Radius	55027	10/17/18 11:41
OD Chamfer	67503	10/17/18 11:41
ID Chamfer	67668	10/17/18 11:41
Part Off	56334	10/17/18 11:41

VPS

- "OD GROOVE" WILL ALLOW YOU TO CREATE AN OUTER DIAMETER GROOVING PROCESS
- "ID GROOVE" WILL ALLOW YOU TO CREATE AN INNER DIAMETER GROOVING PROCESS
- "OD RADIUS" WILL ALLOW YOU TO FILLET OR AN ARC ON AN OUTER DIAMETER
- "ID RADIUS" WILL ALLOW YOU TO FILLET OR CREATE AND ARC ON AN INNER DIAMETER
- "OD CHAMFER" WILL ALLOW YOU TO CREATE A CHAMFER ON AN OUTER DIAMETER
- "ID CHAMFER" WILL ALLOW YOU TO CREATE A CHAMFER ON AN INNER DIAMETER
- "PART OFF" WILL ALLOW YOU TO CREATE AN OPERATION TO CUT OFF A PART

File Name	Size	Last Modified
Face Turn	35949	10/17/18 11:41
OD Turn	54709	10/17/18 11:41
ID Turn	54900	10/17/18 11:41
OD Thread	48054	10/17/18 11:41
ID Thread	47329	10/17/18 11:41
Drilling	28345	10/17/18 11:41
OD Groove	109935	10/17/18 11:41
ID Groove	110114	10/17/18 11:41
OD Radius	54998	10/17/18 11:41
ID Radius	55027	10/17/18 11:41
OD Chamfer	67503	10/17/18 11:41
ID Chamfer	67668	10/17/18 11:41
Part Off	56334	10/17/18 11:41


VPS

- "OD PROFILE REMOVAL CYCLES" ARE CYCLES THAT ALLOW YOU TO CREATE QUICK OUTER DIAMETER REMOVAL
- "ID PROFILE REMOVAL CYCLES" ARE CYCLES THAT ALLOW YOU TO CREATE QUICK INNER DIAMETER REMOVAL
- "OD THREAD REPAIR" CAN BE USED TO FIX OUTER DIAMETER THREADS
- "ID THREAD REPAIR" CAN BE USED TO FIX INNER DIAMETER THREADS
- "CENTER DRILL" IS USED TO CENTER DRILL OPERATION
- "TAPPING" IS USED FOR CREATING A TAPPING OPERATIONS

Program Generation

Editor VPS Shape Creator

VPS



To Switch Boxes [F4]
Load [ENTER]

OD ROUGH AND FINISH CYCLES

◀ Back Forward ▶ Search (TEXT) [F1], or [F1] to clear.

Current Directory: VPS/

File Name	Size	Last Modified
OD Profile Removal Cycles	36710	10/17/18 11:41
ID Profile Removal Cycles	40416	10/17/18 11:41
OD Thread Repair	38511	10/17/18 11:41
ID Thread Repair	41250	10/17/18 11:41
Center Drill	23521	10/17/18 11:41
Tapping	35614	10/17/18 11:41

VPS

MANY OF THESE OPERATIONS REQUIRE SIMILAR INFORMATION INCLUDING:

- "TOOL_NUMBER" IS YOUR TOOL NUMBER
- "TOOL_OFFSET_NUMBER" IS THE OFFSET YOU'D LIKE TO SAVE THIS IN
- "WORK_OFFSET_NUMBER" IS THE WORK OFFSET YOU'D LIKE TO USE
- "MAXIMUM_SPINDLE_RPM" WILL ALLOW YOU TO SET A MAX RPM TO LIMIT THE RPM DURING THE CYCLE
- "SURFACE_SPEED_MINUTE" IS THE SURFACE SPEED YOU'D LIKE TO CUT AT

The screenshot shows the 'Program Generation' window with the 'VPS' tab selected. It displays a 3D model of a lathe part and a tool. On the right, there are control buttons: 'Run in MDI [CYCLE START]', 'Generate Gcode [F4]', 'Reset [ORIGIN]', 'Devices [F2]', and 'Jog Axis+ [HAND JOG]'. Below the model is a 'Back' button. At the bottom, there is a table of variables and their values.

Variable	Value	Ranges
TOOL_NUMBER	1	[1 - 12]
TOOL_OFFSET_NUMBER	1	[1 - 99]
WORK_OFFSET_NUMBER	54	[54 - 59]
MAXIMUM_SPINDLE_RPM	1800	[1 - 5000.0]
SURFACE_SPEED_MINUTE	500	[50 - 2000]
Z_RAPID_APPROACH	0.2	[0.01 - 5.0]
X_RAPID_APPROACH	0.2	[0.01 - 5.0]
FLOOD_COOLANT	8	8 9
HPC_COOLANT	89	88 89
OD_DIAMETER	0.0	[0.0 - 9.12]
FINISH_DIAMETER	0.0	[0.0 - 0.0]

The current active TOOL NUMBER is default

VPS

MANY OF THESE OPERATIONS REQUIRE SIMILAR INFORMATION INCLUDING:

- "Z_RAPID_APPROACH" IS THE DISTANCE, IN THE Z AXIS, FROM THE PART YOU WOULD LIKE TO RAPID TO
- "X_RAPID_APPROACH" IS THE DISTANCE, IN THE X AXIS, FROM THE PART YOU WOULD LIKE TO RAPID TO
- "FLOOD_COOLANT" IS WHETHER YOU WANT FLOOD COOLANT ON OR OFF
- "HPC_COOLANT" IS WHETHER YOU WANT HIGH PRESSURE COOLANT ON OR OFF
- "OD_DIAMETER" IS THE OUTER DIAMETER OF THE STOCK MATERIAL
- "FINISH_DIAMETER" IS THE OUTER DIAMETER OF THE FINISHED PART

This screenshot is identical to the one above, showing the 'Program Generation' window with the 'VPS' tab. It displays a 3D model of a lathe part and a tool. On the right, there are control buttons: 'Run in MDI [CYCLE START]', 'Generate Gcode [F4]', 'Reset [ORIGIN]', 'Devices [F2]', and 'Jog Axis+ [HAND JOG]'. Below the model is a 'Back' button. At the bottom, there is a table of variables and their values.

Variable	Value	Ranges
TOOL_NUMBER	1	[1 - 12]
TOOL_OFFSET_NUMBER	1	[1 - 99]
WORK_OFFSET_NUMBER	54	[54 - 59]
MAXIMUM_SPINDLE_RPM	1800	[1 - 5000.0]
SURFACE_SPEED_MINUTE	500	[50 - 2000]
Z_RAPID_APPROACH	0.2	[0.01 - 5.0]
X_RAPID_APPROACH	0.2	[0.01 - 5.0]
FLOOD_COOLANT	8	8 9
HPC_COOLANT	89	88 89
OD_DIAMETER	0.0	[0.0 - 9.12]
FINISH_DIAMETER	0.0	[0.0 - 0.0]

The current active TOOL NUMBER is default

VPS

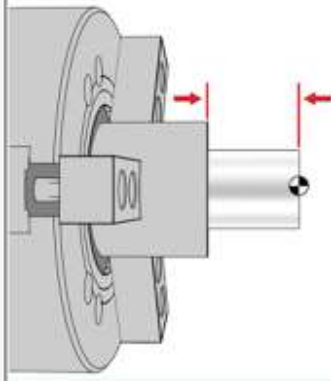
MANY OF THESE OPERATIONS REQUIRE SIMILAR INFORMATION INCLUDING:

- "Z_CUT_LENGTH_G54" IS THE DISTANCE YOU'D LIKE TO CUT IN THE Z AXIS
- "FILLET_RADIUS" IS THE RADIUS OF FILLET, IF DESIRED
- "FEEDRATE_ROUGH" IS THE FEED RATE WHEN DOING THE ROUGHING PASS
- "FINISH_PASS" IS WHETHER YOU WANT A FINISH PASS OR NOT (70 = NO FINISH AND 71 = FINISH PASS)
- "X_FINISH_STOCK" IS THE AMOUNT OF MATERIAL YOU'D LIKE TO LEAVE IN THE X AXIS
- "Z_FINISH_STOCK" IS THE AMOUNT OF MATERIAL YOU'D LIKE TO LEAVE IN THE Z AXIS

Program Generation

Editor VPS Shape Creator

OD TURN CYCLE



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

Back

Variable	Value	Ranges
Z_CUT_LENGTH_G54	0.0	[0.0 - 11.2]
FILLET_RADIUS	0.0	[0.0 - 0.0]
DEPTH_OF_CUT	0.05	[0.0 - 0.0]
FEEDRATE_ROUGH	0.01	[0.001 - 0.05]
FINISH_PASS	71	70 71
X_FINISH_STOCK	0.01	[0.00 - 0.05]
Z_FINISH_STOCK	0.003	[0.00 - 0.05]
RETRACT_X_HOME	Y	Y N
RETRACT_Z_HOME	Y	Y N
END_M_CODE	1	0 1 30

Enter the Z-AXIS CUT LENGTH

VPS

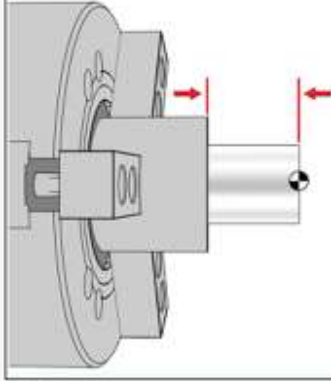
MANY OF THESE OPERATIONS REQUIRE SIMILAR INFORMATION INCLUDING:

- "RETRACT_X_HOME" IS WHETHER YOU WANT THE MACHINE TO RETURN TO HOME IN THE X AXIS ONCE THE CYCLE IS DONE
- "RETRACT_Z_HOME" IS WHETHER YOU WANT THE MACHINE TO RETURN TO HOME IN THE Z AXIS ONCE THE CYCLE IS DONE
- "END_M_CODE" IS WHAT KIND OF M-CODE YOU'D LIKE AT THE END OF THE CYCLE (0 = STOP, 1 = OPTIONAL STOP AND 30 = END PROGRAM)

Program Generation

Editor VPS Shape Creator

OD TURN CYCLE



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

Back

Variable	Value	Ranges
Z_CUT_LENGTH_G54	0.0	[0.0 - 11.2]
FILLET_RADIUS	0.0	[0.0 - 0.0]
DEPTH_OF_CUT	0.05	[0.0 - 0.0]
FEEDRATE_ROUGH	0.01	[0.001 - 0.05]
FINISH_PASS	71	70 71
X_FINISH_STOCK	0.01	[0.00 - 0.05]
Z_FINISH_STOCK	0.003	[0.00 - 0.05]
RETRACT_X_HOME	Y	Y N
RETRACT_Z_HOME	Y	Y N
END_M_CODE	1	0 1 30

Enter the Z-AXIS CUT LENGTH

Company Name:

Machine type:

Serial number:

Build Date:



15 Easy Steps for Preventive Maintenance

1. Clean chips from way covers and bottom pan (50 hours of running)
2. Clean coolant collector (50 hours of running)
3. Inspect oil levels (50 hours of running)
4. Grease chuck jaws (50 hours of running)
5. Clean coolant filter screen, concentration and clean sediment inside tank (50 hours powered on)
6. Inspect way covers and lubricate (200 hours of running)
7. Grease the tailstock (200 hours of running)
8. Check axes grease reservoir level and lubrication tank level (200 hours of running)
9. Clean vector drive air vents and filters (200 hours powered on)
10. Inspect gearbox oil level (200 hours powered on)
11. Replace the oil filter (1200 hours powered on)
12. Inspect hoses for cracking (1200 hours powered on)
13. Clean the coolant filter, replace the coolant and clean coolant tank (1200 hours powered on)
14. Check oil, replace oil filter and clean oil tank (2400 hours of running)
15. Back Up Machine Data (2400 hours of running)



All maintenance intervals are recommended but can vary based on use.

(201)327-5215

service@alldalemachinery.com

www.HFOAllendale.com

Resources

[Haas Automation, Inc YouTube Channel](#)

HAASCNC.com

[HAAS Online Certification Program](#)