

## Tech 149 CNC Lecture

- CNC stands for **Computer Numerically Controlled**.
- It is the method of controlling a machine tool by the application of digital electronic computers and circuitry using **alpha-numerical data**.
- Machine movements (actuated and controlled by cams, gears, levers, or screws) are directed by computers and digital circuitry.

### **Specific Safety Precautions in Operating CNC Machines.**

The precautions outlined here assume that the operator/student has received appropriate instructions and demonstrations on how to safely program, set up and operate a HAAS computer numerically controlled (CNC) machine tool. Since CNC equipment follows only programmed instructions, these safety instructions must be adhered to when operating any such machine in the Manufacturing Systems lab.

- 1. Obtain instructor's permission.**
- 2. Do not alter or modify any machinery, tooling or accessory unless you contact an instructor and obtain permission.**
- 3. Review all CNC set up and operating procedures provided.**
- 4. Review all CNC programming instructions provided.**
- 5. Prepare and review your program carefully.**
- 6. Edit your program for safety, format, correctness and clarity.**
- 7. It is highly recommended that all programs be verified before the actual trial on the machine. Verification can be by a dry run on the machine, or through a graphic display of the tool path on the controller's screen. Do not operate any machine tool unless you are thoroughly familiar with it.**
8. Wear safety shoes.
9. Secure long hair or loose clothing that could become caught or tangled in the moving parts of machine. Long hair poses an extreme safety hazard around machine tools, and, therefore, must be netted for safety.
- 10. Wear your safety glasses.**
11. Determine the tools needed and get them ready. Tool length should not protrude too much from the holder. Use only properly sharpened tools. Use caution when changing tools – no interference with fixture or work.
12. Clamp all work securely before starting machine. Only approved materials can be machined. Abrasive dust-generating materials will wear machine components.
13. Do not use compressed air to blow chips from parts, machine surfaces, cabinets, controls, or floor around machine.

14. Avoid bumping any NC machine or controls. Work must not be held by hand while machining. Clamp it properly and securely in the vise.
15. Avoid using machine in wet, damp or poorly lighted work areas.
16. Perform all setup work with spindle stopped. Always stop the spindle completely before changing or adjusting the work piece, fixture or tool.
17. Wrenches, tools, and other parts should be kept off the machine and all its moving units. Do not use machine elements as a workbench.
18. Do not remove any guards or shields from any piece of equipment.
19. It is very unsafe to use gloves while operating rotating machinery.
20. Press the **green Power on** button so you can load your program to the machine controller.
21. Press the **Power Up** button to home the machine spindle.
22. Load your program via the RS232 interface (see the **RS 232 Interface Procedures** attached for directions).
23. Use the six-step **VMC Quick Start-Up Guide** (attached) to locate your work's X and Y zero position and tool length offset (Z zero). **Instructor must inspect and approve this step before you can proceed.**
24. When installed the chip guard doors must be kept locked at all times during machining.
25. With one hand very close to **Emergency Stop** button, press the **Cycle Start** button for machining to begin. **Stop the machine immediately if you notice any irregularity!** In all emergency situations, always push **EMERGENCY STOP** button.
26. Keep hands clear! Machine operates automatically and may move unexpectedly.
27. Never place any part of your body near moving parts of this machine. Do not machine flammable or toxic materials.
28. **Never place your hand on the tool in the spindle and press ATC, FWD, ATC REV, NEXT TOOL, or cause a tool change cycle. The tool changes will move in and crush you hand!**
29. **Allow the machine to complete the machining cycle and return to its home position, before reaching in to unclamp and remove your part.**
30. Shut off machine when not in use.
31. Never modify machine.
32. Do not disable hold-to-run switch.
33. Never open electrical compartment doors. Only qualified service personnel should open them.
34. Always unplug machine from electrical power before servicing.

**35. The table, vise, work piece, ways and chip pan must be kept clean after machining.**

36. Use a brush (table brush or paint brush) to clear chips from machine tools; do not use your hands, or a rag.
37. All dust and debris generated in machining should be vacuumed off daily. Never use fingers to remove chips. Use a brush instead.
38. Load and unload work pieces with spindle stopped. Never place hands near a revolving spindle.
39. Always ensure the spindle direction is correct. Check machine speed setting before starting machine to assure spindle is not started at an unsafe speed.
40. No horseplay of any kind is allowed in the CNC lab.
41. Any oil spill, coolant, or other fluid spill must be removed from the floor immediately. Use paper towels, wiping cloth, or a mop.
42. Rags must be kept clear of the rotating parts of machinery. If for any reason a rag gets caught in a machine, switch off the machine and stand clear of it until it comes to a complete stop.
43. All soiled rags must be stored in the covered metal containers provided.
44. Remove burrs/sharp edges from parts immediately after they are machined to avoid cuts on your hands. In addition, parts with burrs or sharp edges will receive reduced credit when evaluated for grading.
45. If any equipment is found to be in need of repair, report it to the instructor immediately. Do not attempt to use the equipment or repair it.
- 46. Students must clean the machines and area used during lab periods. Equipment must be returned at the close of the lab period. Students must sign out for any instrument, tool, or material they check out.**
47. Students will be held financially responsible for breakage or damage due to their own negligence or abuse.
48. Do not leave a machine unsafe for the next operator. Turn the power off when leaving a machine for an extended period.
49. Do not attempt to lift heavy work. Use help, hoist, or shop lift.
50. It is the responsibility of the operator to remove all chips, oil and residue from their machine, including the chip pan at the end of a shift, or when he/she is through using the machine. (No machine shall be left dirty for the next operator). Chips around a machine will be swept up and kept to a minimum by the operator. When cleaning a machine, use only a brush, rag, or towel. **USE OF HIGH PRESSURE AIR FOR CHIP REMOVAL OR MACHINE CLEANING IS PROHIBITED.** Don't alter OSHA approved air nozzles. Practice good housekeeping. Please report violators to responsible supervisors.
51. Do not dispose of oily paper towels in chip pans or rubbish receptacles. Use only the steel oil rag receptacles.

52. A dirty shop means accidents. Do not leave waste material or refuse lying around. Places are provided for storing them. Do your part to keep the shop clean and safe.
53. Protect your fellow students around you from possible injury from the carelessness on your part.
54. Never attempt to make electrical repairs. Ask you supervisor for approval.
55. PRACTICAL JOKES, HORESPLAY, THROWING OBJECTS, AND AIR HOSE GAMES ARE PROHIBITED.
56. Report any unsafe or hazardous conditions to your supervisor.
57. Smoking, eating food, drinking beverages, running or acting in a manner that might produce unsafe conditions, is prohibited in all laboratory and classroom areas.
58. NO ONE IS PERMITTED TO LEAVE THE LABORATORY UNTIL EVERYTHING IS CLEANED UP AND PUT AWAY. ANYONE LEAVING WILL RECEIVE A REDUCTION IN HIS OR HER FINAL GRADE.

## **G-Code and Other Letter Addresses Programming**

G-codes are also called preparatory codes, and are any word in a CNC program that begins with the letter "G". Generally, it is a code telling the machine tool what type of action to perform, such as:

- rapid move
- controlled feed move in a straight line or arc
- series of controlled feed moves that would result in a hole being bored, a workpiece cut (routed) to a specific dimension, or a decorative profile shape added to the edge of a workpiece.
- set tool information such as offset.

<b>Variable</b>	<b>Description</b>	<b>Corollary info</b>
A	Absolute or incremental position of A axis (rotational axis around X axis)	
B	Absolute or incremental position of B axis (rotational axis around Y axis)	
C	Absolute or incremental position	

	of C axis (rotational axis around Z axis)	
D	Defines diameter or radial offset used for cutter compensation	
E	Precision feedrate for threading on lathes	
F	Defines <u>feed rate</u>	
G	Address for preparatory commands	G commands often tell the control what kind of motion is wanted (e.g., rapid positioning, linear feed, circular feed, fixed cycle) or what offset value to use.
H	Defines tool length offset; Incremental axis corresponding to C axis (e.g., on a turn-mill)	
I	Defines arc size in X axis for <u>G02</u> or <u>G03</u> arc commands. Also used as a parameter within some fixed cycles.	
J	Defines arc size in Y axis for <u>G02</u> or <u>G03</u> arc commands. Also used as a parameter within some fixed cycles.	
K	Defines arc size in Z axis for <u>G02</u> or <u>G03</u> arc commands. Also used as a parameter within some fixed cycles, equal to <u>L</u> address.	
L	Fixed cycle loop count; Specification of what register to edit using <u>G10</u>	<i>Fixed cycle loop count:</i> Defines number of repetitions ("loops") of a fixed cycle at <i>each</i> position. Assumed to be 1 unless programmed with another integer. Sometimes the <u>K</u> address is used instead of L. With incremental positioning ( <u>G91</u> ), a series of equally spaced holes can be programmed as a loop rather than as individual positions.

		<u>G10 use</u> : Specification of what register to edit (work offsets, tool radius offsets, tool length offsets, etc.).
<b>M</b>	Miscellaneous function	Action code, auxiliary command; descriptions vary. Many M-codes call for machine functions, which is why people often say that the "M" stands for "machine", although it was not intended to.
<b>N</b>	Line (block) number in program; System parameter number to be changed using <u>G10</u>	<u>Line (block) numbers</u> : Optional, so often omitted. Necessary for certain tasks, such as <u>M99 P</u> address (to tell the control which block of the program to return to if not the default one) or <u>GoTo</u> statements (if the control supports those). <u>N</u> numbering need not increment by 1 (for example, it can increment by 10, 20, or 1000) and can be used on every block or only in certain spots throughout a program. <u>System parameter number</u> : <u>G10</u> allows changing of system parameters under program control.
<b>O</b>	Program name	For example, O4501.
<b>P</b>	Serves as parameter address for various G and M codes	<ul style="list-style-type: none"> <li>• With <u>G04</u>, defines dwell time value.</li> <li>• Also serves as a parameter in some canned cycles, representing dwell times or other variables.</li> <li>• Also used in the calling and termination of subprograms. (With <u>M98</u>, it specifies which subprogram to call; with <u>M99</u>, it specifies which block number of the main program to return to.)</li> </ul>
<b>Q</b>	Peck increment in canned cycles	For example, <u>G73</u> , <u>G83</u> (peck drilling cycles)
<b>R</b>	Defines size of arc radius or defines retract height in canned cycles	
<b>S</b>	Defines <u>speed</u> , either spindle speed or surface speed depending on mode	Data type = integer. In <u>G97</u> mode (which is usually the default), an integer after S is interpreted as a number of <u>rev/min</u> (rpm). In <u>G96</u> mode (CSS), an integer after S is interpreted as <u>surface speed</u> —sfm ( <u>G20</u> ) or m/min ( <u>G21</u> ). See also <u>Speeds and feeds</u> . On multifunction (turn-mill or mill-turn) machines, which spindle gets the input (main spindle or subspindles) is determined by other M codes.
<b>T</b>	Tool selection	To understand how the T address works and how it interacts (or not) with <u>M06</u> , one must study the various

		<p>methods, such as lathe turret programming, ATC fixed tool selection, ATC random memory tool selection, the concept of "next tool waiting", and empty tools. Programming on any particular machine tool requires knowing which method that machine uses.</p>
<b>U</b>	<p>Incremental axis corresponding to X axis (typically only lathe group A controls) Also defines dwell time on some machines (instead of "<u>P</u>" or "<u>X</u>").</p>	<p>In these controls, X and U obviate <u>G90</u> and <u>G91</u>, respectively. On these lathes, G90 is instead a fixed cycle address for roughing.</p>
<b>V</b>	<p>Incremental axis corresponding to Y axis</p>	<p>Until the 2000s, the V address was very rarely used, because most lathes that used U and W didn't have a Y-axis, so they didn't use V. (Green et al 1996<sup>[2]</sup> did not even list V in their table of addresses.) That is still often the case, although the proliferation of live lathe tooling and turn-mill machining has made V address usage less rare than it used to be (Smid 2008<sup>[1]</sup> shows an example). See also <u>G18</u>.</p>
<b>W</b>	<p>Incremental axis corresponding to Z axis (typically only lathe group A controls)</p>	<p>In these controls, Z and W obviate <u>G90</u> and <u>G91</u>, respectively. On these lathes, G90 is instead a fixed cycle address for roughing.</p>
<b>X</b>	<p>Absolute or incremental position of X axis. Also defines dwell time on some machines (instead of "<u>P</u>" or "<u>U</u>").</p>	
<b>Y</b>	<p>Absolute or incremental position of Y axis</p>	
<b>Z</b>	<p>Absolute or incremental position of Z axis</p>	<p>The main spindle's axis of rotation often determines which axis of a machine tool is labeled as Z.</p>

## Commonly Used G-Codes

Code	Description	Milling ( M )	Turning ( T )	Corollary info
G00	Rapid positioning	M	T	On 2- or 3-axis moves, G00 (unlike <u>G01</u> ) traditionally does not necessarily move in a single straight line between start point and end point. It moves each axis at its max speed until its vector is achieved. Shorter vector usually finishes first (given similar axis speeds). This matters because it may yield a dog-leg or hockey-stick motion, which the programmer needs to consider depending on what obstacles are nearby, to avoid a crash. Some machines offer interpolated rapids as a feature for ease of programming (safe to assume a straight line).
G01	Linear interpolation	M	T	The most common workhorse code for feeding during a cut. The program specs the start and end points, and the control automatically calculates ( <u>interpolates</u> ) the intermediate points to pass through that will yield a straight line (hence " <u>linear</u> "). The control then calculates the angular velocities at which to turn the axis <u>leadscrews</u> . The computer performs thousands of calculations per second. Actual machining takes place with given feed on linear path.
G02	Circular interpolation, clockwise	M	T	Cannot start <u>G41</u> or <u>G42</u> in <u>G02</u> or <u>G03</u> modes. Must already be compensated in earlier <u>G01</u> block.
G03	Circular interpolation, counterclockwise	M	T	Cannot start <u>G41</u> or <u>G42</u> in <u>G02</u> or <u>G03</u> modes. Must already be compensated in earlier <u>G01</u> block.
G04	Dwell	M	T	Takes an address for dwell period (may be <u>X</u> , <u>U</u> , or <u>P</u> ). The dwell period is specified in <b>milliseconds</b> .
G05 P10000	High-precision contour control (HPCC)	M		Uses a deep look-ahead <u>buffer</u> and simulation processing to provide better

				axis movement acceleration and deceleration during contour milling
G05.1 Q1.	<u>Ai</u> Nano contour control	M		Uses a deep look-ahead <u>buffer</u> and simulation processing to provide better axis movement acceleration and deceleration during contour milling
G07	Imaginary axis designation	M		
G09	Exact stop check	M	T	
G10	Programmable data input	M	T	
G11	Data write cancel	M	T	
G12	Full-circle interpolation, clockwise	M		Fixed cycle for ease of programming 360° circular interpolation with blend-radius lead-in and lead-out. Not standard on Fanuc controls.
G13	Full-circle interpolation, counterclockwise	M		Fixed cycle for ease of programming 360° circular interpolation with blend-radius lead-in and lead-out. Not standard on Fanuc controls.
G17	XY plane selection	M		
G18	ZX plane selection	M	T	On most CNC lathes (built 1960s to 2000s), ZX is the only available plane, so no <u>G17</u> to <u>G19</u> codes are used. This is now changing as the era begins in which live tooling, multitask/multifunction, and mill-turn/turn-mill gradually become the "new normal". But the simpler, traditional form factor will probably not disappear—just move over to make room for the newer configurations. See also <u>V</u> address.
G19	YZ plane selection	M		
G20	<u>Programming in inches</u>	M	T	Somewhat uncommon except in USA and (to lesser extent) Canada and UK. However, in the global marketplace, competence with both G20 and G21 always stands some chance of being necessary at any time. The usual minimum increment in G20 is one ten-thousandth of an inch (0.0001"), which

				is a larger distance than the usual minimum increment in G21 (one thousandth of a millimeter, .001 mm, that is, one <u>micrometre</u> ). This physical difference sometimes favors G21 programming.
G21	Programming in <u>millimeters</u> (mm)	M	T	Prevalent worldwide. However, in the global marketplace, competence with both G20 and G21 always stands some chance of being necessary at any time.
G28	Return to home position (machine zero, aka machine reference point)	M	T	Takes X Y Z addresses which define the intermediate point that the tool tip will pass through on its way home to machine zero. They are in terms of part zero (aka program zero), NOT machine zero.
G30	Return to secondary home position (machine zero, aka machine reference point)	M	T	Takes a P address specifying <i>which</i> machine zero point is desired, <i>if</i> the machine has several secondary points (P1 to P4). Takes X Y Z addresses which define the intermediate point that the tool tip will pass through on its way home to machine zero. They are in terms of part zero (aka program zero), NOT machine zero.
G31	Skip function (used for probes and tool length measurement systems)	M		
G32	Single-point threading, longhand style (if not using a cycle, e.g., <u>G76</u> )		T	Similar to <u>G01</u> linear interpolation, except with automatic spindle synchronization for <u>single-point threading</u> .
G33	Constant- <u>pitch</u> threading	M		
G33	Single-point threading, longhand style (if not using a cycle, e.g., <u>G76</u> )		T	Some lathe controls assign this mode to G33 rather than G32.
G34	Variable-pitch threading	M		
G40	Tool radius compensation off	M	T	Cancels G41 or G42.

G41	Tool radius compensation left	M	T	<p><b>Milling:</b> Given righthand-helix cutter and <u>M03</u> spindle direction, G41 corresponds to <u>climb milling (down milling)</u>. Takes an address (<u>D</u> or <u>H</u>) that calls an offset register value for radius.</p> <p><b>Turning:</b> Often needs no D or H address on lathes, because whatever tool is active automatically calls its geometry offsets with it. (Each turret station is bound to its geometry offset register.)</p>
G42	Tool radius compensation right	M	T	<p>Similar corollary info as for G41. Given righthand-helix cutter and M03 spindle direction, G42 corresponds to <u>conventional milling (up milling)</u>.</p>
G43	Tool height offset compensation negative	M		<p>Takes an address, usually H, to call the tool length offset register value. The value is <i>negative</i> because it will be <i>added</i> to the gauge line position. G43 is the commonly used version (vs G44).</p>
G44	Tool height offset compensation positive	M		<p>Takes an address, usually H, to call the tool length offset register value. The value is <i>positive</i> because it will be <i>subtracted</i> from the gauge line position. G44 is the seldom-used version (vs G43).</p>
G45	Axis offset single increase	M		
G46	Axis offset single decrease	M		
G47	Axis offset double increase	M		
G48	Axis offset double decrease	M		
G49	Tool length offset compensation cancel	M		<p> Cancels <u>G43</u> or <u>G44</u>.</p>
G50	Define the maximum spindle speed		T	<p>Takes an <u>S</u> address integer which is interpreted as rpm. Without this feature, <u>G96</u> mode (CSS) would rev the spindle to "wide open throttle"</p>

				when closely approaching the axis of rotation.
G50	Scaling function cancel	M		
G50	Position register (programming of vector from part zero to tool tip)		T	Position register is one of the original methods to relate the part (program) coordinate system to the tool position, which indirectly relates it to the machine coordinate system, the only position the control really "knows". Not commonly programmed anymore because <u>G54 to G59</u> (WCSs) are a better, newer method. Called via G50 for turning, <u>G92</u> for milling. Those G addresses also have alternate meanings (which see). Position register can still be useful for datum shift programming.
G52	Local coordinate system (LCS)	M		Temporarily shifts program zero to a new location. This simplifies programming in some cases.
G53	Machine coordinate system	M	T	Takes absolute coordinates (X,Y,Z,A,B,C) with reference to machine zero rather than program zero. Can be helpful for tool changes. Nonmodal and absolute only. Subsequent blocks are interpreted as "back to <u>G54</u> " even if it is not explicitly programmed.
G54 to G59	Work coordinate systems (WCSs)	M	T	Have largely replaced position register ( <u>G50</u> and <u>G92</u> ). Each tuple of axis offsets relates program zero directly to machine zero. Standard is 6 tuples (G54 to G59), with optional extensibility to 48 more via G54.1 P1 to P48.
G54.1 P1 to P48	Extended work coordinate systems	M	T	Up to 48 more WCSs besides the 6 provided as standard by G54 to G59. Note floating-point extension of G-code data type (formerly all integers). Other examples have also evolved (e.g., <u>G84.2</u> ). Modern controls have the <u>hardware</u> to handle it.
G70	Fixed cycle, multiple repetitive		T	

	cycle, for finishing (including contours)			
G71	Fixed cycle, multiple repetitive cycle, for roughing (Z-axis emphasis)		T	
G72	Fixed cycle, multiple repetitive cycle, for roughing (X-axis emphasis)		T	
G73	Fixed cycle, multiple repetitive cycle, for roughing, with pattern repetition		T	
G73	Peck drilling cycle for milling - high-speed (NO full retraction from pecks)	M		Retracts only as far as a clearance increment (system parameter). For when chipbreaking is the main concern, but chip clogging of flutes is not.
G74	Peck drilling cycle for turning		T	
G74	Tapping cycle for milling, <u>lefthand thread</u> , M04 spindle direction	M		
G75	Peck grooving cycle for turning		T	
G76	Fine boring cycle for milling	M		
G76	Threading cycle for turning, multiple repetitive cycle		T	
G80	Cancel canned cycle	M	T	<b>Milling:</b> Cancels all cycles such as <u>G73</u> , <u>G83</u> , <u>G88</u> , etc. Z-axis returns either to Z-initial level or R-level, as programmed ( <u>G98</u> or <u>G99</u> , respectively). <b>Turning:</b> Usually not needed on lathes, because a new group-1 G address ( <u>G00</u> to <u>G03</u> ) cancels whatever cycle was active.

G81	Simple drilling cycle	M		No dwell built in
G82	Drilling cycle with dwell	M		Dwells at hole bottom (Z-depth) for the number of <b>milliseconds</b> specified by the <u>P</u> address. Good for when hole bottom finish matters.
G83	Peck drilling cycle (full retraction from pecks)	M		Returns to R-level after each peck. Good for clearing flutes of <u>chips</u> .
G84	<u>Tapping</u> cycle, <u>righthand thread</u> , <u>M03</u> spindle direction	M		
G84.2	Tapping cycle, righthand thread, <u>M03</u> spindle direction, rigid toolholder	M		
G90	Absolute programming	M	T (B)	Positioning defined with reference to part zero. <b>Milling:</b> Always as above. <b>Turning:</b> Sometimes as above (Fanuc group type B and similarly designed), but on most lathes (Fanuc group type A and similarly designed), G90/G91 are not used for absolute/incremental modes. Instead, <u>U</u> and <u>W</u> are the incremental addresses and <u>X</u> and <u>Z</u> are the absolute addresses. On these lathes, G90 is instead a fixed cycle address for roughing.
G90	Fixed cycle, simple cycle, for roughing (Z-axis emphasis)		T (A)	When not serving for absolute programming (above)
G91	Incremental programming	M	T (B)	Positioning defined with reference to previous position. <b>Milling:</b> Always as above. <b>Turning:</b> Sometimes as above (Fanuc group type B and similarly designed), but on most lathes (Fanuc group type A and similarly designed), G90/G91 are not used for absolute/incremental modes. Instead, <u>U</u> and <u>W</u> are the incremental addresses and <u>X</u> and <u>Z</u> are

				the absolute addresses. On these lathes, G90 is a fixed cycle address for roughing.
G92	Position register (programming of vector from part zero to tool tip)	M	T (B)	Same corollary info as at <u>G50</u> position register. <b>Milling:</b> Always as above. <b>Turning:</b> Sometimes as above (Fanuc group type B and similarly designed), but on most lathes (Fanuc group type A and similarly designed), position register is <u>G50</u> .
G92	Threading cycle, simple cycle		T (A)	
G94	Feedrate per minute	M	T (B)	On group type A lathes, feedrate per minute is <u>G98</u> .
G94	Fixed cycle, simple cycle, for roughing ( <u>X</u> -axis emphasis)		T (A)	When not serving for feedrate per minute (above)
G95	Feedrate per revolution	M	T (B)	On group type A lathes, feedrate per revolution is <u>G99</u> .
G96	Constant surface speed (CSS)		T	Varies spindle speed automatically to achieve a constant surface speed. See <u>speeds and feeds</u> . Takes an <u>S</u> address integer, which is interpreted as <u>sfm</u> in <u>G20</u> mode or as m/min in <u>G21</u> mode.
G97	Constant spindle speed	M	T	Takes an <u>S</u> address integer, which is interpreted as rev/min (rpm). The default speed mode per system parameter if no mode is programmed.
G98	Return to initial Z level in canned cycle	M		
G98	Feedrate per minute (group type A)		T (A)	Feedrate per minute is <u>G94</u> on group type B.
G99	Return to R level in canned cycle	M		
G99	Feedrate per revolution (group type A)		T (A)	Feedrate per revolution is <u>G95</u> on group type B.

## M-Codes

Code	Description	Milling ( M )	Turning ( T )	Corollary info
M00	Compulsory stop	M	T	Non-optional—machine will always stop upon reaching M00 in the program execution.
M01	Optional stop	M	T	Machine will only stop at M01 if operator has pushed the optional stop button.
M02	End of program	M	T	No return to program top; may or may not reset register values.
M03	Spindle on (clockwise rotation)	M	T	<p>The speed of the spindle is determined by the address <b>S</b>, in <u>surface feet per minute</u>. The <u>right-hand rule</u> can be used to determine which direction is clockwise and which direction is counter-clockwise.</p> <p>Right-hand-helix screws moving in the tightening direction (and right-hand-helix flutes spinning in the cutting direction) are defined as moving in the M03 direction, and are labeled "clockwise" by convention. The M03 direction is always M03 regardless of local vantage point and local CW/CCW distinction.</p>
M04	Spindle on (counterclockwise rotation)	M	T	See comment above at M03.
M05	Spindle stop	M	T	
M06	Automatic tool change (ATC)	M	T (sometimes)	<p>Many lathes do not use M06 because the <u>T</u> address itself indexes the turret. To understand how the T address works and how it interacts (or not) with M06, one must study the various methods, such as lathe turret programming, ATC fixed tool selection, ATC random memory tool selection, the concept of "next tool waiting", and empty tools. Programming on any particular</p>

				machine tool requires knowing which method that machine uses.
M07	Coolant on (mist)	M	T	
M08	Coolant on (flood)	M	T	
M09	Coolant off	M	T	
M10	Pallet clamp on	M		For machining centers with pallet changers
M11	Pallet clamp off	M		For machining centers with pallet changers
M13	Spindle on (clockwise rotation) and coolant on (flood)	M		This one M-code does the work of both <u>M03</u> and <u>M08</u> . It is not unusual for specific machine models to have such combined commands, which make for shorter, more quickly written programs.
M19	Spindle orientation	M	T	Spindle orientation is more often called within cycles (automatically) or during setup (manually), but it is also available under program control via <b>M19</b> . The abbreviation OSS (oriented spindle stop) may be seen in reference to an oriented stop within cycles.
M21	Mirror, <u>X</u> -axis	M		
M21	Tailstock forward		T	
M22	Mirror, <u>Y</u> -axis	M		
M22	Tailstock backward		T	
M23	Mirror OFF	M		
M23	Thread gradual pullout ON		T	
M24	Thread gradual pullout OFF		T	
M30	End of program with return to program top	M	T	
M41	Gear select - gear 1		T	
M42	Gear select - gear 2		T	
M43	Gear select - gear 3		T	
M44	Gear select - gear 4		T	
M48	Feedrate override allowed	M	T	

M49	Feedrate override NOT allowed	M	T	This rule is also called (automatically) within tapping cycles or single-point threading cycles, where feed is precisely correlated to speed. Same with spindle speed override and feed hold button.
M60	Automatic pallet change (APC)	M		For machining centers with pallet changers
M98	Subprogram call	M	T	Takes an address <u>P</u> to specify which subprogram to call, for example, "M98 P8979" calls subprogram O8979.
M99	Subprogram end	M	T	Usually placed at end of subprogram, where it returns execution control to the main program. The default is that control returns to the block following the M98 call in the main program. Return to a different block number can be specified by a P address. M99 can also be used in main program with block skip for endless loop of main program on bar work on lathes (until operator toggles block skip).

## Sample Program

```

%
O0000
(PROGRAM NAME - OBI1)
( DATE=DD-MM-YY - 14-02-07 TIME=HH:MM - 12:05)
N100G20
N102G0G17G40G49G80G90
( 1/4 FLAT ENDMILL TOOL - 1 DIA. OFF. - 21 LEN. - 2 DIA. - .25)
N104T1M6;
N106G0G90G54X-.125Y-.125S2139M3;
N108G43H2Z.25;
N110Z.1;
N112G1Z-.1F6.42;
N114Y2.125;
N116X2.125;
N118Y-.125;
N120X-.125;
N122G0Z.25;
N124M5;
N126G91G28Z0;

```

N128G28X0.Y0;  
 N130M30;  
 %

## Sample Program Interpretations

Line	Code	Description
O4968		(Sample face and turn program)
N01	M216	(Turn on load monitor)
N02	G20 G90 G54 D200 G40	(Inch units. Absolute mode. Call work offset values. Moving coordinate system to the location specified in the register D200. Cancel any existing tool radius offset.)
N03	G50 S2000	(Set maximum spindle speed rev/min - preparing for G96 CSS coming soon)
N04	M01	(Optional stop)
N05	T0300	(Index turret to tool 3. Clear wear offset (00).)
N06	G96 S854 M42 M03 M08	(Constant surface speed [automatically varies the spindle speed], 854 sfm, select spindle gear, start spindle CW rotation, turn on the coolant flood)
N07	G41 G00 X1.1 Z1.1 T0303	(Call tool radius offset. Call tool wear offset. Rapid feed to a point <i>about</i> 0.100" from the end of the bar [not counting 0.005" or 0.006" that the bar-pull-and-stop sequence is set up to leave as a stock allowance for facing off] and 0.050" from the side)
N08	G01 Z1.0 F.05	(Feed in horizontally until the tool is standing 1" from the datum i.e. program Z-zero)
N09	X-0.002	(Feed down until the tool is slightly past center, thus facing the end of the bar)
N10	G00 Z1.1	(Rapid feed 0.1" away from the end of the bar - clear the part)
N11	X1.0	(Rapid feed up until the tool is standing at the finished OD)
N12	G01 Z0.0 F.05	(Feed in horizontally cutting the bar to 1" diameter all the way to the datum, feeding at 0.050" per revolution)
N13	G00 X1.1 M05 M09	(Clear the part, stop the spindle, turn off the coolant)
N14	G91 G28 X0	(Home X axis - return to machine X-zero passing through no intermediate X point [incremental X0])
N15	G91 G28 Z0	(Home Z axis - return to machine Z-zero passing through no intermediate Z point [incremental Z0])
N16	G90 M215	(Return to absolute mode. Turn off load monitor)
N17	M30	(Program stop, rewind to beginning of program)
%		End of program

## Debugging Your Program

The fact that you have successfully posted your program is no guarantee that it will work without a few snags. It is recommended that you debug your program before downloading it to the CNC machine's controller. Following are some tips and recommendations to help you provide a more fool-proof program.

1. Determine and note the starting block of your program. This point must be in agreement with the portion of the part graphic where the cross hairs cross in your CAMWorks setup. This point will become your X and Y part zero location during set up
2. Ensure that your program begins and ends with %.
3. Every program block should end with a semicolon (;).
4. Your program should have a program name that begins with the letter "O" and a four-digit number (Onnnn).
5. Define the unit of your program (G20 for inches)
6. Define whether you are programming in absolute (G90) or incremental (G91)
7. Cancel any X and Y cutter compensation with G40
8. Cancel the tool length compensation with G49
9. Set the plane to XY plane with G17
10. Delete unnecessary axes such as "A" and "B" since the two CNC mills are not enabled for more than three axes.
11. Delete unnecessary auxiliary functions like coolant (M08) and chip pan since the two CNC mills are not enabled for these options.
12. Check all your program blocks to ensure there is no garbage in the program.
13. Save your program on a flash drive and take it to the CNC lab for downloading to the machine controller.



Figure 1: Haas Controller

## Procedure to Download (from PC to Haas)

1. On the Haas control shown in Figure 1, press the **List Prog** button
2. Use cursor keys to highlight **ALL**
3. Press the **RCV RS232** button
4. On the PC open the Predator Editor software
5. Click on File and Open (**to open your flash drive**)
6. Select your program and click Open
7. Click on the DNC menu bar and select **Send to CNC** option
8. After the status line on the Haas machine says "**Download complete**" you can perform your work on the Haas

## Procedure to Upload (from Haas to PC)

1. On the Haas use cursor to select program of interest (it should have a star by it)
2. Press the **List Prog** button
3. Use cursor key to select program and press **SELECT PROG** key
4. Use cursor keys to highlight ALL
5. Press the **SEND RS232** button
6. On the PC open the Predator Editor software
7. Click on the DNC menu bar and select **Receive from CNC** option
8. After the status line on the Haas machine says "**Upload complete**" you are done.

## **Setting Up and Machining Your Workpiece on HAAS CNC Machine**

### **CLAMPING YOUR PART**

1. Place your part in the soft jaws of vise on the machine table
2. Ensure the part sits in such a way that at least .2" of it is above the top of the vise jaws
3. Tighten with just enough pressure so that the acrylic is not marred or damaged
4. Use a soft hammer to set it as needed

### **LOCATING X AND Y ZERO POINTS OF YOUR PART (PART ZERO POINT) USING AN EDGE FINDER, INDICATOR OR YOUR TOOL'S CENTER POINT**

1. Use the jog function keys and dial to move the tool to part zero point
2. Press the **OFFSET** key
3. **PAGE UP** on work coordinate page to **G54 X**
4. Push **PART ZERO SET** key and the **X-axis** value will be stored as the offset
5. The cursor will automatically move to the **G54 Y** location
6. Push **PART ZERO SET** key and the **Y-axis** value will be stored as the offset
7. Leave the Z value at zero at this point

## **SETTING THE TOOL OFFSETS**

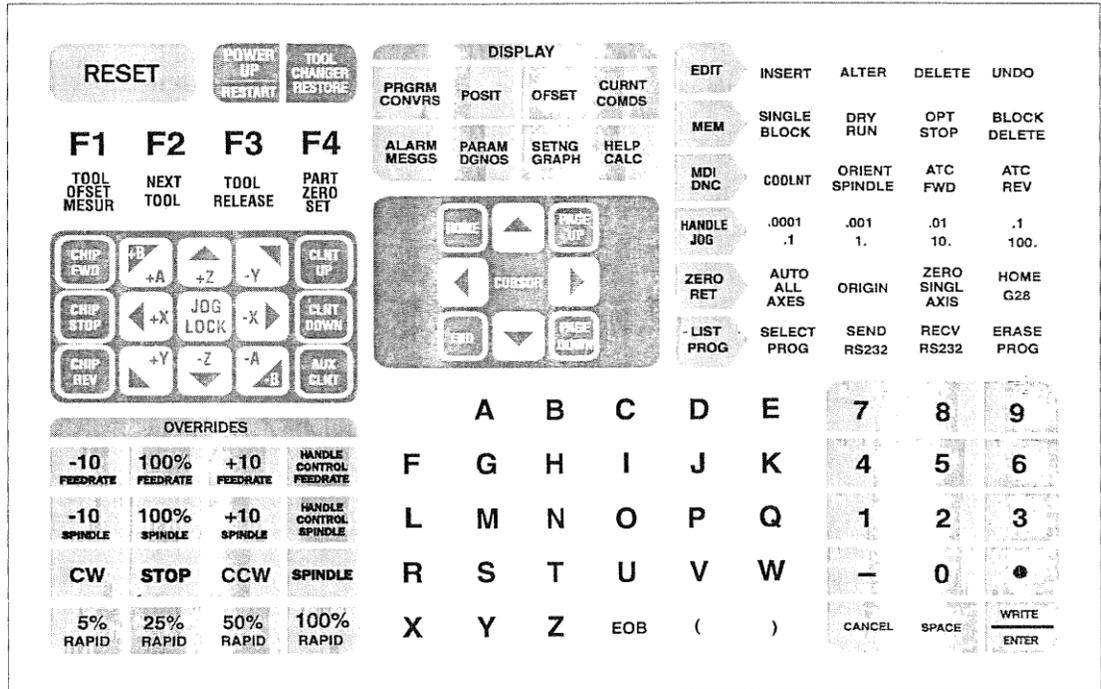
1. Use the jog handle to accurately position tool tip to **Z0** on top of your part
2. Press the **OFFSET** key
3. **PAGE DOWN** to the Tool Offset page and cursor to **Tool #1**
4. Press the **TOOL OFFSET MESUR** key and the **Z value** will be stored in tool **offset #1**
5. Repeat 1-4 for other tools if using multiple tools
6. Press **Power Up** button to return tool to its home position

## **RUNNING YOUR PROGRAM**

1. Press the **LIST PROG** key
2. Cursor to your program
3. Press **SELECT PROG** key
4. Press **MEM**
5. Hold the safety switch and press the **CYCLE START** button to begin execution



**2.10 OPERATOR'S CONTROL PANEL**



Control panel keypad with operating and display keys highlighted.

In operation, it is important to be aware of the operating mode selected for the CNC. There are six operating modes and one simulation mode in this control. The operating mode is selected with the six buttons labeled:



# VMC QUICK START-UP GUIDE



The purpose of this quick-reference guide is to assist the operator in setting up and running a HAAS mill in as little time as possible. Refer to the Operators Manual for more programming and operating information.  
**IMPORTANT!!** Be sure ALL Installation procedures have been completed before operating the machine.

## POWERING ON THE MACHINE

- Press the green **POWER ON** button in the upper left hand corner of the control.
- With the front doors closed, press the **POWER UP / RESTART** button. This will zero return all available axes.



## CREATING A PROGRAM

- Press the **LIST PROG** key. This will show a list of any programs presently in the control.



- Press the **O** (letter, not number) key, then nnnn, where nnnn can be any four numbers. Then press the **WRITE** key. This will create a program named Onnnn. Program names must follow this format.



The new program may now be created on the line following the program number. All further entries are made by typing a letter followed by a numeric value and pressing the **WRITE**, **INSERT** or **ALTER** key.

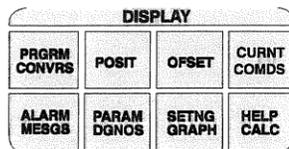


To make changes to an existing program, use the following keys:

- The **INSERT** key will insert any text to the right of the current cursor location.
- The **ALTER** key will replace highlighted information with the desired text.
- The **DELETE** key will delete any highlighted text.

## LOCATING X AND Y ZERO POINTS OF THE PART

- Use an edge finder or indicator to locate the part zero point with the jog handle.
- Press the **OFFSET** key and **PAGE UP** until the Work Coordinate page appears.



CONTINUED

## GENERAL INFORMATION

### HELP FUNCTION

This function is selected by pressing the **HELP/CALC** display button. This will bring a mini-manual up on the screen. The A.Z keys can then be used to find specific information on 26 topic areas.

- Pressing **HELP/CALC** key a second time will select the calculator function.

### MANUAL DATA INPUT (MDI)

Another way of writing and editing programs, without using the **EDIT** mode is in **MDI** (Manual Data Input) mode. **MDI** is a scratch pad memory that can execute many lines of instruction without having to disturb your main program in memory.

- A program in **MDI** can be saved as a normal program by placing the cursor at the beginning of the first line, typing Onnnn (new program number), then pressing **ALTER**. This will add that program name to the program list and clear **MDI**.

### EMERGENCY STOP

This button will instantly stop all motion of the machine including the servo motors, the spindle, the tool changer, and the coolant pump. It will also stop any auxiliary axes.

### CYCLE START

This button will start a program running in **MEM** or **MDI** mode, continue motion after a **FEED HOLD**, or continue after a **SINGLE BLOCK** stop.

### FEED HOLD

This button will stop all axis motion until the **CYCLE START** button is pressed. **FEED HOLD** will not stop the spindle, the tool changer, or the coolant pump. It also does not stop motion of any auxiliary axes.

- Use the cursor arrows to get to G54 X or any of the other available work offsets.
- Push the **PART ZERO SET** key and the X-axis value will be stored as this offset.



- The cursor will automatically move to the G54 Y location. Repeat the steps above to set the G54 Y.
- Usually the Z axis value will not have to be set and should be zero.
- Repeat the steps to set the G54 A.

### LOADING A TOOL INTO THE TOOL CAROUSEL

- Press the **MDI** button. Enter "T1", then press the **ATC FWD**, instead of **WRITE** key.



- Press the **TOOL RELEASE** button, then place tool #1 into the spindle.

### SETTING THE TOOL OFFSETS

- Press the **OFFSET** key and **PAGE DOWN** to the Tool Offset page. Cursor to Tool #1.
- Use the jog handle to accurately position the tool tip to Z0. The top of your part will usually be Z0.
- Press the **TOOL OFFSET MESUR** key and the Z value will be stored in tool offset #1.
- Close the machine doors. Press the **NEXT TOOL** key and the Z-axis will retract to tool change position and tool #2 (empty) will be installed in the spindle.
- Put tool #2 into the spindle and jog to Z0 as you did for tool #1. The cursor will automatically be on Offset #2, so press the **TOOL OFFSET MESUR** key. Close the machine doors and press **NEXT TOOL**.
- To return to tool #1, press the **MDI** key, enter T1, and press **ATC FWD**.

### RUNNING A PROGRAM

- Press the **LIST PROG** key. Use the cursor to find the desired program and then press the **SELECT PROG** key. The program list will include the program name and the first comment.
- Press the **CYCLE START** button to begin execution. To start a program other than at the beginning, scan to the desired starting place using the down arrow or **PAGE DOWN** key, and then press the **CYCLE START** button.

The **GRAPHICS** function is a visual dry run of your part program without moving the axes and risking tool damage from programming errors. To select **GRAPHICS** mode press **MEM** or **MDI** and the **SETG / GRAPH** key twice.

## GENERAL INFORMATION

### CURSOR KEYS

These keys give the user the ability to move to various screens and fields in the control. They are used extensively in CNC programs.

- The **HOME** key generally moves the cursor to the topmost item on the screen.
- The arrow keys move one item, block or field.
- The **PAGE UP** and **PAGE DOWN** keys are used to change displays up or down one page in the editor, or zoom in or out when in graphics.
- The **END** key generally moves the cursor to the bottom most item on the screen.

### JOG KEYS

These keys give the user the ability to jog a certain axis in either direction.

- When a key is pressed briefly, that axis is selected for use by the jogging handle.
- When a key is pressed and held down, that axis is moved as long as the key is held down.
- If a "+" key is pressed and held, that axis is moved so that the tool position is changed in a positive direction relative to the work coordinates.
- If a "-" key is pressed and held, the axis is moved so that the tool position is changed in a negative direction relative to the work coordinates. The jog keys are locked out if the machine is running.
- The A axis key selects the B axis when used with the **SHIFT** key and the control is configured with a fifth-axis option.
- When the **JOG LOCK** key is pressed prior to one of the above keys, the axis is moved in a continuous motion without the need to hold the axis key depressed. Another press of the key stops jogging motion.



EDIT	To edit a program already in memory
MEM	To run a program stored in memory
MDI / DNC	To directly run a manually entered program or to select DNC mode
HANDLE JOG	To use jog keys or jog handle
ZERO RET	To establish machine zero
LIST PROG	To list, send, or receive programs

The Graphics simulation mode is entered with the DISPLAY select buttons.

\* In MEM or MDI mode, a program can be started with the CYCLE START button. While a program is running, you cannot change to another mode; you must wait until it finishes or press RESET to stop the program.

When already in MDI, a second push of the MDI button will select DNC if the DNC mode is enabled by settings and parameters in your machine.

In any of the above modes, you can select any of the following displays using the eight DISPLAY buttons:

PRGRM / CNVRS	To show the program selected
POSIT	To show the axes' positions
OFSET	To show or enter working Offsets
CURNT COMDS	To show Current Commands and times
ALARM / MESSAGES	To show Alarms and User Messages
PARAM / DGNOS	To show Parameters or Diagnostic data
SETNG / GRAPH	To show or enter Settings or to select Graphics simulation mode
HELP / CALC	To show the Help data and Calculator

In addition to the above displays, when a program is already running, you may press LIST PROG to select a list of the programs in memory. This is useful to determine what programs can be edited in BACKGROUND EDIT. BACKGROUND EDIT is selected from the PROGRAM DISPLAY.

\* All operation of the CNC is controlled from the operator's panel. The control panel is composed of the CRT display, the keypad, On/Off switches, Load meter, Jog handle, and EMERGENCY STOP, CYCLE START, and FEED HOLD buttons.

The **keypad** is a flat membrane type that requires approximately eight (8) ounces of pressure. The **SHIFT** button replaces the function of the numeric buttons with the white characters in the upper left corner. The **SHIFT** button must be pressed once before each shifted character. Pressing the **SHIFT** button twice will turn off shift.

The **jog handle** is used to jog one of the axes. Each step of the crank can be set to 0.0001, 0.001, 0.01 or 0.1 inch (0.001, 0.01, 0.1, or 1.0 degree per step for a rotary axis). When using metric units, the smallest handle step is 0.001 mm and the largest is 1.0 mm. The handle has 100 steps per rotation. It can also be used to move the screen cursor while in EDIT mode, or to change feed/spindle overrides by  $\pm 1\%$ .

The EMERGENCY STOP button will instantly stop all motion of the machine, including the servo motors, the spindle, the tool changer, and the coolant pump. It will also stop any auxiliary axes.



CYCLE START will start a program running in MEM or MDI mode, continue motion after a FEED HOLD, or continue after a SINGLE BLOCK stop. The CYCLE START button on the optional remote jog handle performs exactly the same functions.

FEED HOLD will stop all axis motion until the CYCLE START is pressed. The FEED HOLD button on the optional remote jog handle will perform exactly the same functions.

#### WARNING!

FEED HOLD will not stop the spindle, the tool changer, or the coolant pump. It will not stop motion of any auxiliary axes.

The optional **Memory Lock Key Switch** will prevent the operator from editing programs and from altering settings when turned to the locked position.

The following describes the hierarchy of locks:

- Key switch locks Settings and all programs.
- Setting 7 locks parameters; parameters 57, 209 and 278 lock other features.
- Setting 8 locks all programs.
- Setting 23 locks 9xxx programs.
- Setting 119 locks offsets.
- Setting 120 locks macro variables.

The SINGLE BLOCK button on the keypad will turn on and off the SINGLE BLOCK condition. When in SINGLE BLOCK, the control will operate one block and stop. Every press of the START button will then operate one more block.

The RESET button on the keypad will always stop motion of the servos, the spindle, the coolant pump, and tool changer. It will also stop the operation of a running program. This is not, however, a recommended method to stop the machine as it may be difficult to continue from that point. SINGLE BLOCK and FEED HOLD provide for continuation of the program. RESET will not stop motion of any auxiliary axes but they will stop at the end of any motion in progress.

#### Function Buttons

The **F1** Button: In EDIT mode and PROGRAM DISPLAY, this will start a block definition. In LIST PROG mode, F1 will duplicate a program already stored and give it a new name from the command line. In offsets display, F1 will set the entered value into the offsets.

The **F2** Button: In EDIT mode and PROGRAM DISPLAY, this will end a block definition.

The **F3** Button: In EDIT and MDI modes, the F3 key will copy the highlighted circular help line into the data entry line at the bottom of the screen. This is useful when you want to use the solution developed for a circular motion. Push INSERT to add that circular motion command line to your program. In the Calculator Help function, this button copies the value in the calculator window to the highlighted data entry for Trig, Circular, or Milling Help.

The **F4** Button: In MEM mode and PROGRAM DISPLAY, this will select either BACKGROUND EDIT or PROGRAM REVIEW. BACKGROUND EDIT is selected by entering **Onnnnn** with the program number to edit. Program review is selected with just F4. Program review shows the running program on the left half of the screen and allows the operator to review the program on the right half of the screen. In the Calculator Help function, this button uses the highlighted Trig, Circular, or Milling data value to load, add, subtract, multiply, or divide with the calculator.

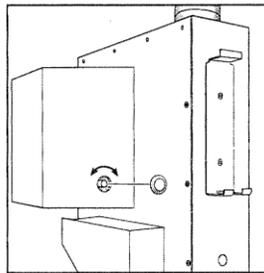
**REAL-TIME CLOCK**

Displays current Date and Time. The date and time as supplied by the real-time clock are displayed on the diagnostics screen.

Alarm history tagged with date and time. The alarm history is now displayed with the date and time that each alarm occurred. See the Alarms section on how to access alarm history.

Macro variables. Macro variable #3011 contains the date in the format yymmdd (two digit year\* 10000+ month\* 100+ day). Macro variable #3012 contains the time in the format hhmmss (hours\* 10000 + minutes\* 100 + seconds).

Parameter output contains date and time. When outputting a parameter file to a floppy disk or the serial port, it will contain two new comments near the top containing the current date and time.

**DISPLAY BRIGHTNESS ADJUSTMENT**

Remove plug to access the brightness adjustment knob. Be sure to replace plug.

**2-11 KEYBOARD**

The control panel keyboard is divided into nine separate regions. They are:

RESET keys	Three (3) keys
FUNCTION keys	Eight (8) keys
JOG keys	Fifteen (15) keys
OVERRIDES	Sixteen (16) keys
DISPLAYS	Eight (8) keys
CURSOR keys	Eight (8) keys
ALPHA keys	Thirty (30) keys
MODE keys	Thirty (30) keys
NUMERIC keys	Fifteen (15) keys

The following are short descriptions of the control panel keys' usage.