CNC Lathe Overview
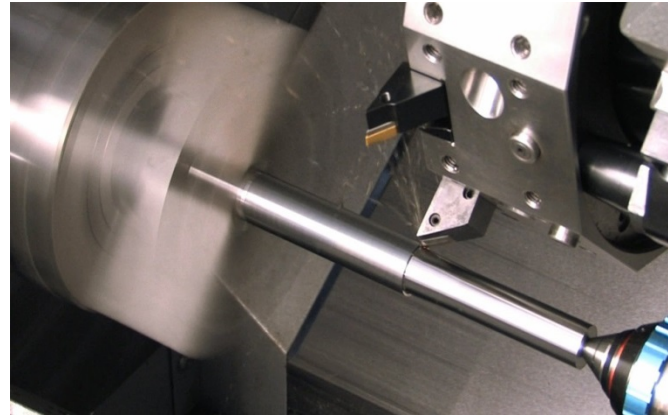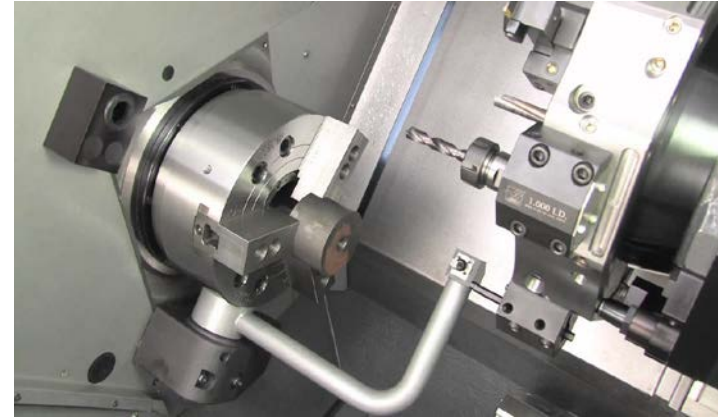
Prof. Steven S. Saliterman
Introductory Medical Device Prototyping
Department of Biomedical Engineering, University of Minnesota
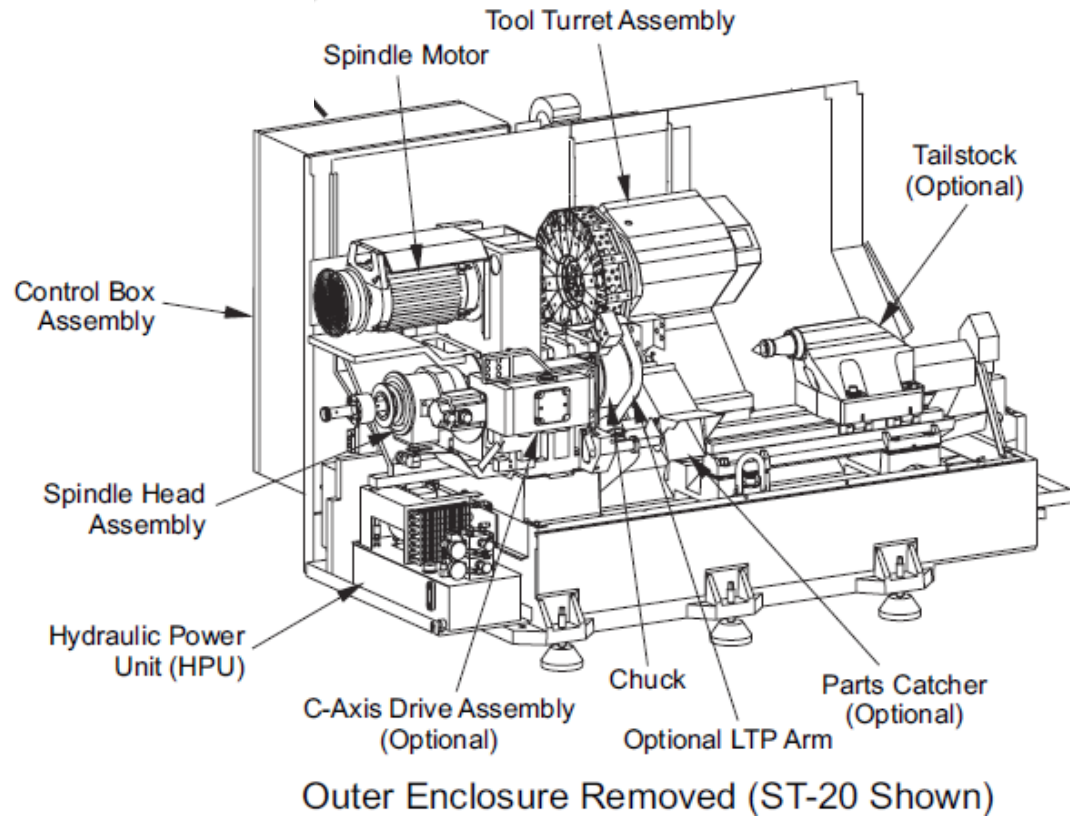http://saliterman.umn.edu/

Haas ST-10 CNC Lathe

Images courtesy of Haas

Safety Notice

- You must complete safety instruction before using tools and equipment in the Medical Device Center, ME Student Shop and CSE Workshops.
- All machinery can be dangerous. You must have a trained individual instruct you first when using unfamiliar equipment.
- Only authorized and trained individuals may operate CNC equipment.
- Code examples shown are for illustration purposes only, and are not meant for operation or programming actual equipment. They may be incomplete or contain errors.
- Always abide by shop safety instructions and never engage in horseplay.
- Remember to wear OSHA approved eye protection in the shop, short sleeves, leather or steel toed shoes, and secure long hair, avoid loose clothing, and take off rings, watches and bracelets when using power equipment.
- These slides are part of the "Introductory Medical Device Prototyping" course at the University of Minnesota, and are not meant for any other purpose.
- Formal training in Haas is available from Productivity, Inc.

Lathe Design

Outer Enclosure Removed (ST-20 Shown)
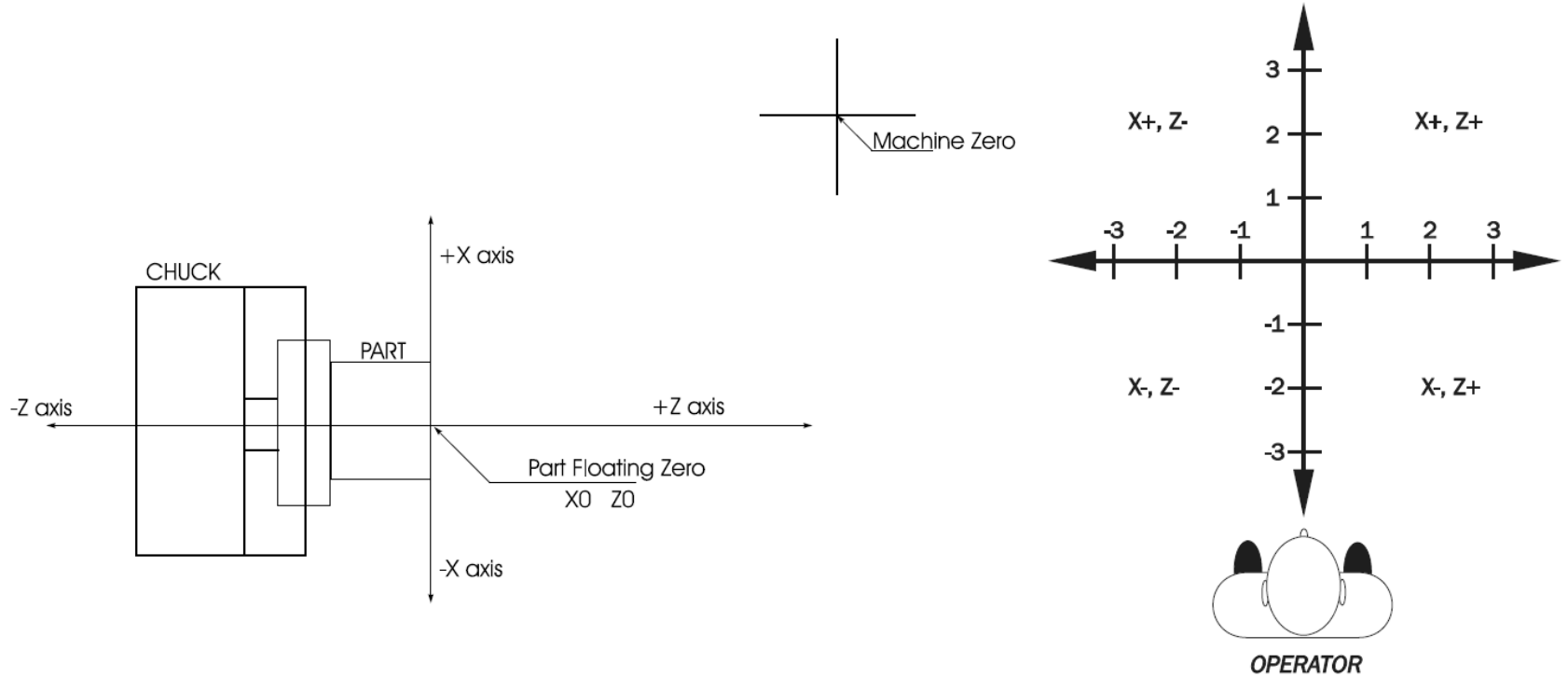
Image courtesy of Haas

Lathe Chuck and Tool Turret
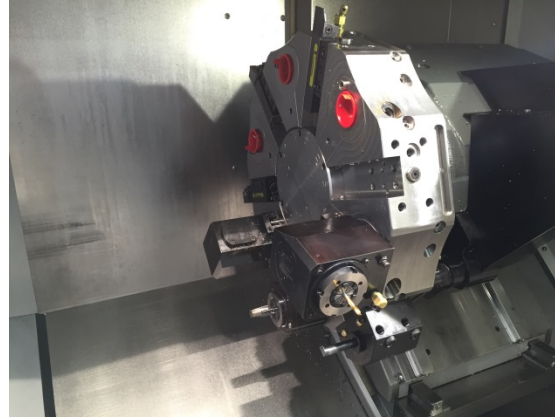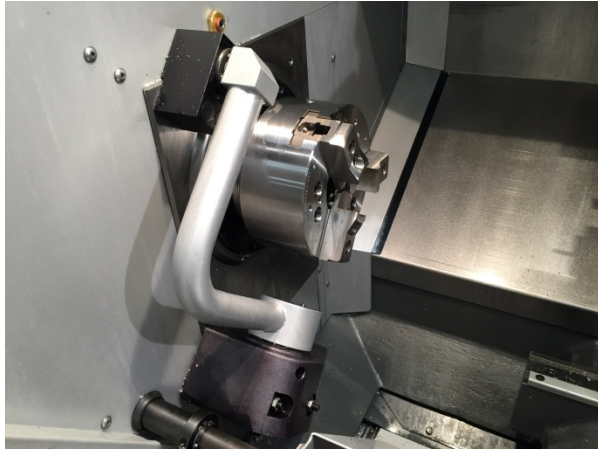
CNC Lathe Axis

- X is the back to front motion, with the *part* X0 being coincident with the Z axis. *Determines diameter.*

- Z is the spindle axis, and the *part* Z0 is normally the front finished face. *Determines location of faces, shoulders and grooves.*

- *Home reference position* – movement at startup to extreme limits. A zero-return at POWER/STARTUP. This is the *machine* zero.

- We use a *floating zero* referred to as the *part zero* or *part origin*.

- We touch off the face and diameter, and store the offset from the machine zero in X and Z register of the *Tool Offsets Page*. (Not the same as mill which uses G54!)

X & Z Axis

Machine Zero

CHUCK

PART

+X axis

-Z axis

+Z axis

Part Floating Zero
X0   Z0

-X axis

X+, Z-	X+, Z+
X-, Z-	X-, Z+

OPERATOR

Image courtesy of Productivity and Haas
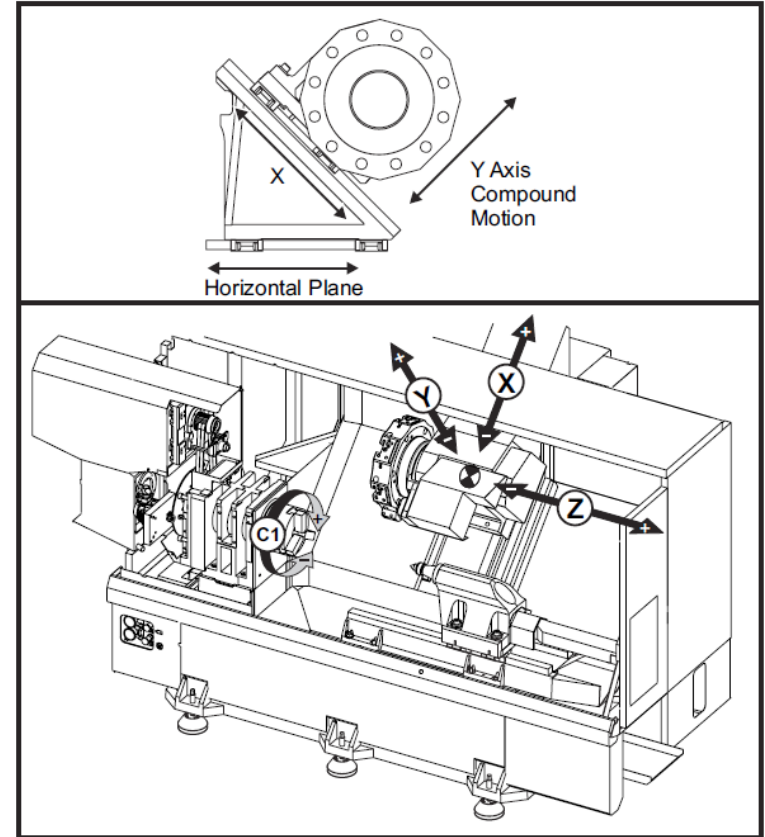
▸ Normally most of the X values are going to be positive in a part program. A negative X will occur only if you face a part past the centerline.

◦ X values are in <u>diameters (not radius)</u>, hence a X1.0 will be moving the machine along the x-axis 0.5" in the positive direction.

▸ Z values will tend to be negative since zero is at the face. (Max. -1" travel beyond the *spindle* zero.)

Absolute & Incremental Positioning

- Absolute positioning: X and Z codes are based on the Zero point of the part.
  - If a diameter of 1.0000 inches is needed, you input X1.0000.
  - If you are facing a shoulder 0.3" back from the face of the part, Z-3.0000 is input.
- Incremental positioning: U and W based on current position of the machine.
  - So a change of U-.5000 would be a smaller diameter of 0.5" from where the machine is presently at.
  - A grooving tool moving back ¾" behind the previous groove would be input as W-.7500.
  - Also called point to point.
- *Simultaneous* moves are possible e.g. G01 X2.000 W-.25 Moves X in absolute and Z in incremental at the same time.

Optional Y Axis

An optional Y-axis moves tools perpendicular to the spindle center line. This motion is achieved by compound motion of the X-axis and Y-axis ball screws.

Image courtesy of Haas

Control Pendant with Display

Current Mode

Example Inactive Pane

Example Active Pane

Program Display Pane

Main Display Pane

Unlock / Lock Messages

Speed and Feed Status / Editor Help

Position Display / Axis Load Meters / Clipboard

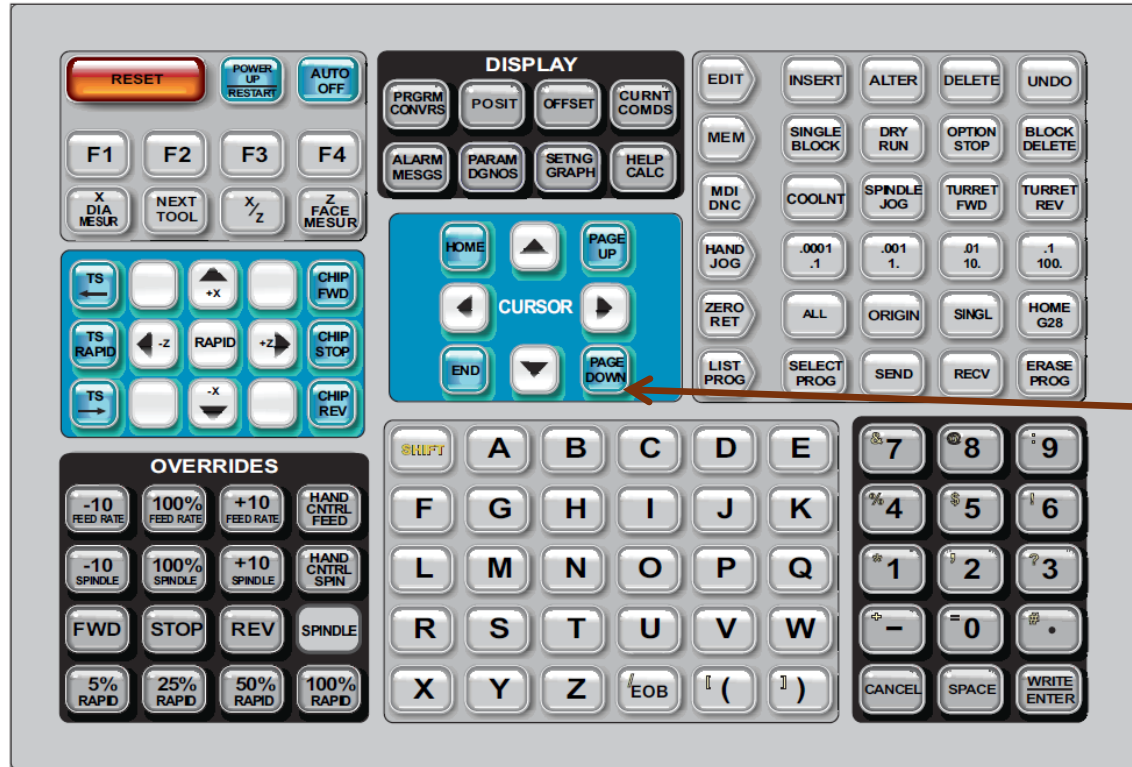Timers, Counters/ Tool Mgmt.

Messages

Input

Right Image courtesy of Haas
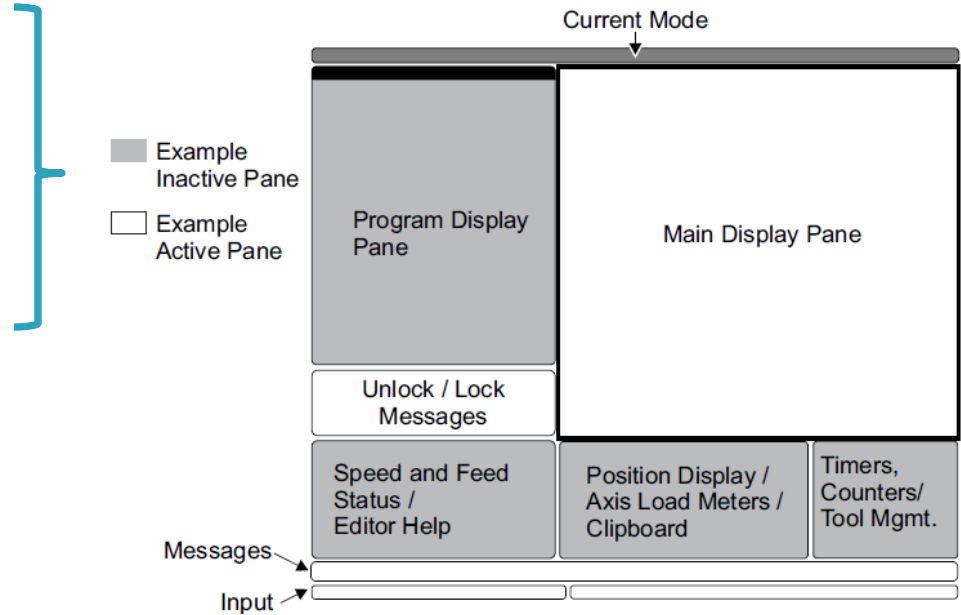
Keyboard Sections

Display buttons

Function buttons

Jog buttons

Override buttons

Mode buttons

Cursor buttons

Number buttons

Alpha buttons

Image courtesy of Haas

Powering On

- **Power On–** Turns CNC machine on. All axis to machine zero X then Y.
- **Power Off–** Turns CNC machine tool off.
- **Emergency Stop** – Stops all axis motion, spindle, tool changer and coolant pump.
- **Jog Handle–**Jogs axis selected; scroll through programs, menu items while editing and also altering feeds and speeds.

- **Cycle Start–** Starts program in run mode or graphics mode.
- **Feed Hold–** Stops all axis motion. Spindle will continue to turn.
- **Reset–** Stops machine, will rewind program.
- **Power Up/Restart–** Axis will return to machine zero
- **Recover–** If a tool change is stopped in middle of a cycle an alarm will come up. Push the Recover button and follow the instructions to bring the tool change cycle to the beginning.

Setup - Tool Offset

1. Tool (selected)) and work offset tables
2. Spindle, position and active tool figure.
3. ZERO RET, HAND JOG key mode buttons active

Prof. Steven S. Saliterman

Edit

1. Provides two program editing panes.
2. Allows program editing, management, and transfer functions.
3. EDIT, MDI/DNC, LIST PROG buttons active
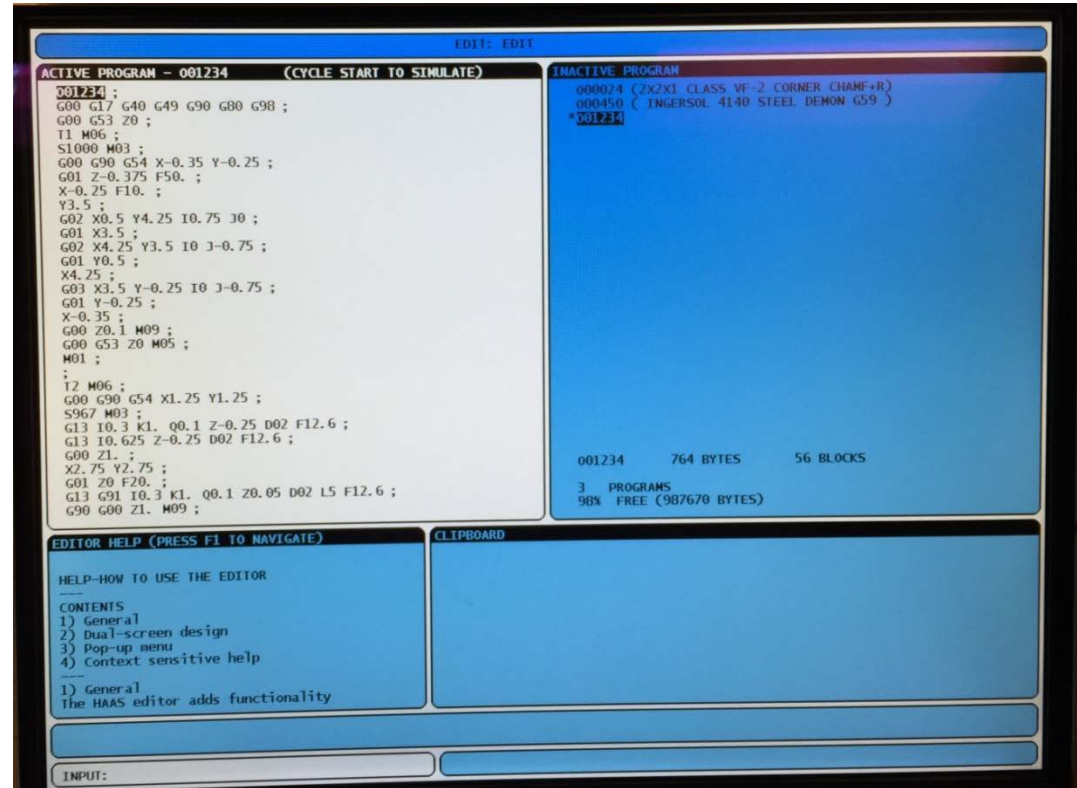
Operating

1. Provides all control features necessary to make a part.
2. MEM button active

Pendant Front Panel

Power On - Turns the machine on.

Power Off - Turns the machine off.

Emergency Stop - Large red button with a yellow bezel. Press to stop all axis motion, disable servos, stop the spindle and tool changer, and turn off the coolant pump. Turn to reset.

Jog Handle - This is used to jog axes (select in Hand Jog Mode). Also used to scroll through program code or menu items while editing.

Cycle Start - Starts a program. This button is also used to start a program simulation in Graphics mode.

Feed Hold - Stops all axis motion. Press Cycle Start to cancel. Note: The spindle will continue to turn during cutting.

Jog Mode

- Jog Mode allows you to jog each of the axes to a desired location. Before jogging the axes it is necessary to home (beginning axes reference point) the axes. (ZERO/RET AND AUTO/ALL AXES OR HOME/G28)
- To enter jog mode press HANDLE JOG, then press the desired axis (e.g. X, Z, etc.) and either use the handle jog keys or the jog handle to move the axis.
- There are different increment speeds that can be used while in jog mode; .0001, .001, .01 and .1.

Function Keys

- ❑ **F1- F4 Keys** –Different action in Editing Program and Offset mode.
- ❑ **X Dia Mesur (X Diameter Measure)** – Record X-axis tool shift offsets on the offset page during part setup.
- ❑ **Next Tool** – Next tool from the turret (usually used during part setup).
- ❑ **X/Z** – Toggle between X-axis and Z-axis jog modes during part setup.
- ❑ **Z Face Measur** (**Z Face Measure**) – Record Z-axis tool shift offsets on the offset page during part setup.

Jog Keys

- **Chip FWD** (Chip Auger Forward) – Move chips out of the machine.
- **Chip Stop** (Chip Auger Stop) – Stops auger movement.
- **Chip REV** (Chip Auger Reverse) - Starts the optional chip auger in the "Reverse" direction.
- **X/-X and Z/-Z** (axis keys) – Manually jog axis by holding down the individual key or pressing the desired axes and using the jog handle.
- **Rapid** – When pressed simultaneously with one of the above keys (X+, X-, Z+, Z-), that axis will move in the selected direction at maximum jog speed.

- **<- TS** – Pressing this key moves the tailstock towards the spindle.
- **TS Rapid** – Increases the speed of the tailstock when pressed simultaneously with one of the other tailstock keys.
- **-> TS** – Pressing this key moves the tailstock away from the spindle.
- **XZ (2-Axis) Jogging**

Override Keys

- ❑ **10** – Decreases current feed rate by 10%.
- ❑ **100%** – Sets control over ride feed rate to programmed feed rate.
- ❑ **+10** – Increases current feed rate by 10%.
- ❑ **−10** – Decreases current spindle speed by 10%.
- ❑ **100%** – Sets overridden spindle speed to programmed speed.
- ❑ **+10** – Increases current spindle speed by 10%.

Manual Tool Offset

- Manual offset entry:
  1. Load a tool into the tool turret.
  2. Press the HANDLE JOG key (A)
  3. Press .1/100. (B) (The lathe will move at a fast rate when the handle is turned).
  4. Toggle between the X and Z jog keys until the tool is touching the side of the part at about a 1/8 of an inch from the front edge.
  5. Place a sheet of paper between the tool and the part. Carefully move the tool as close as possible while still able to move the paper.

Images courtesy of Haas
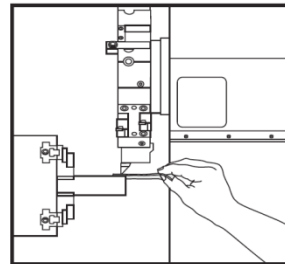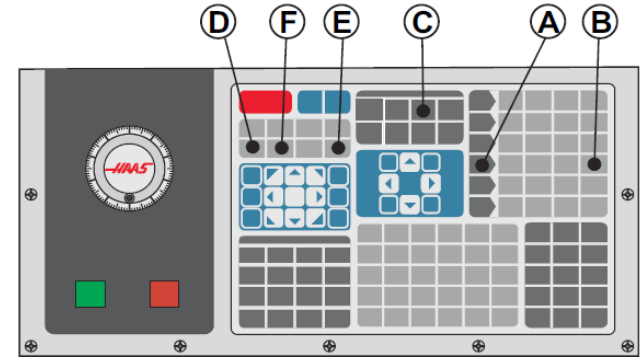
6. Press OFFSET (C), until the Tool Geometry table is displayed.
7. Press X DIA MESUR (D). The control will then prompt for the diameter of the part. This will take the X position located in the bottom left of the screen and the diameter of the part and put it with the position of the tool.
8. Back the tool from the part and position the tool tip so that it is touching the face of the stock.
9. Press Z FACE MEAS (E). This will take the current Z position and write it to the tool offset.
10. The cursor will move to the Z-axis location for the tool.
11. Press NEXT TOOL (F).

Setting Part Zero

- Part Zero is a user defined reference point that the CNC control will use to program all moves from.

- Select Tool #1 by pressing MDI/DNC, enter "T1" and press TURRET FWD.

- Jog X and Z until the tool is just touching the face of the part.

- Press Z FACE MEAS to set part zero. a

Tool Pre-Setter
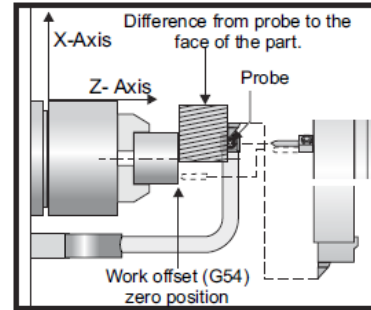
- Alternative to manually recording tool offsets.
- Each tool is "touched-off" on a probe.
- After that, only one tool is touched off on the part's zero position, and work offsets are automatically set for all tools.
- When the probe is down, axes can only be moved with handle jog.
- Once the probe is touched, the control will beep, the turret will halt, and the position of the tool will be stored typically in G54.
- Faster than manual method.

Prof. Steven S. Saliterman

Image courtesy of Haas

Automatic Tool Setting Probe

- The probe is first set up for tooling in manual mode, where initial tool measurements are made.
- After this setup, automatic mode is available to reset offsets when inserts are changed.
- Tool break detection is also available to monitor tool wear and breakage.
- The software generates G code which can be inserted into lathe programs to enable probe use during automatic operation.
- First press MDI/DNC and then PRGRM CONVRS to access the IPS tabbed menu set. Use the right cursor key to navigate to the PROBE tab and press WRITE/ENTER.

Probe Menu

Menu Item	Explanation
OP MODE	Use the left and right cursor arrow keys to choose between Manual, Automatic, and Break Detect modes.
TOOL NUMBER	The tool number to be used. This value automatically sets to the current tool position in Manual mode. It can be changed in Automatic and Break Detect Modes.
TOOL OFFSET	Enter the tool offset number that is being measured.
TOOL TIP DIR	Use the left and right cursor arrow keys to choose the tool nose vector V1-V8. See "Tool Tip Direction" for more information.
TOLERANCE	Sets the measurement difference tolerance for Break Detect mode. Not available in other modes.
X OFFSET, Z OFFSET	Displays the offset value for the specified axis. Read only.

MANUAL | SETUP | TURN & FAC | CHAMFER & RADI | DRILL & TA | THREAD | GROOVI | VC | PROBE
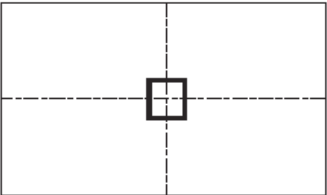
OP MODE
MANUAL

X OFFSET
0. 0000 in

TOOL NUMBER
1

Z OFFSET
0. 0000 in

TOOL OFFSET
1

TOOL TIP DIR
0

TOLERANCE
0.0000 in

This mode is used to manually enter the X and Z offset using the probe. The direction is governed by the TOOL TIP DIR selection.
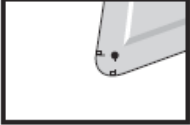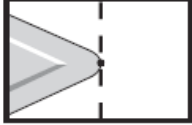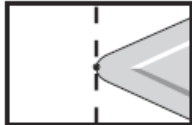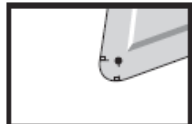
F1 – Lower / Raise probe arm.

Alarm Messages

Help Messages

Image courtesy of Haas

Tool Tip Direction
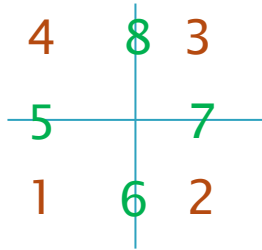
Tip Code	Imaginary Tool Tip Orientation	Tool Center Location
0		Zero (0) indicates no specified direction. It is usually not used when Tool Nose Compensation is desired.
1		Direction X+, Z+: Off tool
2		Direction X+, Z-: Off tool
3		Direction X-, Z-: Off tool
4		Direction X-, Z+: Off tool
5		Direction Z+: Tool edge
6		Direction X+: Tool edge
7		Direction Z-: Tool edge
8		Direction X-: Tool edge
9		Same as Tip 0

```
4   8   3

5   |   7

1   6   2
```

Image courtesy of Haas

Steps: First Manual Mode...

1. Tools must be touched off in manual mode before automatic mode can be used.
2. Enter the probe menu by pressing MDI/DNC, then PRGRM CONVRS, and selecting the Probe tab. Press F1 to lower the probe arm.
3. Select the tool to be touched off using TURRET FWD or TURRET REV.
4. Select Op Mode "Manual" using the left / right cursor arrow keys, then press WRITE/ENTER or the down cursor arrow key.
5. The tool offset option is set according to the currently selected tool position. Press WRITE/ENTER or the down cursor arrow key.
6. Type the tool offset number to be used, then press WRITE/ENTER. The offset number is entered and the next menu option, Tool Tip Dir, is selected.

7. Use the left / right cursor arrow keys to select a tool tip direction, then press WRITE/ENTER or the down cursor arrow key. See the "Tool Tip Direction" section for more information on this topic.

8. Use the jog handle to move the tool tip to within approximately 0.25" (6 mm) of the tool probe in the direction indicated by the on-screen tool tip direction diagram. Note that if the tool tip is too far from the probe, the tool will not reach the probe and the operation will alarm out.

9. Press CYCLE START. The tool tip is touched off and offsets are recorded and displayed. A G code program for the operation is generated in MDI and is used for tool movement.

10. Repeat steps 1-7 for each tool to be touched off. Be sure to jog the tool turret away from the probe before selecting the next tool position.

11. Press F1 to raise the tool arm.

Then Automatic Mode...

1. Use the left / right cursor arrow keys to select a tool tip direction, then press WRITE/ENTER or the down cursor arrow key. See the "Tool Tip Direction" section for more information on this topic.

2. Use the jog handle to move the tool tip to within approximately 0.25" (6 mm) of the tool probe in the direction indicated by the on-screen tool tip direction diagram. Note that if the tool tip is too far from the probe, the tool will not reach the probe and the operation will alarm out.

3. Press CYCLE START. The tool tip is touched off and offsets are recorded and displayed. A G code program for the operation is generated in MDI and is used for tool movement.

4. Repeat steps 1-7 for each tool to be touched off. Be sure to jog the tool turret away from the probe before selecting the next tool position.

5. Press F1 to raise the tool arm.

Mode Keys

- Edit
  - Used to make changes in a program stored in memory. When you press EDIT two panes appear at the top of the screen. In the left pane the active program appears. In the right an inactive program appears or the select program screen appears.
  - On the bottom left a editor help pane appears and on the right a clipboard pane. Editing may be performed in either the active or inactive panes.
  - Pressing EDIT toggles between the two panes, (changes background to white). To call up a program from memory and put it in one of the edit panes press SELCT/PROG. Highlight the program desired by using the up or down cursor buttons and press WRITE/ENTER.
  - In the edit mode you are able to use the edit keys in the same row as the EDIT key.

- **Insert** – Enters commands keyed into the input panel in lower left pane of CRT after the cursor highlighted word in a program.
- **Alter** – Highlighted words are replaced by text input into the input panel.
- **Delete** – Highlighted words are deleted from a program. Will undo up to the last 9 edit changes.

- **F1 key** – While in the edit mode pressing Fl will bring up an edit pop up window. Using the sideways cursor buttons will toggle thru HELP, MODIFY, SEARCH, EDIT AND PROGRAM MENUS. The up and down buttons will cursor thru the different options in each of the above.
- **Modify** – Gives options on changing line numbers.

Programs

- A block is a series of words on a single line ended with a ";" also known as the end-of-block (EOB) symbol.
  - Leading zeros and + signs are not needed.
  - Modal commands with G, X, Z,F, S, T and M need not be repeated in the following blocks unless a different word or change of value is needed.
  - Only one M code at the end of the block is permitted.
- Preparatory "G" codes make the tool do specific operations.
- "M" codes cause action to occur at the end of the block. Always executed last.
- N1-N9999 are optional sequence (line) numbers especially useful in macro subroutine.
- Programs begin and end with "%".
- The second line is the title of the program. Format is "O" followed by 5 digits.
- Enter comments by enclosing in parenthesis.
- A forward slash "/" denotes an optional block. BLOCK DELETE will skip these lines when running.

Subroutines

- Subroutines (subprograms) are usually a series of commands that are repeated several times in a program. Instead of repeating the commands many times in the main program, subroutines are written in a separate program.
- The main program has a single command that "calls" the subroutine program.
- A subroutine is called using M97 or M98 and a P address.
- The P code is the same as the program number (Onnnnn) number of the subroutine.
- The subroutines can include an **L** or repeat count. If there is an **L**, the subroutine call is repeated that number of times before the main program continues with the next block.

Address Codes

A	Forth Axis	R	Canned cycle and circular optional data
B	Linear B-axis motion		
C	Fifth axis rotary motion	S	Spindle speed (1-99999) No decimals allowed.
D	Canned cycle data		
E	Feed rate	T	Tool selection Txxyy, where xx is tool location with respect to the turret position and yy selects the tool offset (1-50).
F	Feed rate		
G	Type of operation, 0 to 255. G0x are non-modal, referring only to that block.		
		U	Incremental X axis motion
I, J, K	Canned cycle and circular optional data	V	Optional macro parameter
		W	Incremental Z axis motion
L	Loop count for repeated cycles.	X	Linear X axis motion
M	Control miscellaneous functions	Y	Linear Y-axis motion
N	Line number	Z	Linear Z-axis motion
O	Program number (name) Oxxxxx		
P	Delay time		
Q	Canned cycle optional data.		

M Codes (One per Block)

M00 axes	Program stop (spindle, coolant)		M24	Thread chamfer off
M03	Start spindle clockwise		M30	Program end and rewind
M04	Spindle counterclockwise		M41	Low gear
M05	Spindle stop		M42	High gear
M08	Coolant on		M85	Automatic door open
M09	Coolant off		M86	Automatic door close
M10	Clamp spindle chuck		M88	High pressure coolant off
M11	Unclamp spindle chuck		M97	Local sub-Program call (P or L)
M21	Tailstock forward		M98	Sub-program call (P or L)
M22	Reverse tailstock		M99	Return for subprogram or loop
M23	Thread chamfer (G76 or G92)			

Machine Defaults

- On power up the machine will go to part zero entered into G54 from prior probing.
- Automatic G Codes
  ◦ G00   Rapid traverse
  ◦ G18   X, Y circular plane section
  ◦ G40   Cutter compensation cancel
  ◦ G54   Work coordinate Zero #1 (1 of 26 available)
  ◦ G64   Exact stop cancel
  ◦ G97   Constant surface speed cancel
  ◦ G99   Feed per revolution
- Safety line usually not used with lathe:
  ◦ G18 G20 G40 G54 G80 G97 G99

Sample Program

G28;	(Rapid all axis machine zero)
G53 G00 X–3. Z–4.;	(Safe locate tool change turret)
T101;	(Indexes turret to tool)
G50 S2000;	(Spindle speed max.)
G97 S1146 M03;	(Cancel surface speed mode; spindle speed; clockwise)
G54 G00 X1.5 Z.02 M08;	(Rapid movement to start X, Z; coolant on)
G96 S450;	(Constant surface speed; surface feet per minute (SFM))
G00 Z1. M09;	(Move from part. coolant off)
G28;	(Rapids to machine home)
G53 G00 X–3. Z–4. T0;	(Back to tool change location)
M30;	(End of program. Stop spindle, turn off coolant, cancel tool length offsets)

Display Keys

- ❑ **Prgrm/Convrs** – Active program pane . In EDIT:MDI mode, press to access VQC and IPS (if installed).
- ❑ **Posit (Position)** – Positions pane, located in the lower center of most screens. Displays the current axis positions. Toggle between relative positions by pressing the POSIT key. To filter the axes displayed in the pane, type the letter for each axis you want to display and press WRITE/ENTER.

- ❑ **Offset** – Toggle between the two offsets tables. Tool, radius, wear offsets, and coolant position. Select the Work Offsets table to display and edit the G-code specified work offset locations used in programs.
- ❑ **Curnt Comds** (Current Commands) – Press PAGE UP / PAGE DOWN to cycle through menus for Maintenance, Tool Life, Tool Load, Advanced Tool Management (ATM), Barfeeder, System Variables, Clock settings and timer / counter settings.

- ❑ **Alarm / Mesgs (Alarms / Messages)** – Displays the alarm viewer and message screens. Pressing a second time will display a page for user messages and notes.
- ❑ **Param / Dgnos (Parameters / Diagnostics)** – Displays parameters that define the machine's operation. A second press key will display the first page of diagnostic data.
- ❑ **Setng / Graph (Settings / Graphics)** – Displays and allows changing of user settings. Pressing a second time enables Graphics mode. You can view the generated tool path of the program and, if necessary, debug the program before running it
- ❑ **Help / Calc (Help / Calculator)** – Displays help topics in a tabbed menu. Available help includes brief descriptions of G and M codes, definitions of control features, troubleshooting and maintenance issues. The help menu also includes several calculators.

Spindle Codes

- M03 Spindle Forward
- M04 Spindle Reverse
- M05 Spindle Stop
- G50 Maximum spindle speed (RPM – revolutions per min.)
- G96 Constant surface speed ON (SFM),
  - Tool speed increases automatically as diameter decreases.
  - $SFM = 0.2618 \; x \; Diameter \; x \; RPM$
- G97 Constant surface speed OFF (DEFAULT) & RPM ON,
  - $RPM = \dfrac{3.82 \; x \; SFM}{Diameter}$
- G98 Feed (inches) per minute, IPM = Current RPM x IPR
- G99 Feed (inches) per revolution(DEFAULT), IRP = IPM/Current RPM

Graphics Mode

- Safest way to troubleshoot (vs. Dry Run)
- Can run from Memory, MDI, DNC or Edit
- Press the SETNG/GRAPH key.
- In Edit mode, press CYCLE START
- F1 – Help key
- F2 – Zoom key
- F3 & F4 – Simulation Speed.

Prof. Steven S. Saliterman

Pendant Side Panel

USB - Plug compatible USB devices into this port.

Memory Lock - Keyswitch. Toggle to the lock position to protect programs and settings from alteration. Unlock to allow changes.

Setup Mode - Keyswitch. Locks and unlocks machine safety features for setup purposes (see the "Setup Mode" in the Safety section of this manual for details)

Second Home - Press this button to rapid all axes to the coordinates specified in G54 P18.
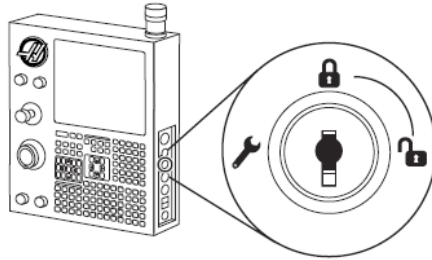
Autodoor Override - Press this button to open or close the Autodoor (if equipped).

Worklight - These switches toggle the internal worklight and High Intensity Lighting (if equipped).

Keyboard Beeper - Located at the top of the parts tray. The volume can be adjusted by turning the cover.

Prof. Steven S. Saliterman

Image courtesy of Haas

Lock & Unlock Behavior

Machine Behavior with Door Open

MACHINE FUNCTION 🔧	LOCKED 🔒	UNLOCKED 🔓
Maximum Rapid	Not allowed.	25%
Cycle Start	Not allowed, no machine motion or program execution.	With the Cycle Start button held down, machine motion is initiated provided the commanded spindle speed does not exceed 50 RPM.
Spindle CW/CCW	Yes, but user must press and hold FWD/REV key. Max. 50 RPM	Yes, but maximum 50 RPM.
Tool Change	Not allowed.	Not allowed.
Next Tool feature	Not allowed.	Allowed with the Next Tool key pressed and held.
Opening door while program is running.	Not allowed. Door is locked.	Yes, but axis motion will stop and the spindle will slow to a maximum 50 RPM.
Conveyor motion	Not allowed.	Yes, but user must press and hold conveyor button.

Images courtesy of Haas

Tailstock

- The optional tailstock is designed to travel to position at 2 rates. High pressure is called "rapid" and can be programmed with G00. Low pressure is called "feed" and can be programmed with G01.

- Recommended hydraulic tailstock operating pressure is 120 psi.

- Set a Restricted Zone for the tailstock with setting 93 (Tail ST. X Clearance) and Setting 94 (Z/TS Diff @ X Clearance)

Summary

- Lathe Axis and Absolute & Incremental Positioning.
- Control Pendant with Display
  - Power On; Set Up, Edit and Operating Displays
- Jog Mode
  - Function, Jog and Override Keys
- Manual Tool Offset
- Setting Part Zero
- Tool Pre-Setter with Probe
- Mode Keys
- Programming & G Codes
  - Programs, Subroutines, Address Codes, M Codes, Defaults, Safety Line
- Display Keys
- Spindle Codes
- Graphics Mode
- Pendant Side Panel & Lock and Unlock Behavior
- Tailstock

References

- Haas CNC Lathe Operator 2014
- Haas CNC Lathe Programming 2015
- Haas Lathe Operator's Manual 2016
- Haas Programming Workbook 2015
- Haas EBay Tutorials

Prof. Steven S. Saliterman