WinCNC Controller 2.4
Revision 3

Copyright © 1999 - 2009
Microsystems World CNC, LLC
14 East Lincoln Street, Buckhannon, WV 26201
(304) 472-7206

Microsystems World CNC, LLC makes no warranties either expressed or implied, including any implied warranties of fitness for a specific purpose, for this software. This package is provided solely on an "as is" basis. All risk involved in the use of this software is the responsibility of the purchaser.

Program errors, improper software configuration, hardware failures, or other problems can result in damage to the machinery being controlled. Failure of limit switches, cabling, or port components can result in machine movement past limits, potentially damaging the machine and endangering operators. It is imperative that a separate Emergency Stop system be in place, which bypasses all computer control. Software features to abort motion from the keyboard and limit switch inputs are provided as conveniences only, and must not be relied upon to stop motion in an emergency.

Microsystems World CNC, LLC disclaims all liability for any and all damages, including incidental and consequential damages in connection with use of this software. The sole and exclusive liability of Microsystems World CNC, LLC will be limited to the purchase price of this software package.

If you do not understand and agree with these disclaimers, please do not use this product. Return all materials to the place of purchase for a full refund.

WinCNC Controller Contents

Installation

CAUTION: DO NOT USE THIS SOFTWARE ON A COMPUTER CONNECTED TO A MACHINE UNLESS THERE IS AN EMERGENCY STOP READILY AVAILABLE WHICH WILL INDEPENDENTLY STOP THE MACHINE.

Software features to abort moves from the keyboard and limit switch inputs can NOT be relied upon in emergency situations and are provided as conveniences only!

Requirements

WinCNC Controller 2.4 requires a Windows XP/Vista (32 Bit) computer system with Pentium IV or faster processor, 1GB or more of RAM, a hard drive, and an enabled USB port. Slower computer systems may result in abnormal acceleration and other problems. An Internet connection is highly recommended to ensure proper technical support and access to Windows Updates from Microsoft
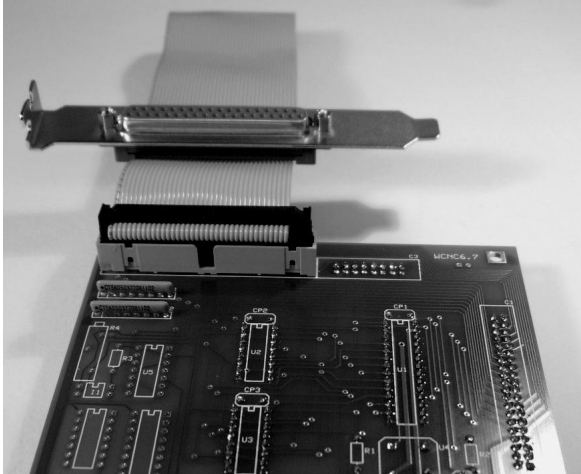
Hardware Installation and Configuration of the PCI-7200 Card

**It is highly recommended that you use an anti-static wrist strap before handling any components.

Below are the components for the PCI-7200 installation. These include a PCI-7200 card, a Daughter Board, and a ribbon cable connector.
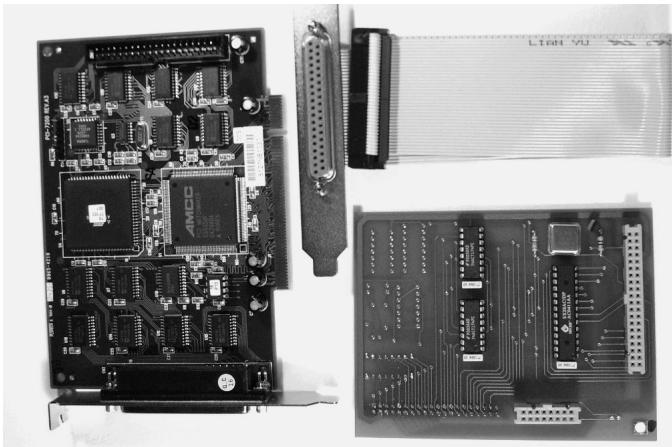
Insert the ribbon cable connector into the Daughter Board.

Below is the PCI-7200 card on the left, the Daughter Board on the right and the spacer/screw assembly. Note the matching holes in the Daughter Board and the PCI-7200 card. The small washer is included for a precise fit and must be installed between the boards, at one end of the spacer or the other. This may already have been done prior to shipping.
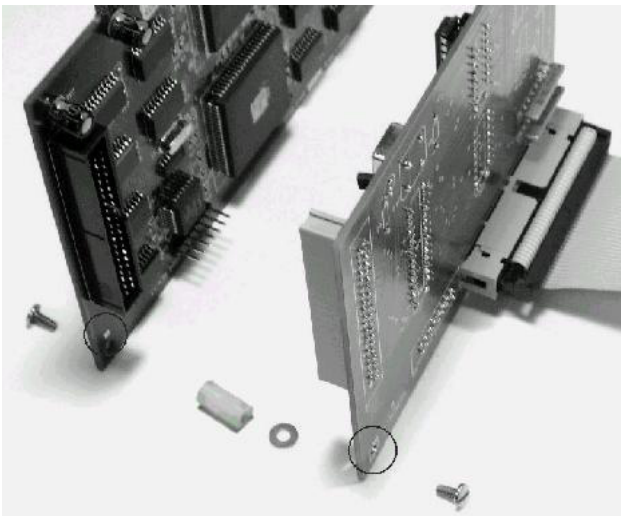
Insert the Daughter Board on to the 40-pin connector of the PCI-7200 card. Attach the two cards with the included spacer, washer, and screws. This will insure constant connection between the two boards.
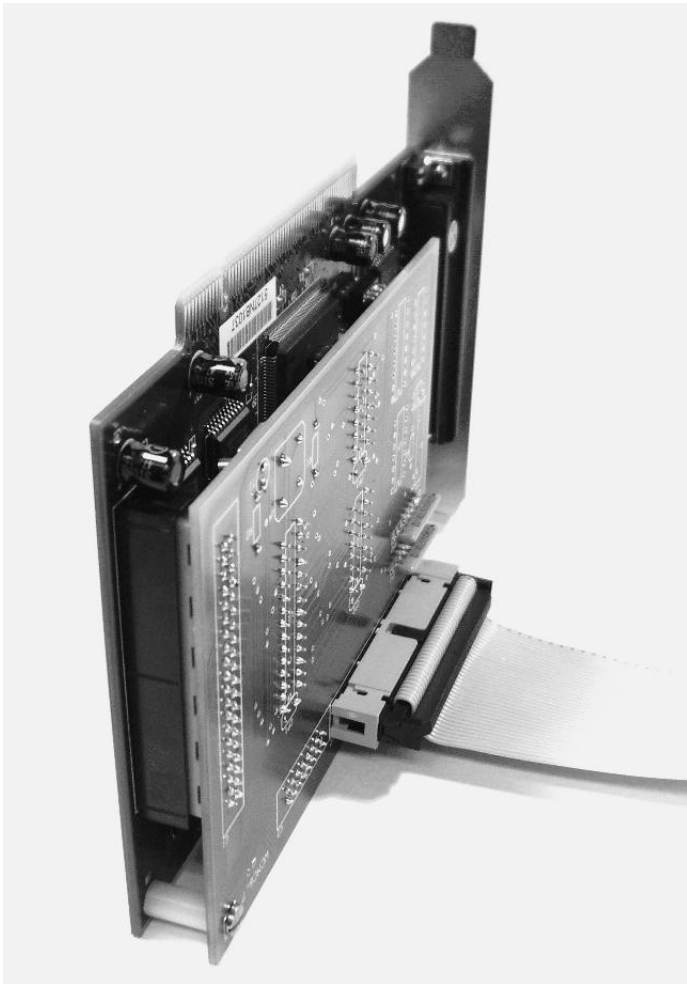
Power down and unplug the PC from any power.

Insert the PCI-7200 card into an empty PCI slot on the motherboard. Install the extra 37-pin connector into an empty cover slot. Install the extra 37-pin connector into an empty cover slot so that when you are looking at the back of the computer, you have two identical 37-pin ports, one being blue and the other black. **The blue DB37 connector is the daughterboard, and the black is the PCI-7200 card.**

Very Important! Install the 37 pin cable from the control box into the daughter card (BLUE) connector, not the PCI (BLACK) connector. The cards have different power and ground pins and the cards will be damaged if connected incorrectly.

Make sure the PCI-7200 card is using its own IRQ. IRQ settings must be altered from the BIOS to insure no conflicts occur between the PCI-7200 card and other system resources. Changing these settings from within Windows alone will not insure this.

After installing your PCI-7200 card and the extra connector, boot up your computer for driver and software installation.

**Note: When your computer boots up, you may get a hardware installation wizard. Close it. You will be installing the driver after the software is installed.

Install software

Insert the WinCNC Controller CD-ROM into the CD-ROM drive. If the installation does not automatically start, you will need to manually run the "Setup.exe" file on the disk. To do this, click "START",

then "My Computer", then right click on the 'D:, E:, or F:" drive that is labeled 'WinCnc.". You should get a menu that pops up.
On that menu, click "Explore". Once the files on the CD are visible, double click the "Setup.exe" file. Follow the prompts of the installation wizard.   The install routine will create the specified directory if it does not exist and copy the necessary files into it. If there is a previous installation of WinCNC controller the files will be overwritten.

Driver Installation for PCI-7200 Card:

**Normally, driver installation is done during installation.

A driver is required for the PCI-7200 card. The driver is included in the PCI-7200 folder on the root directory of the WinCNC installation disk for user convenience. If the installation disk isn't available, the driver can be downloaded from the Microsystems World CNC web site. After connecting to the internet, go to www.wincnc.net and click on the "Support" link at the top left of the page, then click "Downloads". Next, find the "CNC Windows/" link and click that, then click "Device Manager", then click the version driver you need (either 2.3 or 2.4).   Most new installations will require the most recent version. After selecting which version you need, click "PCI7200", and then click "PCI7200.exe".   Next click "Save" and browse to your desktop, and then click "Save" again. Once the download is complete, you should have an option to "Open" the file. You need to make sure at this point, that you "extract all files" into the WinCNC folder on your C:\ drive.

Go into the PCI7200 folder inside of the WinCNC folder and find "uninstdrvr.bat". Run that file, and be careful not to close it before it is done. Watch for prompts asking you to press a key to continue, and such. After that is done, run "instdrvr.bat". Same thing....either or both of these may take longer than you think, and may pause and not do anything long enough for you to think it might not be working correctly. Sometimes those two files run fairly quickly, but there have been instances when they take a couple of minutes to complete.

After running both files, reboot your computer and try opening WinCNC.

Run the program by double clicking on the WinCNC icon on your Windows desktop.

Overview

WinCNC Controller has advanced features to provide the smoothest possible cuts. G-Code input is constantly buffered to 'vector match' moves. This means your machine only slows down when it needs to and then only as much as needed to stay within the acceleration parameters programmed. WinCNC Controller also does S-Curve acceleration through arcs and matches arc tangential velocities with straight moves to provide smooth arc moves. The result of these features is simply smoother cuts and less need for finishing operations. WinCNC Controller has a highly customizable, user-friendly interface that lets you take control of your machine, providing features previously only found on custom industrial controllers.

Any stepper, micro-stepper, or servo system with step and direction inputs can be interfaced using the included PCI card and microprocessor daughter board, or one of several available PCI I/O expansion cards. In addition to axis motion, up to 128 auxiliary outputs and 128 separate inputs may be defined. The intuitive user interface is easy to learn and convenient to use. All the features you need and expect are readily accessible via industry standard G-Codes. Simple keystrokes provide the ability to start, pause, restart, jog, and rapid transit any axis without navigating multiple screens and windows. Programs may be run from any line as needed. The programmed feed rates may be adjusted from 1% to 200% without stopping machine motion.

Run the program by double clicking the WinCNC icon on your Windows desktop. Program options may be selected from the pull down menu using the mouse, ALT+ menukey, or function keys while the machine is stopped. Files can either be typed into the command line or you can use the File menu's open option to bring the file name to the command line. Using the open command does not automatically run the file, it just brings the name to the command line.

Screen Display

The following section contains a sample of the WinCNC software display. Common components are labeled to provide clarity.
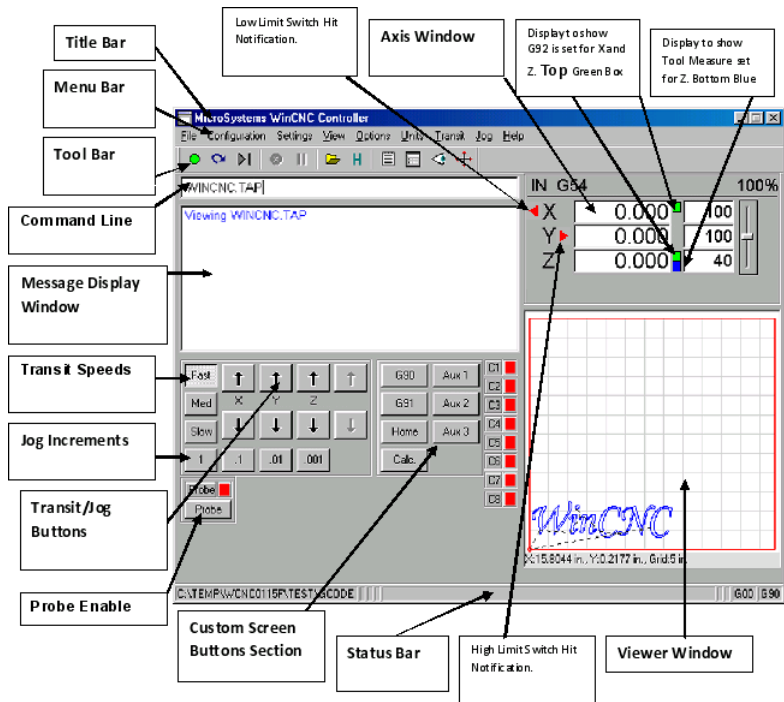
Screen Display Component Breakdown

Menu Bar

The Menu Bar contains many of the main features of the WinCNC software. It is broken down into several generalized sections, as shown below. The menu bar selections are described in the following pages. If a menu option has a shortcut key, the shortcut key combination is listed in parenthesis () after the name of the menu option.

File Configuration Settings View Options Units Transit Jog Help

File Menu

Open (CTRL+O) - Opens a file.

Edit (CTRL+E) - Opens the editor. If a filename is in the command line, the editor opens that file.

Simulate (CTRL+S) - Simulates a file running to check for errors and estimate run time.

View (CTRL+V) - Views a file in the viewer window.

Restart (CTRL+R) - Opens the file restart dialog box to restart a file.

Create Home File (CTRL+H) - Creates a home file at the current machine position.

Import => DXF - Allows the user to import DXF (Drawing Exchange Format) file.

Import => HPGL - Allows the user to import HPGL (Hewlett-Packard Graphics Language) file.

*Note: The DXF and HPGL import feature is intended for cutting simple designs. WinCNC does not compensate for tool diameter when converting these files. For more complicated design, where intelligent tool pathing is important, please use a compatible CAD/CAM program.

Digitize => Manual - Enable/Disable the Manual Digitizing Toolbar.

Digitize => Automatic - Automatic digitizing (L802, L803).

Digitize => Frame - Skeletal Digitizing (L810). This scanning method scans a center line or spine, and horizontal lines or ribs of the object.

Digitize => Outline - Creates an outline trace around an object.

Digitize => Laser => Run Scan - Opens the laser digitization dialog box to set up laser scanning parameters.

Digitize => Laser => Reprocess Data - Opens a data file from a previous laser scan used to re-generate a 3D object with different filtering options to improve the quality of the 3D object.

Exit - Closes WinCNC.

Configuration Menu

Machine Settings - Opens the WINCNC.INI file in an editor and closes WinCNC until the editor is closed. After closing the editor WinCNC will restart activating any changes to the INI file. In order to use this feature, the presence of the MachineXonfig.exe file is required. (This is an application that edits your .ini file, and may or may not be provided by your machine manufacturer.)

D/A Calibration - Calibration Settings for D/A (Digital to Analog) spindle speed control. This can only be used if D/A has been activated in the WinCNC Program.

Settings Menu

Positions - Opens the positions dialog box with options to view and edit stored positions. (Ex: G53 P# to call position in G-Code)

Home Positions - Opens the positions dialog box with options to view and edit stored home positions. (Ex: G0 H# to call home pos.)

Tool Positions - Opens the tool positions dialog box with options to view and edit stored tool positions for use with automatic tool changers. (This requires an additional software feature to be enabled)

Tool Library - Edits and views the defined tool library.

Local Coordinates - View XYZ coordinates.

View Menu

Resolution - View current resolution settings for each axis.

Acceleration - View current acceleration settings for each axis.

Limits - View current limit settings for each axis.

Inputs - Views real-time input states.

Outputs - Views real-time output states.

Main Toolbar - Enable/Disable the main toolbar.

Refresh View (F5) - Refreshes the screen.

Clear Screen (CTRL+C) - Clears the message display window.

Options Menu

Soft Limits (CTRL+L) - Enable/Disable the software limits.

Keyboard (CTRL+K) - Enable/Disable the keyboard transit/jog control used to move the machine with the keyboard.

Buttons - Used to customize user buttons.

Auto Repeat - Auto populate the command line with the last command executed.

Auto Run - Enable/Disable the auto run feature.

Auto Preview - Enable/Disable the auto preview feature.

Single Step (CTRL+P) - Enable/Disable single line execution of G-Code files.

Units Menu

Inches - Sets the unit of measure to inches.

Centimeters - Sets the unit of measure to centimeters.

Millimeters - Sets the unit of measure to millimeters.

Transit Menu

Slow (F2) - Sets the transit speed to slow.

Medium (F3) - Sets the transit speed to medium.

Fast (F4) - Sets the transit speed to fast.

Jog Menu

0.001 (F6) - Sets jog increment to 0.001 units of measure.

0.01 (F7) - Sets the jog increment to 0.01 units of measure.

0.1 (F8) - Sets the jog increment to 0.1 units of measure.

1 (F9) - Sets the jog increment to 1 unit of measure.

Custom (F10) - Allows the user to set a custom jog increment.

Help Menu

Help Topics (F1) - Activates the in-program help menu.

Update Program - Opens the update utility.

About - Displays the WinCNC "About" box which contains important information about your specific software package. The About box displays your security key serial number, the software version number, your user level, the maximum number of axes, the table size limitation, and which additional features you have enabled. If you do not have a feature enabled that you would like or need, you can contact your software vendor for an upgrade.

Tool Bar

The tool bar is a collection of shortcut buttons that perform specific actions. The toolbars in WinCNC are floating and can be positioned or docked to the user's preference. The image below shows both the standard and manual digitizing toolbars both docked in the normal toolbar area. The additional images show each toolbar separately as a floating toolbar. The explanation of each button on the toolbar is as follows. The explanations move sequentially from left to right along the toolbar.

Start Motion - This button will begin the command or job listed in the command line.

Restart - Allows you to pick a job file and the line to start that file on. This is useful if you want to skip over lines in a job file, or if you have aborted a job and want to start back at the point you aborted from. You can also choose to run the file in single step mode.

Single Step - When selected, this allows you to execute one line of a G-Code file each time you hit ENTER.

Abort Motion - Aborts a command or job that is running.

Pause/Continue Motion - Pauses/Continues a command or job that is running.

Open File - Opens a browse box that is used to open a job file.

View History - Opens the command history box, which allows a user to execute a command used previously. When an error is found in the WINCNC.INI file, the line containing the syntax error is displayed here in red when WinCNC starts.

Edit - Opens the default editor specified in the WINCNC.INI file. WinCNC uses Notepad by default. The editor can be used to open job files, listed in the command line in the editor.

Simulate - Simulates a file to check for errors and run-time.

View - Opens a file for viewing in the viewer window

Soft Limits - Enables/Disables Softlimit features

Manual Digitize Buttons - The seven buttons on the manual digitization toolbar are for use only with the Manual Digitize feature, and will only be visible after showing the manual digitize toolbar under the View->Toolbars section of the menu bar.

Start a Manual Digitized File - Starts a manual digitized file and enables the manual digitize mode.

Add a Rapid Move - Adds a rapid move to the manual digitized file.

Add a Feed Move - Adds a feed move to the manual digitized file.

Add an Arc Point - Used to add arcs into a manual digitized file.

Close Shape - Used to close the last move in a shape without moving the machine.

Undo - Used to undo previous moves.

Save - Allows you to save the manual digitized file in either G-Code or DXF file format.
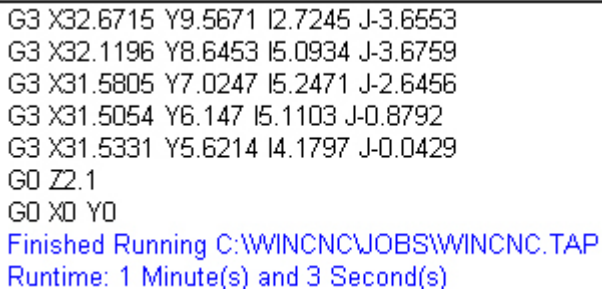
Command Line

```
WTEST.TAP
```

The command line is the input line for users to enter commands or job files to be executed.

Message Display Window

```
G3 X32.6715 Y9.5671 I2.7245 J-3.6553
G3 X32.1196 Y8.6453 I5.0934 J-3.6759
G3 X31.5805 Y7.0247 I5.2471 J-2.6456
G3 X31.5054 Y6.147 I5.1103 J-0.8792
G3 X31.5331 Y5.6214 I4.1797 J-0.0429
G0 Z2.1
G0 X0 Y0
Finished Running C:\WINCNC\JOBS\WINCNC.TAP
Runtime: 1 Minute(s) and 3 Second(s)
```

The message display window is the main output window displaying the commands that have been executed, messages to the user, or errors that have occurred during an operation.

Axis Window

IN	G54	G43	G92		100%
X▶			0.000 ◻	700	
◀Y			0.000	700	
Z			0.000 ◻	50	
W			0.000	50	
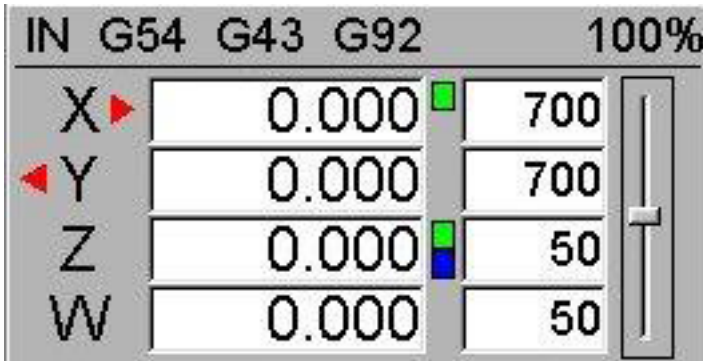
The axis window is the primary display for current information about each axis on your machine. The current positions and velocities for each axis are displayed in the text boxes. Displayed above the position and velocity boxes are the current units of measure, any currently active modes, and the current feed rate override percentage. Red triangles to the left and right of each axis label indicate low or high limit switch hits. A red triangle to the left of an axis label indicates a low limit switch hit, and to the right indicates a high limit switch hit. A green box displayed to the right of the position box indicates a temporary workspace (G92) setting, and a blue box indicates a tool length measure or workspace setting. To the far right side of the axis window is the feed rate override control slide bar.

Transit Speed Buttons

These buttons allow you to specify the transit speed of your machine.

Fast

Fast transit is the default.
In the cncscrn.ini file, change the "default speed" field to 1, for the speed you would like to be default.

Med

Slow

Ex: for default medium speed, use the following:
"JogMode", 0,5,35,25,7,**1,**"Med","","",-1,-1,"med.bmp"

All others must stay 0.

Jog Increment Buttons

1	.1	.01	.001
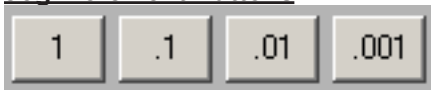
These buttons allow you to specify the increment of a jog move.

Transit/Jog Buttons
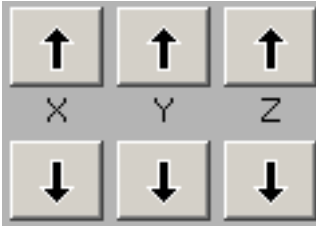
These buttons are what allow you to initiate a manual transit or jog movement from the console of the computer that runs the machine. In transit mode, holding the button down will continuously move the machine. In jog mode, each time a button is pressed the machine will move according to the jog increment that has been selected. Holding the button down in jog mode will move the machine only once.

Custom Screen Buttons Section

This section of controls is a customizable button section that allows the user to place shortcut buttons to activate heavily used commands.

Note: See Section 4 Screen Configuration

Status Bar

The status bar is the label along the bottom of the WinCNC window that looks similar to the one shown below. This bar provides the user with the status of several features in WinCNC. Each section in the example below is labeled for clarity.

Elapsed Time for Current File

Current Line in Job File

G00/G01 Mode Indicator

WTEST.TAP | Line:2 | 00:09 | 0% | | Tool 2 | No Tool | G00 | G90

Current File

Percentage of File Completion

Tool Changer ON/OFF and current tool being used

G90/G91 Mode Indicator
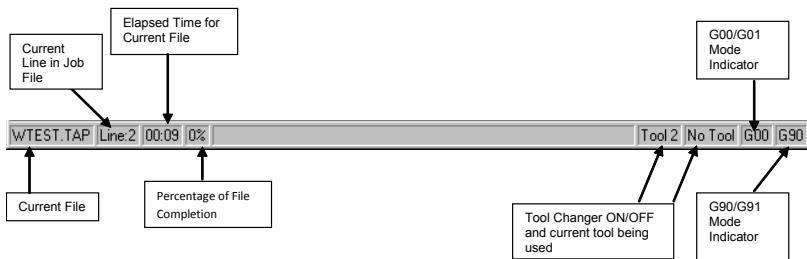
* Note: There are TWO tool changer indicators

Shortcut Keys

File Menu Shortcuts	
Open	Enter (with blank command line), CTRL+O
Edit	CTRL+E
Simulate	CTRL+S
View	CTRL+V
Create Home File	CTRL+H
View Menu Shortcuts	
Refresh View	F5
Clear Messages	CTRL+C
Options Menu Shortcuts	
Keyboard	CTRL+K
Soft Limit	CTRL+L
Transit Menu Shortcuts	
Slow	F2
Medium	F3
Fast	F4
Jog Menu Shortcuts	
.001	F6
.01	F7
.1	F8
1	F9
Custom	F10
Help Menu Shortcuts	
Help	F1

Shortcut Keys continued

Tool Bar Shortcuts	
ESC	Aborts the current file or command
SPACE	Pauses a file or command
ENTER	Starts/restarts a file or command
TAB	Opens the command history box
CTRL+R	Opens the restart file box
Feed Rate Override Shortcuts	
INSERT	Increases override rate
DELETE	Decreases override rate
CTRL+either	Resets feed rate to 100%. No override settings
Manual Digitize Shortcuts After opening the Manual Digitize toolbar and clicking the green start button, the following shortcut key combinations become active for adding moves to the digitized file.	
Add Rapid Move	CTRL+J
Add Feed Move	CTRL+F
Add Arc Midpoint/Endpoint	CTRL+A
Undo Last Move	CTRL+U
Close Current Shape	CTRL+D
Save	CTRL+W
Spindle Speed and Laser Power Shortcuts	
Spindle Speed Alone or Laser Power Alone	+ - (plus and minus)
Spindle Speed and Laser Power Together	Laser Power uses + - Spindle Speed uses Shift + -

Limit Switches

The default channel settings will normally provide proper interfacing with the machine's limit switches. Before attempting to move the machine, verify that all limit switches are functioning properly. Limit switch status is displayed in the position window using a red triangular indicator to the left or right of the axis label.
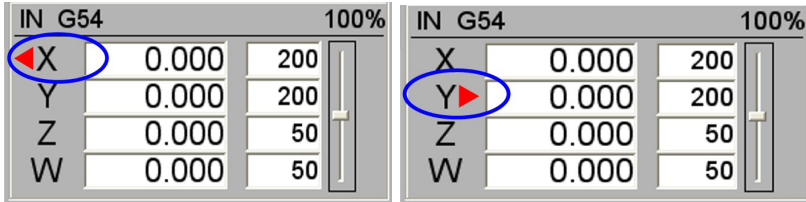
The indicator to the left of the axis label shows low limit status for that axis. The indicator to the right of the label shows high limit status. If the indicator is present then the limit switch is closed, otherwise it is open. The lim_mode setting is used to control how WinCNC responds to limit switch triggers.

First toggle your limit switches by hand. Verify that the appropriate limit display toggles. If it does not toggle then you need to adjust WINCNC.INI. See the WINCNC.INI reference sections of the manual if the default settings are incorrect for your machine.

It is recommended that you verify limit switch operation at the beginning of each session. Do not attempt movement until the limit switch displays toggle correctly. Once the limits are working try a small move. If nothing moves, the wrong axis moves, or movement is in the wrong direction then adjust WINCNC.INI.

Perhaps the easiest way to do preliminary testing of movement is with the Jog mode. Select "Jog .1", and select the KEYBOARD option under the OPTIONS menu. If you are using the default jog key configuration, use the LEFT and RIGHT arrow keys to move X, the UP and DOWN arrows to move Y, the PAGEUP and PAGEDOWN keys to move Z, and the HOME and END keys to move U. Otherwise use the buttons that are defined for your system.

Homing the Machine

When WinCNC Controller is started the display is set to the last known position. If the machine has been moved manually or has drifted while powered down, this position will not be accurate. The G28 command must be used to home the machine. G28 moves the WZ motors up to the hi limits, then moves the XY motors to their low limits. The heads are then moved away from the limits by the values specified in WINCNC.INI and each axis is set to zero. This position is Machine Zero.

It is important to use G28 to set Machine Zero since many WinCNC features are calculated from this position. Soft Limits and Boundaries cannot be used if Machine Zero is not properly set. G28 will search for the limits at 50" per minute. The "lim_step=" setting in WINCNC.INI can help minimize the impact to the machine when stopping at the limit switches.

G28 can also be used to home only specified axis. (i.e. G28Z homes only the Z axis).

Normal machine operation using WinCNC Controller would be:
> Start the program.
> Enter G28. Push ENTER. (Machine Goes Home)
> Enter part program to cut. Push ENTER. (Job file runs)

Feed Rate Override

WinCNC accepts feed overrides to increase/decrease the feed rate. The feed rate can be adjusted from 1% to 200% of the programmed rate. The override rate cannot be set higher than the max velocity or G0 rate for a given axis. Use the Insert key to increase and the Delete key to decrease the programmed feed rate, holding CTRL and pressing either Insert or Delete will result in the feed rate being reset to the programmed value. There is also a slider bar to the right of the feed rate screen display that can be used to initiate a feed rate override. The override feed rates will be displayed in place of the programmed feed rates in the display. The feed rate will change colors to depict that an override is in use. If the override rate is higher than the programmed rate the color of the feed rate display will be red, if it is lower than the programmed feed rate it will be blue.

Coordinate System

The Machine Zero (MZ) set by homing the machine becomes the anchor point for all positions specified in subsequent G-Code commands. This is also considered absolute zero, not taking into account any temporary homes (G92's) or tool measures.

Local Zeros (LZ) are set using the G92, G92.1 and G92.2 commands.

The position display box displays the current axis coordinates and the axis coordinate modes. The axis coordinate mode is displayed to the right of the axis position.

A green box indicates that the axis has a Local Zero applied. (G92, or temporary home).

A blue box on a horizontal axis indicates that a workspace other than G54 is in effect. (G55, G56, G57: ex: using W axis instead of Z).

A blue box on a vertical axis indicates that tool length measure is in effect.

The coordinate modes show how the axis values in a given G-Code command will be interpreted.

MZ Coordinates - values are relative to MZ. (Or absolute zero)

G92 Coordinates - values are relative to LZ specified by G92
(Ex: enter the command G1 X10 Y10, then enter. Next type G92, then enter. X & Y will now show as x0, Y0)

G55, G56, G57 Coordinates for horizontal axes - horizontal axes values are offset for vertical axes heads.

G43 Coordinates for vertical axes - vertical axes values are offset for M37 tool measure values.

G92 and G55, G56, G57, G43 - values are offset for both.

G91 Mode

G91 mode is called relative or incremental mode. In G91 mode values specify distances. For instance if the X position is currently 20 and G0 X8 is specified the machine would move +8 units of measure in X to X28.

A G-Code program written in G91 mode may be run from any position. Since the moves are relative, the starting position does not matter. The program will run properly from any starting position. However, to get the proper results, the machine must be set to the proper position in relation to the workpiece before the piece is run.

The axis coordinate status does not have any effect on how the program runs. It does however affect the coordinate display while the program runs.

G90 Mode

G90 mode is called absolute mode. In G90 mode values specify positions. If the X position is 20 and G0 X8 is specified the machine would move -12 units of measure in X to X8.

In G90 mode the situation is different. The initial of the machine position will not affect where the part runs since the first move will go to the absolute position specified on the table. Instead of setting the machine position properly before running a G90 program.

The G92 commands are used to change the coordinate system so that any position may be temporarily made to be Local Zero (LZ).

Most G90 programs are written relative to a starting position of X0Y0 with Z0W0 being either the bottom or top of the workpiece. All that is then necessary to run the program is to insure that the current LZ is set to match the program before running. There are several ways to do this.

The machine may be positioned to the proper position and the G92 command used to set the proper coordinates for running.

G90 Mode continued

Example: The workpiece is fixtured at X20Y20, is 1" thick and Z0 in the part file refers to the top of the workpiece.

G90	Commands in G90 Mode
G92	Turn off any current G92s
G0Z2	Set Z 2" above the table and 1" above the workpiece
G0X20Y20	Go to corner of workpiece
G92X0Y0Z1	Set this position to be 1" above LZ

The Axis Window shows:
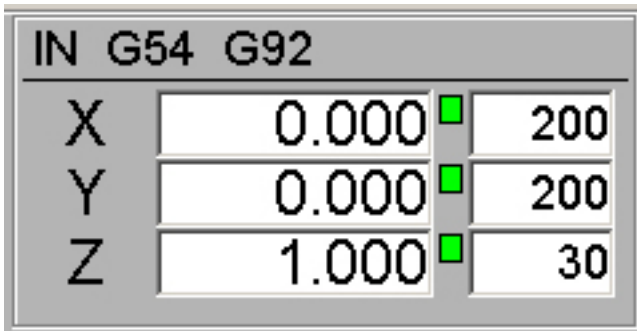
Alternately LZ may be set by shifting MZ the desired amount using G92.1. Using G92.1 it is not necessary to move to the workpiece first.

Example

G90	Commands in G90 mode
G92.1 X20Y20Z1	Move LZ 20" in XY and 1" above the table

The coordinate display shows the current position relative to the workpiece.

When writing a G-Code program there must always be a starting point. Typically it would be the lower left corner (in XY) of the piece to be machined and either the upper or lower surface of the workpiece or a position a known distance above the workpiece (ZW). In a G91 mode program this point does not need to be given a coordinate value.

In G90 mode this point is assigned a coordinate value (usually 0,0,0) and all positions are specified relative to this starting point.

When running the program the machine is positioned to this starting point and G92 is used to set position to the start position for the program. Alternately G92.1 may be used to shift MZ.

Workspace
The G54/G55/G56/G57 functions are used to allow for easy switching between workspace coordinates. Each G54/G55/G56/G57 line you wish to use must be specified in the WINCNC.INI file.

WINCNC.INI Setup: For descriptions see the Systems Configuration Section of the manual.
G54=X# Y# T# A# R# O#

After configuring the G54/G55/G56/G57, the G54/G55/G56/G57 commands can be used to switch workspace coordinates, or for rotating a rotational axis.

If using the type 0 G54/G55/G56/G57 for switching vertical heads just use the G54/G55/G56/G57 command alone to switch workspace coordinates.

If using the type 3 G54/G55/G56/G57 for rotational axis command G54/G55/G56/G57 X# Y#, where X and Y are the positions to rotate towards.

Head Swap
L12 provides a means of cutting with a head not specified within a program and of using multiple heads simultaneously.

Example:
L12WZ moves Z whenever W is specified and moves W when Z is specified.
L12ZZ moves Z and W together whenever Z is specified.
L12 alone or L12ZW sets normal operation.

Tool Length Offset

M37 is used to measure tool length and set ZWUV workspace.

By using M37 it is not necessary to measure tool length for every workpiece thickness change. Using M37 in combination with Soft Limits and Boundaries also provides a means of protecting the table from being routed accidentally. After M37 is set G28 will set the ZW to the actual position of the tool tip above the table.

Example: (without automatic tool changer)
Move the Z head tool tip to the table.
Command M37 Z0.

M37 turns on G43 mode, indicating that tool length offsets are active. Tool length offsets can be disabled using G49 and re-enabled with G43 without re-measuring.

Soft Limit / Boundaries

Soft Limits and Boundaries are used to keep programs and command from moving into the limit switches. Values must be set in WINCNC.INI before using.

Soft Limits check absolute position command against the limit switch positions. If a position is commanded that would result in a limit switch being hit a limit error will be displayed and the move will not be run.

lolim=X# Y#	Distance from MZ (machine zero) to XY low limit switches less .1"
hilim=X# Y# Z# W#	Distance from MZ (machine zero) to XYZW high limit switches less .1"

ZW low limits normally are not specified since desired positions vary with tool length.

Boundaries check workspace positions. In G54 mode the Z head will not be allowed to leave the table. In G55 mode the Z head might move off the table but the W head will be kept on the table.

lobound=X# Y# Z# W#	Distance from MZ to XY table edge and top
hibound=X# Y#	Distance from MZ to XY table edge

ZW high boundaries normally are not specified since desired positions very with tool length.

ZW low boundary checking is available if M37 is used. Use M37 to set Z0W0 to the tabletop. The ZW hi limits and low boundaries are then both set to 0. This allows movement between the limits and the table.

Limits and Boundaries may be enabled or disabled in the Settings Menu and in WINCNC.INI using softlim=0 or 1 and boundary=0 or 1.

Repeat Command
The command or file name history may be viewed by pushing the TAB key. To repeat a command from the history list select the desired command or file name using the mouse, and then click OK.

Simulate
WinCNC supports the simulation of files to check for errors, estimate runtime, and check min/max positions. To use the simulate function of WinCNC make sure the file name is typed into the command line and instead of running the file, you will use the simulate call to simulate the file. This is done by using either the Simulate command found under the File menu, by pressing CTRL+S, or by pressing the simulate button on the toolbar. Once you initiate the simulate feature, the software will display a screen showing the specifics of the file simulated which includes the starting (x,y) positions, the ending (x,y) positions, the min and max (x,y) positions, the number of lines ran in the file, and the total time it will take to run the file.

NOTICE: The time function in the simulation feature does not take into account smoothing. If you have smoothing set up on your machine, the time it takes for the file to run should be less than what is displayed in the simulation data.

Viewer

WinCNC contains a built in viewer window that will allow you to display the output of G-Code files before the job is actually ran. The viewer can also display a G-Code file line by line as it is run by WinCNC. When the file is displayed in the viewer you can distinguish G0 moves from G1 moves by the color of the line. By default, G0 moves are shown as a black dashed line. G1 moves are displayed as a solid blue line. See the Screen Configuration section if a different color configuration is desired.

You can also re-center and zoom in/out of the display within the viewer. To view a file before you run it, enter the file name into the command line and then press CTRL+V, select View from the File menu, or press the viewer button on the toolbar. The file will be displayed line by line automatically when you run the file. Once the object is loaded into the viewer the following controls can be used:

Zoom In Incrementally - click the left mouse button.
Zoom Out Incrementally - click the right mouse button.
Selected Zoom - hold down the left mouse button and drag the box around the area to zoom in on.
Pan or Re-Center Object - hold CTRL and click either mouse button.
Reset Image - hold shift and click either mouse button.

To set up the viewer window you must include lines in both the WINCNC.INI and the CNCSCRN.INI files.

WINCNC.INI Setup (for descriptions see the System Configuration section of the manual)
table=X# Y# W# H# B#

CNCSCRN.INI Setup (for descriptions see the Screen Configuration section of the manual)
"Display", "Viewer", 370, 150, 270, 270

Pump Control

Wiring for the Daughter Board

Pulse Width Modulation
-Jumper Pin 1 of connector 4 (C4) on the daughterboard to
Pin 2 of C4.
-Disconnect trace between Pin 2 of C4 and Pin 3 of C4.
-Jumper Pin 3 of C4 to Pin 4 of C4
-Pin 11 of CN2 is the pump control signal.
-Pin 15 of CN2 is the pump reverse signal.

Frequency Mode
-Pin 11 of CN2 is the pump control signal.
-Pin 15 of CN2 is the pump reverse signal.

WINCNC.INI Setup (for descriptions see the System Configuration
section of the manual)
pumpcont=T# L# H# A# B# I# R#

Commands (for descriptions see the Commands section of the
manual)
L55, L56, L57, L58, L59

Buttons (see also the State Buttons section in the Screen Configu-
ration section of the manual)
Pump Mode On
"StateBtn", 280, 5, 50, 25, 4, 0, "Pump On", "L55", "", -1, -1, ""
Pump Mode Off
"StateBtn", 280, 35, 50, 25, 5, 0, "Pump Off", "L58", "", -1, -1, ""
Purge Mode
"StateBtn", 280, 65, 50, 25, 6, 0, "Purge", "L56", "L57", -1, -1, ""

Manual Digitize

WinCNC supports the ability to manually digitize a part and then output the file as either G-Code or DXF format. The manual digitize toolbar is started by clicking File->Digitize->Manual. The toolbar icons are displayed and summarized in the screen display section of the manual. The Manual Digitize feature supports the addition of rapid moves, feed moves, and arc moves. It also has support for undo, close shape, and save. To digitize a part manually, place the part on the table and transit the machine to the selected starting point and then enable the manual digitize mode.

Now that manual digitization is enabled, to add a move into your digital file, transit the machine to the desired point and then press either the add rapid (G0) or add feed (G1) button on the toolbar depending on the type of move needed. To add an arc, move to the beginning of the arc and add either a rapid (G0) or feed (G1) move, then transit to any point along the arc and press the add arc point button on the toolbar. The first time the add arc button is pressed it is setting a mid-point for the given arc. Once you have the midpoint selected transit to the end of the arc and press the add arc button again. This will complete the arc move and add it to your digital file. Continue transiting along the part adding in the required move types.

Once you have your part completely digitized click the save button on the toolbar. If you saved the digitized file as a G-Code formatted file, you can then use that file with WinCNC. If you saved the file as a DXF file, you can import this file into any CAD/CAM package that supports DXF file formats and make any needed adjustments.

The Manual Digitize feature of WinCNC also supports the ability to undo any previous moves during the digitization. If at any point you wish to go back before moving the machine or adding a move press the undo button in the toolbar and the machine will lift the vertical head, move back to a previous location, and lower the vertical head back to its initial position. The location the undo button moves to depends on the last type of move you made. If it was simply a transit move out into the table, undo will move the machine back to the point you started at. If undo is used after adding in a new move, the machine will move back to the last point that was added to the file.

<u>Manual Digitize continued</u>

The manual digitize feature also has support for closing shapes. The close shape button works by adding a feed style move from the current point to the first point. The close shape feature handles only 1 shape at a time, and any rapid moves that are added repositions the first point of the shape to the ending point of the rapid move. If no rapid moves are added, the starting position for the shape remains at the initial point.

For example: Two squares that are 10"x10" with a 5" separation along the X Axis would be digitized as follows. Start by transiting the machine the desired starting point and enabling the manual digitize feature. Once it is enabled, transit the machine to X10Y0 and add a feed move. Next transit the machine to X10Y10 and add a feed move. Next transit the machine to X0Y10 and add a feed move. At this point, three of the four needed sides of the square have been added. We will now use the close shape button to continue with the example. Once the close shape button is pressed, the vertical head will lift, the machine will move to the starting point of the shape, and the vertical head will drop back to its previous position. Once the machine finishes moving, transit the machine to X15Y0 and then add a rapid move using the add rapid button in the toolbar. This will set the current position as the starting point of the next shape. This example would then output a file with two separate 10"x10" squares with a 5" separation between them.

<u>**Auto Digitizing**</u>
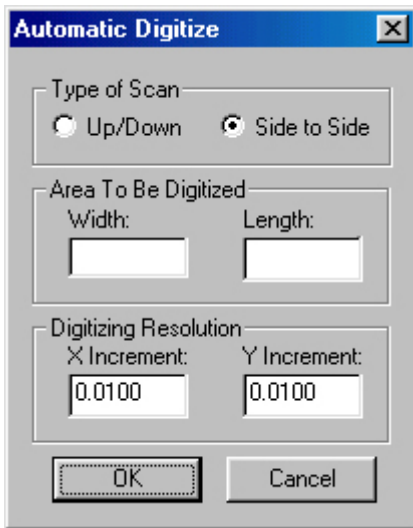
WinCNC supports 3D digitized scanning. The procedure for using a digitizer is to position the probe over the Z0 plane and perform an L82 command to measure the 0 plane. Once this has been done, position the probe over the beginning of the scan rectangle, you can perform a G92XY at this point if you want the corner to be (0,0). If the probe is enabled and it comes into contact with something while transiting, you must use the transit keys to move the probe up in order to release it. Now select the File->Digitize->Automatic menu item to show the Auto Digitize dialog box. Enter the area to scan, the desired scan increments, and the type of probe you are using. Then click the Start button. A successfully completed scan will create a file named SCAN.STL, a scan that is aborted in the middle of the scanning process will generate a partial file with all points after the abort set to Z0.

The specifics of setting up the WINCNC.INI file and each of the scanning commands are explained in detail in their respective sections of the manual. The INI settings needed are an auxin connected to the probe that is on when the probe is touched and a scan line. Specifications for the INI file settings are listed under the 3D Digitizing section of the System Configuration portion of the manual. The commands to reference for the digitizer are L82, L83, L800, L801, L802, and L803. These commands are explained in detail in the Command Reference.

Frame Scan Digitizing

Default Values	
XY Increment	0.5
Z Increment	0.1
XY Feedrate	30
Z Feedrate	30
Bottom Offset	0
Top Offset	0
XY tolerance	0.01
Scan Width	0
Scan Length	0
Scan Center	Unchecked
Center Scan Direction	Negative
Slice Scan Direction	Positive
Side to Scan	Top
Z Height to Calibrate	0
Distance to Raise Stuck Probe	1

Scan Increment - the distance the probe will attempt to move to reach new points. The actual amount can be less if the probe hits something during a move.

Scan Feedrate - the speed the machine will travel between points.

Bottom Offset - distance to shift the bottom scan data. This distance can be positive or negative.

Top Offset - distance to shift the top scan data. This distance can be positive or negative.

XY Tolerance - the allowable difference between points on the X or Y axis when finding an edge. Example: XY Tolerance is 0.01, the current X point is -0.4370 and the previous point was -0.4467. The difference is less than the allowable tolerance, so the slice will end.

Maximum Scan Dimensions - these dimensions should be set to the maximum distance that the user wants the probe to search for an edge. If the board will be clamped on the ends or sides, input in the distance between the clamps for that axis. For the other horizontal axis, set the size to the maximum amount that the probe should travel assuming that it doesn't find an edge. Always use positive values.

Center Scan Direction - the direction that the probe should travel on the Y axis. If set to negative, the probe will travel from high Machine Y to low Machine Y.

Slice Scan Direction - the direction that the probe will travel on the X axis. If set to negative, the probe will travel from high Machine X to low Machine X.

Slice Position Boxes - the input boxes beside the checkboxes in the slice scan group. This is the position on the Y axis where the slice will be measured. Always use positive values, even for negative scans.

Side to Scan - the side of the board to be scanned, Top or Bottom.

NOTE: A complete frame scan will consist of a Top and a Bottom scan. This means that the Frame Scan utility must be ran once for each side. The values that you set in the previous scan will be retained in their appropriate locations, so there is no need to retype them. The utility will compile all slices after each scan.

How to Use the Frame Scan Digitizing Feature

1. Start WinCNC. **Important!** Make sure to do a G28Z or an L28Z to set the machine Z zero to a level above the highest point of the workspace. This is necessary because the machine does a G53 (moves to machine Z zero) between slices. If you do a G53 and machine Z zero is below material height, the probe will hit the surface at rapid speed and be forced to stop very quickly, which could result in damage to the probe or the part.
2. Move the probe to the XY location where you want to start the scan.
3. Do a G92XY to set this beginning point to local zero.
4. If you want to do an L82, move the probe to the desired position.
5. Click File->Digitize->Frame to open the Frame Scan dialog box.
6. Click Browse and set the file name and path to use. The file will always save with a .CSV file extension.
7. Set up the Scan Increments (positive numbers).
8. If a Top or Bottom offset is to be used, set it. The default is zero.
9. Check the Scan Center box if a center scan is to be performed.
10. Choose the center scan direction.
11. Choose the slice scan direction.
12. Check the box next to a slice, then enter a Y position (positive). Repeat this for the number of slices desired.
13. After slices have been chosen and values are entered, choose the side to scan - Top or Bottom.
14. If an L82 is to be performed, enter the new Z value for the surface and press Calibrate. The probe will descend to the board and, when it touches, will calibrate the board surface to the height set in the box. This is very useful if you know the thickness of the board. Just calibrate to half of the thickness for each side and the final point cloud will center around Z zero. Example: The board is measured at 2.5" thickness. If the board is calibrated to 1.25" for each side at the point where it was measured, the finished scan should center around Z zero - half above zero, half below.
15. Set the height for the probe to raise if stuck. If the probe's Z zero is below its current position and it sticks, the probe will try to raise the distance specified here. If the Z zero is above the current position and the probe sticks, the probe will raise to zero. In either case the user is asked to "unstick" the probe and hit Enter - after which the scan will continue starting with the point where the probe stuck.

16. Click Run. **Important!** This will start the digitizing process and the machine will begin moving. If Options->AutoPreview is enabled the user will have to press ENTER again to start digitizing.

17. After the first side scan is completed, turn the part over to set up scanning for the other side.

18. If the starting point has changed, do a G92XY to reset the local zero.

19. If you are using the L82 setting for bringing the top and bottom together, move the probe the same X,Y point that was used on the other side of the part.

20. Click File->Digitize->Frame to open the Frame Scan dialog box.

21. The values for the previous scan will still be set. Leave all values set "as is" except the slice direction and the side to scan.

22. If the L82 is to be performed, make certain that the correct value is in the box and click Calibrate.

23. Click Run. **Important!** This will start the digitizing process and the machine will begin moving. If Options->AutoPreview is enabled, the user will have to press ENTER again to start digitizing.

24. After this side scan is completed, your CSV file will be complete.

NOTE: A large XY increment, combined with a large Z increment, can cause rough edges. Too large of an XY increment, combined with too small a Z increment might cause the scan to end before the true edge is found, especially if the XY tolerance is set too large.

System Configuration

WINCNC.INI is read on program startup. The preconfigured file is set up to work with minimal change for normal installations. To change configuration use a text editor to edit WINCNC.INI. If the WINCNC.INI is changed while WinCNC is running, you must restart the program to activate your changes.

CAUTION!!
Make sure you have some idea of what you are doing before changing WINCNC.INI. All input and output is controlled by these settings. Improper settings will cause limit switches to not work, wrong axis movement in the wrong directions, and/or other bad and potentially dangerous or damaging incorrect operation.

Please call for help instead of experimenting if you are not very familiar with motion control concepts and computer configuration.

Double-click the WINCNC.INI file from within Windows to open it in a text editor. Change values for parameters as specified below. WINCNC.INI is read every time the program is started. Make sure the file gets saved as ASCII text (without formatting).

WINCNC.INI Settings

Unit Settings
The default unit of measure is inches. If you want to use this WINCNC.INI setting to set the unit of measure, **it must be the first line in the INI file**. You can also set units by choosing Units from the menu bar and selecting the desired unit of measure.

unit=unit
Unit is the unit of measurement you want to use. **IN** - sets units to Inches **CM** - sets units to Centimeters **MM** - sets units to Millimeters Example: unit=IN would set up your configuration to use Inches as the unit of measurement.

Timer Card Setup

timertype=#
must be 7200

Axis Settings

Each axis for a machine must be configured in the WINCNC.INI file. Axis settings MUST be included in the following order:
Note: Values to the right of the equals signs (=) vary by installation.

[Axis Labels]
axischar=XYZ (others may be added here, such as U, B, A W etc)

[X Axis]
axisspec = p0 s0 d0 r818.5111 a400
axisvel = r450 f100 s50 m200 h450
axislo = p2 b6 o1
axishi = p2 b7 o1
[Other axis settings related to the X axis (axismode=, axisadj=, etc.) would follow]

[Y Axis]
axisspec = p0 s1 d1 r818.5111 a400
axisvel = r450 f100 s50 m200 h450
axislo = p3 b0 o0
axishi = p3 b1 o0

[Z Axis]
axisspec = p0 s2 d2 r400 a200
axisvel = r250 f50 s50 m100 h200
axislo = p3 b2 o0
axishi = p3 b3 o0

NOTE: Axes must be configured in the order they appear on the axischar= line. In the example above, the X axis would be set up first using the axisspec, Y next, Z next, etc.

axischar=### (# is desired alpha axis label.)
Example: axischar=XYZWUAB (The first axis MUST be X, and the second MUST be Y)

---

axisspec=P# S# D# R# A# F# O# E# T# B# K#

Configures basic functions for a motor on a machine.

SEE DIAGRAMS IN APPENDIX

P# is port number. (Should be 0 with PCI 7200 timer card)

S# is the bit for the step signal, can be 0-7.

D# is the bit for the direction signal, 0-7.

R# is the actual resolution in steps per unit.

A# is the actual acceleration in units per minute per second.

F# sets acceleration for feed moves. If no F# is used the A# parameter is used for both rapid (G0 and G53) and feed (G1, G2 & G3) moves.

O# changes motor direction, can be 0 or 1. (Default = 1)

E# sets the step signal for negative edge triggered drives, can be 0 or 1. (Default = 0) (For Pos edge triggered drives, use E1)

T# sets the type of axis. 1 is first horizontal, 2 is second horizontal, 3 is vertical, 4 is rotational.

B# is the amount of backlash to compensate for on the axis.

K# is used for keyboard mapping to this axis. (If no K value is present, default is K1 for X axis, K2 for Y, K3 for Z, K4 for W)

Ignores Scroll Lock (Normal Default)

    1   Left/Right Arrow Keys (K1)

    2   Up/Down Arrow Keys (K2)

    3   PageUp/PageDown Keys (K3)

    4   Home/End Keys (K4)

Works Only with Scroll Lock Off

    5   Left/Right Arrow Keys

    6   Up/Down Arrow Keys

    7   PageUp/PageDown Keys

    8   Home/End Keys

Works Only with Scroll Lock On

    9   Left/Right Arrow Keys

   10   Up/Down Arrow Keys

   11   PageUp/PageDown Keys

   12   Home/End Keys

All of the following axis configuration parameters will apply to the preceding "axisspec=" line.

---

axisvel=R# F# S# M# H# A# C#

Sets up velocities for the specified axis.
R# sets the rapid (G0) velocity for the axis.
F# sets the feed (G1) velocity for the axis.
S# sets the velocity for slow keyboard transit.
M# sets the velocity for medium keyboard transit.
H# sets the velocity for fast keyboard transit.
A# specifies an acceleration factor for keyboard transit moves
(this value should normally not be more than 1)
C# sets minimum velocity used in velocity matching. (Using this
setting improves smoothing and reduces cutting times. The veloc-
ity set here will make it so that even while slowing for a change of
direction, the axis will not go any slower than this speed (C#). This
is often used in conjunction with the G09 setting.)

---

axisadj=T# A# V# U# D# I#

---

*Can be used as a height control while the file is running. Adjusts
the position of an axis by: transit keys and input switches. This
should not be used when using an automatic height control (see
THC=).

T# set to 1 to enable transit key control of the specified axis.
A# sets the adjust enabled axis. 0=X, 1=Y, 2=Z, etc.
V# sets the maximum velocity of the specified axis during adjust-
ment, in units per minute.
U# sets the input channel number used to trigger a positive adjust-
ment.
D# sets the input channel number used to trigger a negative ad-
justment.
I# is the initial state of axisadj (1=enabled, 0=disabled).

---

axismap=S# E# M#

Creates a position map for the axis to straighten the path of motion
for an axis. Add this line under the "axisspec=" line for each axis to
be mapped.
S# sets the starting position for the map.
E# sets the ending position for the map.
M# sets the map increment used for this axis.

---

altaxisspec=P# S# D# O# E#

See axisspec= documentation. This setting is used for an axis that has two motors, and specifies the step and direction settings for the second motor. Uses the same resolution, acceleration, type, and backlash as axis specified in the previous axisspec= line.

---

axishi=P# B# O# A# D#
axislo=P# B# O# A# D#
altaxishi=P# B# O# A# D#
altaxislo=P# B# O# A# D#

High and low limits for axis and altaxis.
P# is the input port for limit switch, can be 0-3.
B# is the bit on the port used with the switch, can be 0-7.
O# sets the switch polarity, can be 0 or 1.
A# sets the alternate bit.
D# is the debounce for the limit switch.

Example: axislo=P0 B2 A4 [bit 2, alternate bit 4]

---

exaltaxislo=C# P# A# B# O#
exaltaxishi=C# P# A# B# O#

Used to configure up to 10 alternate input pins that can be used as alternate limit switches.
C# is the alternate limit channel number. There are up to 10 low and 10 high channels available
P# is the switch port number.
A# is the alternate bit.
B# is the switch bit number.
O# sets the polarity for the switch (0 or 1).

---

accel=S#

S# sets the max velocity increment during acceleration. Can be 0 to 200. The default value is 50.
This setting pertains to the s curve setting of acceleration and deceleration. Normally, using the default value of 50 is recommended.

---

ad=L# H# X# Y# A# B#

Reads incoming analog signal used to monitor spindle current.
L# is the low volts for the voltage range.
H# is the high volts for the voltage range.
X# is the low amperage for the voltage range.
Y# is the high amperage for the voltage range.
A# is the low alarm in amps.
B# is the high alarm in amps.

---

anykey=A# P#

By default, the Spacebar is the Pause key and the Escape key is the abort key. By including the anykey= line in the WINCNC.INI file, the user can set WinCNC to read any unassigned key as a pause key or an abort key.
Example: if "anykey=a1" is in the WINCNC.ini file, then when any unassigned key on the keyboard is pressed, the file will abort all operation. If "anykey=p1" is in the WINCNC.ini file, pressing any key on the keyboard will pause all operation.

A# enables or disables abort on any unassigned key. 0 is disabled, 1 is enabled.
P# enables or disables pause on any unassigned key. 0 is disabled, 1 is enabled.

If both a1 and p1 are included, any unmapped key will be read as an abort, overwritting the pause command.

---

arc_err=#

is the value (in units of measure) of allowable errors in arc specifications. Default value is .01. If you get radius errors from your files increase in .002 steps. If errors persist with larger values the problem is probably in your program. Arc radius errors indicate that the distance from the start point to the center point is not the same as the distance from the end point to the center point.

---

arc_min=#

is the value (in units of measure) for smallest arc radius to be cut as an arc. Arcs with radius smaller than this value will be cut as a G1 move. Default value is .002.

---

arcfeed=R# F#

R# specifies the radius
F# specifies the feed rate in units per minute.

Arcfeed lines are not necessary or recommended in setup. Arc feed rates are calculated automatically based on your acceleration settings. If you wish to override the automatic calculations, use a block of "arcfeed=" lines to manually specify radius and feed rates. Three(3) to twenty(20) "arcfeed=" entries can be made in the WINCNC.INI file. When doing a G2 or G3 move, WinCNC will use the feed rate associated with the radius of the arc move. When the arc radius falls between two "arcfeed" radius values, WinCNC will calculate the appropriate feed rate between the two "arcfeed" values. When the arc radius is smaller than the smallest "arcfeed" radius, then the "arcfeed" feed value from that entry will be used. When the arc radius is greater than the largest "arcfeed" radius value, then WinCNC will use the default auto arc feed mode. When no "arcfeed" entries are made in the WINCNC.INI file, WinCNC uses the default auto arc feed mode, where arc feed rates are automatically calculated based on the programmed G1 feed velocity.

---

arctype=T# I# M#

Allows configuring WinCNC to break up arcs into line segments.
T# - 1 for normal arc and 2 for interpolated arcs.
I# - the length of the segment to interpolate.
M# - if set to 1 makes G2/G3 commands modal.

---

atc1=H# O# Z# A# M#
atc2=H# O# Z# A# M#

Configures Automatic Tool Changers.
H# is the height at which the tool is unloaded.
O# is the offset to the tool sensor.
Z# is the height of the bottom of the tool changing spindle.
A# is the axis number for the tool changing spindle. X=0, Y=1, etc.
M# can be set to 0 or 1. Mode 0 stores tool measures in an internal settings file, not accessable from the tool library. Mode 1 specifies the tool measures are stored in the tool library.
(see M37 and L110 documentation)

---

autoaux=C# I# S# E#

C# is the auxiliary output channel number.
I# is the initial state of the autoaux mode (0=off, 1=on).
S# sets the amount of dwell time in seconds used when the autoauxout channel is set to on.
E# sets the amount of dwell time in seconds used when the autoauxout channel is set to off.

This feature allows the user to setup an auxiliary output that will turn on during G1/G2/G3 moves, and will turn off for all other moves, only if the "autoauxout" mode is set to on. (Meaning, the inital state must = 1). The "autoauxout" mode can be enabled using the L34 command and disabled using the L35 command.

**NOTE: When using height control (such as the THC), you cannot use the autoaux=.

---

autodir=c:\wincnc\auto

Specifies the directory where files to be automatically ran are copied.

---

autofdir=c:\wincnc\done

Specifies the directory where files are copied after auto run.

---

automode=#

The autorun mode 0 is disabled, mode 1 will copy name and path to command bar, mode 2 will automatically execute the found file.

auxin=C# P# B# O# D# F# L# I# M"<message>" A"<axes>"

Configures auxiliary inputs.
C# is the channel number, 1-32. Numbers should not be skipped.
P# is the port number, refers to a port specified in a portin spec.
B# is the bit number, can be 0-7.
O# changes the input polarity, can be 0 or 1.
D# specifies debounce. When the input is a switch, debounce should be used. D10-50 is a typical value.
F# specifies the function type of the auxin. Can be set to the same function types as **enab**, or **signal** below. When set to 0, no special function types are enabled. (If the same F command is used for different channels, there will be a conflict.)
   F1 - Enable
   F100 - Signal Abort
   F101 - Signal Pause/Continue
   F102 - Signal Pause
   F103 - Signal Continue
   F110 - Signal Start Cycle
   F201 - Signal User Spec 1, use L201 command
   F202 - Signal User Spec 2, use L202 command
L# can be 201 or 202. If configured as an enable, this causes the input channel to be used as an enable when the machine is running, and as a type 201 or 202 signal when not running.
I# is the initial active/inactive state of an enable or signal. 0=active, 1=inactive.
M"<message>" denotes a custom error message. The message must be enclosed in double quotes - "".
A"<axes>" requires homing of the specified axes if this auxiliary input is used as an enable (F1) and the enable is triggered during motion.

---

auxout=C# P# B# O# S# E# T# X# W# R#

Configures auxiliary outputs.
C# is the channel number, 1-32. Numbers should not be skipped.
P# is the port number, refers to a port specified in a portout spec.
B# is the bit number, can be 0-7.
O# changes the output polarity, can be 0 or 1.
S# set output state on WinCNC startup, can be 0 or 1.
E# set output state on WinCNC exit, can be 0 or 1.
X# set output state on abort, can be 0 or 1.
W# set output state on pause, can be 0 or 1.
T# sets the type of auxout. # can be 2 or 3. T2 sets the auxout channel to be a motion only auxout. This will result in the auxout only working while the machine is moving and will turn off when the machine stops. To use this feature you must also have the S# set to 1 to start up with the software. You can then turn it off and on using the M11 and M12 commands. You must have it enabled to work correctly during motion. The T2 auxouts are available on CN1 and CN2. Auxouts on additional I/O cards cannot use this setting. T3 sets the auxout to a programmed velocity percentage auxout type. T3 auxouts are on when the machine velocity is above the specified percentage (based on the auxoutvel=, see below) and off when the velocity is below the specified percentage.
R# is used to specify whether the output should be turned on or off during a file restart. 0=off, 1=on. When a R value is used, the auxout state will be set by simulating the skipped portion of the file prior to restarting.

---

auxoutvel=#

is the velocity percentage to use (0-100). If the machine velocity, during motion, drops below the specified percentage of the programmed velocity then the auxout signal will be on, otherwise, the auxout signal will be off. (Used with the T3 setting of auxout=)

---

backlash=B# S# F#

B# sets initial backlash compensation to on or off, can be 0 or 1.
S# sets the smoothing factor for backlash compensation moves.
F# sets the "slow down" feed rate setting to eliminate "clicking" during arc backlash compensation moves.
Example: backlash=b1 f100 means enable backlash, slowdown feedrate of 100 units per minute.

bitmapdir=c:\wincnc\bitmaps

Specifies the directory where bitmaps used with the splash screen, background, and screen buttons are kept.

---

bmpeditor=<filename>

Filename is the full path and filename of any editor that will accept file names as a command line parameter.

Example: bmpeditor=paint.exe

---

bmpend=<command>
bmpstart=<command>

Commands to be automatically added at the beginning and end of each bitmap file.
Example: bmpstart=m11c1 AND bmpend=m12c1
or bmpstart=m98 start.tap AND bmpend=m98 end.tap

---

bmpoffset=L# H# S# F#

When using this method an offset between the low and high value is selected based on the speed of the raster axis. If a speed below the low velocity is used, then the low offset is used. If a speed above the high velocity is used, then the high offset is used.
L# is the low offset
H# is the high offset
S# is the low velocity
F# is the high velocity

---

btnoptions=#

This setting enables/disables the Options->Buttons menu option.
# can be one of the following:
   0 = disabled
   1 = enabled (default)

---

cmdabort=<command>

<command> is the command that will run when a file is aborted.
Example: cmdabort=M12C1

---

cmdend=<command>

<command> is the command that will run at the end of every file.
Example: cmdend=M12C1

---

cmdrestart=<command>

When specified, the cmdrestart line will run at the end of a file restart, before the remainder of the file is run.

---

cmdrestartfeed=<command>
cmdrestartrapid=<command>

Allows the insertion of commands prior to motion when using the file restart. These commands are only inserted if the first move at the restart point is a feed or rapid move. This allows restart setup to ensure that all device states are correct for cutting or noncutting moves.

---

cmdstart=<command>

<command> is the command that will run at the start of every file.
Example: cmdstart=M11C1

*NOTE: These settings can also be used to call a subroutine to use multiple commands.

---

comchars=###

represents characters to use to designate comments in a G-Code file. When one of the listed characters is found at the beginning of a line, WinCNC skips the entire line. The characters "[(%" are regarded as comments by WinCNC by default.

Warning: Any character listed here will be considered as a comment character, therefore everything that follows that character on any line will be ignored.

---

comdll=<path>

<path> is the path including filename of the communications DLL file "WINCNC.DLL".

---

contdelay=#

Setting this value causes a # second delay when continuing from a paused state.

---

cutcomp=T# D#

Sets up cutter compensation to account for bit sharpening and wear.
T# is the type (0=does not insert entry move - default, 1=automatically insert entry move).
D# a negative number here will invert G41 and G42 commands left and right compensation.

---

da=T# A# V# I# C#

T# is the type of D/A port used. Can be 0 (parallel port) or 1 (PCI daughter board) or 2 (serial port).
C# is the COM port number and is only used when T is set 2.
A# is the parallel port address and is only used when T is set to 0.
V# is the maximum voltage - generally 5 or 10 volts.
I# is the initial output voltage when WinCNC starts. It can be set from 0 to the maximum set in V**#**.

---

drillbank=C# N#

Configures a multiple drill gang associating auxouts with the associated drill.
C# is the auxout channel, first one that lowers a drill.
 NOTE: AuxOut channels need to be in numerical order.
N# is the number of channels.
 See Also the M11 command.

---

editor=<filename>

Filename is the full path and filename of any editor that will accept file names as a command line parameter.

Example: editor=write.exe

---

enab=C# M"message" T# L# I# A"<axes>"

C# specifies the input channel to be used for the enable.
M"message" denotes a custom error message. The message must be enclosed in double quotes - "".
T# auxin enable type. 1 is default and will abort from any command when the enable switch is triggered. 2 will abort only from commands that would result in machine motion when the enable switch is triggered.
L# can be 201 or 202. If configured as an enable, this causes the input channel to be used as an enable when the machine is running, and as a type 201 or 202 signal when the machine is not running.
I# is the initial state (1=active, 0=inactive).
A"<axes>" requires homing of the specified axes if this auxiliary input is used as an enable (F1) and the enable is triggered during motion.

Used to setup emergency stops. First setup an auxin line for input. Then specify the enable to use the input channel as an enable switch.

---

esc_step=#

is the number of steps to move after ESC key is pushed.

---

filetype=*.tap;*.nc

Specifies file extensions that are recognized by WinCNC as G-Code files.

G09=S# A#

Used to set the smoothing factor. When the XY direction of motion changes, this setting can reduce the "slowdown" to improve the smoothness of motion. Increasing the S# decreases the slowdown of the machine when turning corners.

(continued on next page)

G09=S# A# (Continued)

S# - smoothing factor to be used. (not a specific unit of measure)
A# - used to change the angle used in the velocity matching algorithm. The A# value represents an angle. If a small angle, such as say 20, is specified, the G09=# will not apply to any angle smaller than 20.
 **A minimum velocity can be set for each axis by adding a C# to the Axisvel= line for that axis.

---

g2modal=#

can be 0 or 1, if 0 G2 and G3 commands are non-modal, if 1 G2 and G3 commands are modal. The default is non-modal.

---

g28altmove=X# Y# Z# W#

Used to square a machine with two motors on the gantry. The character to be used will correspond with the axis having the second gantry. The # represents the physical adjustment needed to square the gantry.

---

g28check=#

When enabled, if new absolute home positions are set using G28 or L28, they are checked against the previous positions and the user is prompted to accept or decline the new positions. (0=disabled/default, 1=enabled)

---

g28checktol=X# Y# Z#

sets the tolerance, changes of less than the tolerance are ignored.

---

g28move=X# Y# Z# W# R# F# T# A# M# L#

XYZW# specifies the distance to move from each limit switch while homing.
R# the distance to move off the limit switch after the first contact.
F# is the approach speed to use for the initial contact.
T# is the approach speed to use for second contact. (Default 10)
A# is used to specify that an alternate limit switch is to be used.
M# specifies that each axis will home individually.
L# maximum number of units to move.

---

g28coord=X# Y# Z# W#

Specifies a position after a G28, not used if all positions are zero.

---

g37=X# Y# I# J# D# F#

Configures tool measure switches.
X# is the X position of the tool measure switch.
Y# is the Y position of the tool measure switch.
I# is the X position of the second tool measure switch.
J# is the Y position of the second tool measure switch.
D# is the rapid descent distance.
F# is the feed rate for the remaining descent distance after rapid rate.

---

g54=X# Y# T# R# O# M#
g55=X# Y# T# R# O# M#
g56=X# Y# T# R# O# M#
g57=X# Y# T# R# O# M#

Preconfigured workspace values (default is G54).
X# is the workspace shift value for the X axis.
Y# is the workspace shift value for the Y axis.
T# is the type of workspace shift. Use 0 for vertical head swapping, use 3 for rotating spindle, and use 5 for spindle w/two points of rotation.
R# is the radius of rotation used for type 3.
O# is the offset value, which is used for correcting the 0 degrees position. The 0 degrees position should point toward the positive X direction.
M# is used to determine if axes will move to new positions in the new coordinate system when switching between modes. (1=move to new positions, 0= do not move).

---

g81set=D# R# L#

Configures parameters used in drill cycles.
D# is the dwell time for the drill in seconds.
R# sets the rapid drill for the peck cycle to within R# distance of the hole before switching to feed mode.
L# is the lift distance above the top of the hole for the bit between each peck.

---

g81dwell=#

is the value (in seconds) of the desired dwell at the end of the plunge move. The default value is 0.

---

helpfile=<path>

<path> - directory for the manufacturers help file.

---

hibound=X# Y# Z# W#

Boundaries for the X, Y, Z, W axes that take into account work-space offsets.

---

hilim=X# Y# Z# W#

Absolute position soft limits for the X, Y, Z, W axes that do NOT take into account workspace offsets.

---

homedir=c:\wincnc\homes

Specifies the directory where created home files are written.

---

keypad=C# B# P# S#

Configures keypads for use with WinCNC.
C# is the number of the com port used with the keypad.
B# is the baud rate used with the specified port. Should be 9600.
P# is the type of keypad used. Type 1 is the standard serial key-pad. Type 2 is the Q-Term keypad with LCD display. Type 3 is the 2Tech keypad with LCD display and optional emergency stop.
S# is the sorting type and can be one of the following values:
    1 = sort alphabetically, ascending
    2 = sort by file size, ascending
    3 = sort by modification date, ascending
    4 = sort by creation date, ascending
   -1 = sort alphabetically, descending
   -2 = sort by file size, descending
   -3 = sort by modification date, descending
   -4 = sort by creation date, descending

---

keypaddir=c:\wincnc\keypad

Specifies the directory where files that can be selected from a sup-ported LCD keypad are stored.

knifemode=T# A# L# Z# N# C# D# (Tangential knife)

T# type of knife. T0 = normal knife function. T1= target mode. (Beginning of XY move, a target position is set internally. During the XY move, the knife axis will move into position as quickly as possible based on step velocity of XY. This allows the rotating knife to be used without affecting speed and quality of XY axis motion.) **A#** is the lift angle. If the angle of change is greater than this value then the head will lift before rotating.
L# is the XYdistance where the knife rotation is blended at the beginning and ending of each line segment. This setting can be used to smooth corners.
Z# the lift position used to raise the knife above the material.
P# is the lower position used to lower knife into material for cutting.
N# is the rotational axis number, 0 is X, 1 is Y, etc.
C# specifies auxout channel used to raise and lower knife. If not used, then Z axis is used to raise and lower knife.
D# is the dwell time (in seconds). This is used with C#. Denotes the specific amount of time needed to raise or lower the knife.

laser=P# D# S# M# O# A# B# R#

P# is the initial power scale.
D# is the laser power duty cycle.
S# sets smart engraving off/on (0/1).
M# is the margin for acceleration on either side.
O# is the step offset used to align the bitmap rows.
A# sets the primary (raster) axis number. 0=X, 1=Y, etc.
B# sets the secondary (row) axis number. 0=X, 1=Y, etc.
R# sets the default radius of the engraving surface when a rotary axis is used.

laserscan=S# M# U# W# A# T#   (for scanning 3-D objects)

S# sets the step pin/bit used to trigger laser scanning (0-5).
M# sets the margin in units. The margin is used at the start and end of each pass to allow room for machine acceleration. A value of 0 enables the auto-margin feature.
U# set to 1 or greater to increase the auto-calculate margin size.
W# sets the number of "warm-up" readings used at the beginning and end of each pass to allow the Optimet system to collect more accurate data. Recommended 3-5.
A# is used to specify the axis 0=X axis, 1=Y axis.
T# is the type. 1 is ethernet, 0 is LPT.

lim_cnt=#

sets the debounce for limit switches. (MAXIMUM is 30. If set higher, it will default back to 30)

---

lim_mode=#

0 = same debounce for on/off, any limit switch aborts.
1 = same debounce for on/off, only limit switch in the direction of motion causes an abort.
2 = less debounce for change to off, any limit switch aborts.
3 = less debounce for change to off, only limit switch in the direction of motion causes an abort.

---

lim_step=#

# is the number of steps to compress limits after switch hit. Some machines may be damaged if stopped suddenly at higher speeds. Abort cushions provide a method of decelerating quickly but not instantly when a limit is hit or the escape button is pushed. lim_step specifies the number of steps to continue after a limit is hit. This number must be calculated from the distance your switches may be compressed without damage and the lowest axis resolution of your machine. For instance if your X and Y resolution are 1018.592, your Z and W resolutions are 2000 and your limit switches may be compressed .1" then you would set lim_step as follows:
lim_step=1018.592*.1=100 steps
esc_step may be set to any value that results in a quick stop without moving too far. A 200 setting is suggested.

---

lobound=X# Y# Z# W#

Boundaries for the X, Y, Z, W axes that take into account workspace offsets. Recommend using Z0 & W0 only so that the vertical head will not go below the tool measure of Z0.

---

logdir=c:\wincnc\log

Specifies the directory where the log file is written.

---

logfile=c:\wcnc.csv

Specifies the name of the log file.

---

lolim=X# Y# Z# W#

absolute position soft limits for the X, Y, Z, W axes that do NOT take into account workspace offsets.

---

mapping=I#

I# is the interpolation increment used for all axes.

---

maxstepv=#

is the maximum number of pulses per second per axis. The default value is 30000, and can safely be increased to 50000 using the 7200 timer type.

---

maxtrans=#

is the distance, in units, of the largest possible transit move. Setting this value to a distance equal to or slightly larger than the largest table dimension on a machine will make the transit moves respond more quickly.

---

override=E# L# H#

Gives the user the options of enabling or disabling the feed rate override, and setting the low and high limitations for the feed rate.

E# enables or disables the feed rate override. Can be set to 0 or 1. 0 is disabled, 1 is enabled. The default is enabled.
L# sets the lower limitation of the override percentage. Can be 1 to 200, but must be lower than the high value.
H# sets the high limitation for the override percentage. Can be 1 to 200, but must be higher than the low value.

---

portin=#, *
portout=#, *

is the number of the port, can be 0-7.
* is the hexadecimal memory address to the port.

PCI48H=A# B#
PCI24H=A#

A# - Sets IO for ports A1, B1, and C1. (PCI48H and PCI24H)
B# - Sets IO for ports A2, B2, and C2. (PCI48H only)
WinCNC IO port numbers are automatically assigned to ports 8-12.
A# or B# can be set to 0-15 from the following table.

  0 - A=out/B=out/CU=out/CL=out
  1 - A=out/B=out/CU=out/CL=in
  2 - A=out/B=in/CU=out/CL=out
  3 - A=out/B=in/CU=out/CL=in
  4 - A=out/B=out/CU=in/CL=out
  5 - A=out/B=out/CU=in/CL=in
  6 - A=out/B=in/CU=in/CL=out
  7 - A=out/B=in/CU=in/CL=in
  8 - A=in/B=out/CU=out/CL=out
  9 - A=in/B=out/CU=out/CL=in
 10 - A=in/B=in/CU=out/CL=out
 11 - A=in/B=in/CU=out/CL=in
 12 - A=in/B=out/CU=in/CL=out
 13 - A=in/B=out/CU=in/CL=in
 14 - A=in/B=in/CU=in/CL=out
 15 - A=in/B=in/CU=in/CL=in

---

positions=P# H# T#

P# - maximum number of stored positions.
H# - maximum number of stored home positions.
T# - maximum number of stored tool positions.

---

pumpcont=T# L# H# A# B# I# R#   (Configures pump control.)

T# sets the pump control type. The available types are:
    T0 pulse width 200hz
    T1 pulse width with auto reverse 200hz
    T2 pulse width without auto reverse, double scaled 2hz
    T3 frequency mode
L# sets the low value of the output range in units per minute.
H# sets the high value of the output range in units per minute.
A# sets the low value of the frequency range in pulses per second.
B# sets the high value of the frequency range in pulses per sec.
I# is the initial value of the pump control output (0-99).
R# is the initial value of the pump reverse output (0-15).

---

quadmode=1

Sets WinCNC to generate an A/B quadrature output instead of the traditional step/direction signals used by most motor drives. This applies to all step and direction pins.

rapid_lock=#

can be: 0 - Unlocked status and can change G00 feedrates, or 1 - Locked status and can not change G00 feedrates.
The default setting for this feature if not specified in the INI file is 1, the locked status.

rapidjog=#

can be 0 or 1. If 0, jog moves us G1 speed (this is default). If set to 1, jog moves us G0 speed.

restart=V#

V0 skips vertical axis position restore
V1 restores vertical axis positions (default)

sawmode=T# A# R# O# Z#

T# - type of saw. This setting must be 1. It is reserved for future use.
A# - is the rotational axis number, 0 is X, 1 is Y, etc.
R# - is the radius of the saw blade.
O# - is the offset, in degrees, from the home position to the 0 degree cutting direction.
Z# - is the lift amount used when the auto-lift is used for saw blade rotation.

scan=T# A# C# S# H# F# P# D#   (T, A & C are required)

Configures touch-probe scanning to digitize 3D objects.

T# is the probe type. 1 is an up/down probe. 2 is an up/down/side to side. This setting is **required**.

A# is the axis # the probe is on. 2 for Z. Optional setting. Defaults to 2.

C# is the auxin channel the probe is connected to. This setting is **required.** An Auxin line must be added to the WinCNC.ini file. This channel # cannot be the same as any other used channel.

S# is the # steps to decelerate the probe. Optional setting that defaults to 50 if not specified.

H# is the height to lift when the probe comes into contact with something. Optional setting that defaults to .005 if not specified

F# is the feed rate to scan with. Optional setting that defaults to 30 if not specified.

P# is the plunge for the A#. Optional setting that defaults to 30 if not specified.

D# is the debounce or number of times that the input status is read to determine a state change. Default is 0.

Example: scan=T2 A2 C8 S50 H.005 F30 P30

---

scanfile=T# F# O# A#

T# is the threshold that is to be used. The accuracy that is used for the STL file. The higher the number the smaller the STL file will be, the smaller the number the larger the STL file will be. Using 0 for this value will result in points. Defaults to .003.

F# sets the file type. 0 is ASCII, 1 is binary. 0 is default.

O# sets the output file format. 0 is STL, 1 is DXF. 0 is default.

A# sets the algorithm used to generate the output file. 0 uses the original algorithm, which includes all scan data. 1 uses the new algorithm, which eliminates unneeded data points to reduce the size of the output file. 0 is the default. This setting is used only with STL files.

signal=C# M"message" T# I#

Signal is used to setup control push buttons. First setup an auxin line for the input. Then specify signal as follows to use the input channel as a control signal.

C# specifies the input channel to be used for the signal.
M"message" denotes a custom error message. The message must be enclosed in double quotes - "".
I# is the initial state (1=active, 0=inactive).
T# specifies the signal type.
   *100 Abort
   *101 Pause/Continue
   *102 Pause
   *103 Continue
   *104 This signal will prevent all commands or attempts to run a file but will allow jog/transit moves if the signal is in an "on" state. If the signal turns "on" during a move or while running a file, WinCNC will pause, and will not allow a continue until the signal has turned back to an "off" state.
   *110 Start Cycle
   *201 Run command stored with L201 command
   *202 Run command stored with L202 command

---

skipchars=C"#" N#

A character or group of characters that indicate a block of G-Code that should be skipped or ignored.
C"#" - # represents the character or characters that mark the beginning of the code block to be skipped. Characters must be placed within quotes.
N# - # represents the number of lines to skip.

Example: skipchars=C"#%" N10
Sets WinCNC to skip 10 lines of G-Code each time that it reads a '#' or a '%' character at the beginning of a line in a G-Code file.

softlim=T# M# B#

Configures software limits for motion control.
 T# can be one of the following:
     0 = disabled
     1 = enabled and active
     2 = enabled but inactive
 M# is to tell whether or not to show the softlim's toolbar icon. 1 will result in the toolbar icon being shown, 0 for not shown. Boundaries and Soft Limits are linked, so the state of one represents the state of the other.
 B# can be one of the following:
     0 = exceeding boundaries will cause an abort
     1 = exceeding boundaries will adjust the target position to the boundary location.

spinchar=#
# - character used for spindle speed control.

spindle=T# R# I#
Support for D/A card for spindle speed control.   **T#** is the type of spindle speed control. 1=D/A (CN2/Pins 12-14), 2=Auxout (Spin.mac), 3= Frequency (CN2/Pin 11).   **R#** is the maximum revolutions per minute for the spindle.   **I#** specifies the initial output voltage when WinCNC starts.   Once this line is in the INI file the spindle speed can be set by using S# where # is the desired rpm of the spindle.

---

steppulse=P# D#

P# is the length of each step pulse in microseconds.
D# is the amount of time before each step that the direction signal is set.
NOTE: These settings default to 5, and are generally unused.

---

table=X# Y# W# H# B# Z#

Configures table dimensions for the G-Code/BMP Viewer.
X# is the low X coordinate to start viewer area.
Y# is the low Y coordinate to start viewer area.
W# is the width of the table area to view in viewer.
H# is the height of the table area to view in viewer.
B# is the border size in units (keeps table outline viewable).
Z# is the auto-zoom for the G-Code Viewer. 1 (default) - auto zoom out only. 2 - auto zoom in and out. 3 - no auto zoom.

---

thc=A# F# V# S# E# P# O# I# D# L# T#   (Torch Height Control)

A# is the axis number (x=0, y=1, z=2, etc.) to adjust for THC.
F# is the feed rate or velocity, used to adjust the specified axis up/down for THC. (Default = 100)
V# is the target voltage for the THC, must be between 20 and 250.
S# is the start, or pierce delay, in seconds, used for THC when turning the torch on.
E# is the end delay, in seconds, used for THC when turning the torch off.
P# is the percentage (1-100) of the programmed velocity that when reached will activate the THC.
O# is the auxiliary output channel number used to turn the torch on and off for THC.
I# is the input channel used to detect the "arc good" signal for THC.
D# specifies the compression distance of the switch used for the material touch-off.
L# sets the distance to raise the torch when it is turned off prior to moving to the next pierce location.
T# is the tolerance. Voltage changes of less than T# are ignored.

---

thcdisab=V# T#

V# is the amount of instant voltage change that will trigger a disable.
T# is how long in seconds to disable.

---

thcvel=F# V#

F# is the velocity
V# is the voltage

At F# velocity ignore voltage changes less than V#. Up to three thcvel= lines can be used for different velocity settings.

---

vgroup=<F>###

F is only used for the feedrate command group. It normally locks the X and Y axes together. It isn't used on any other vgroup line. #s are the desired alpha axis labels to group velocity.

Examples:
vgroup=FXY is the special feedrate command group; If the command F300 is given on the command line, it forces the X and Y axes to use 300 as their feedrate.

vgroup=XY would lock the X and Y axes, so that if the command G1 X15 F50 is entered, both axes would be set to 50. Otherwise, Y would remain at whatever feedrate it had previously been set to. Other than the X and Y axes, each axis should have its own vgroup; i.e. vgroup=Z.

---

watch=T# D# C#

The "watch" feature in WinCNC generates a square wave using an Auxout channel, timer chip out 0, or timer chip out 1.

T# is type (1-7).
D# is delay.
C# is auxout channel number.
T# can be 1 for pin 11, 2 for pin 15, or 3 for both.
The feature can only be enabled if the "pumpcont" feature is not used. Output pins 11 and 15 will go high when the PC is powered up. When the "watch" feature is enabled, one or both of these outputs will go low while WinCNC is running and high again when WinCNC exits.

Examples:
watch=T1 Pin 11 on CN2 (Startup=Low, Exit=High)
watch=T2 Pin 15 on CN2 (Startup=Low, Exit=High)
watch=T3 Pin 11 and pin 15 on CN2 (Startup=Low, Exit=High)
watch=T4 D50 - Pin 11 on CN2 (50ms Square Wave)
watch=T5 D50 - Pin 15 on CN2 (50ms Square Wave)
watch=T6 D20 - Pin 11 and pin 15 on CN2 (20ms Square Wave)
watch =T7 C# D20 - Auxout Channel (C#) (20ms Square Wave)
watch=T8 Not Connected
watch=T9 Pin 15
watch=T10 Pin 11

CNCSCRN.INI Settings

Program Icon
"Icon", "c:\wincnc\icons\msicons.ico"

Display Sections
"Display", "Viewer", 370, 145, 265, 180
"Display", "ButtonSet", 0, 200, 680, 520
"Display", "Command", 0, 0, 425, 195
"Display", "Position", 430, 0, 235, 150
"Display", "StoredPosition", 5, 330
"Display", "Main Window", 0, 0, 680, 435
"Display", "Main Color", 180, 180, 180 (R, G, B)
"Display", "Title", "MicroSystems WinCNC Controller"
"Display", "FileStatWindow", XStart, Ystart
"Display", "JobWindow", 5, 350
"Display", "THCWindow", 822,505
"Display", "Position", 527, 0, 260, 140, 1   (the 1 displays new absolute
(ABS) and local (LCL) display - 0 will display standard G92)
"Display", "Position", 527, 0, 260, 140, 0, 2   (the 2 denotes the number of
decimal places in the position window. This can be changed to 2, 3 or 4.
Any other value will default back to 3)

Viewer Sections
All sections except the Background and PosIndicator sections have
the same syntax.

Syntax	"Viewer", "<TYPE>", Red (0-255), Green (0-255), 0=Solid/1=Dash, Width (Pixels)
Defaults	"Viewer", "Table", 255, 0, 0, 0, 2 "Viewer", "Feedline", 0, 0, 255, 0, 1 "Viewer", "RapidLine", 0, 0, 0, 1, 1 "Viewer", "PreFeedLine", 180, 180, 180, 0, 1 "Viewer", "PreRapidLine", 180, 180, 180, 1, 1 "Viewer", "Grid", 210, 210, 210, 0, 1 "Viewer", "Local", 0, 0, 0, 0, 1

Syntax	"Viewer", "Background", Red (0-255), Green (0-255), Blue (0-255)
Default	"Viewer", "Background", 255, 255, 255

Syntax	"Viewer", "PosIndicator", Red (0-255), Green (0-255), Blue (0-255), 1 [Indicator Enabled]
Default	"Viewer", "PosIndicator", 255, 0, 0, 1

WinCNC Controller

Slider Controls

Syntax: "SlideControl", Left, Top, Width, Height, Output, Unused, Unused, Unused, Unused, ScanOn, ScanOff

The Output value can be 0 for Spindle Speed, 1 for D/A, 2 for Laser Power, 3 for Maximum Pump Output percentage, and 4 for Pump Reverse Time percentage. If using Spindle Speed alone, or Laser Power alone, the +- (plus and minus) keys are used to adjust values up or down. If using Spindle Speed and Laser Power together, Laser Power uses the +- (plus and minus) keys and the Spindle Speed uses the *Shift* +- (Shift, plus and minus) key combinations.

"SlideControl", 425, 5, 235, 50, 0, 0, "", "", "", -1, -1

Jog Mode Buttons

Syntax: "JogMode", Left, Top, Width, Height, Mode, Default Speed, Label, Unused, Unused, ScanOn, ScanOff, Bitmap

These buttons allow you to switch jog mode and transit speeds. The mode number can be one of the following:
8 = Custom (Uses the label value)
7 = Fast Transit
6 = Medium Transit
5 = Slow Transit
4 = Jog 1"
3 = Jog 0.1"
2 = Jog 0.01"
1 = Jog 0.001"

Default Speed must all be 0 accept for the one chosen to be default. The default must be 1. (see pg 16 for example)

Label sets the text you will actually see on the button when it is drawn on the screen.

"JogMode", 0, 5, 35, 25, 7, 0, "Fast", "", "", -1, -1, "fast.bmp"
"JogMode", 0, 35, 35, 25, 6, 0, "Med", "", "", -1, -1, "med.bmp"
"JogMode", 0, 65, 35, 25, 5, 0, "Slow", "", "", -1, -1, "slow.bmp"
"JogMode", 50, 95, 35, 25, 4, 0, "1", "", "", -1, -1, "1.bmp"
"JogMode", 90, 95, 35, 25, 3, 0, ".1", "", "", -1, -1, "p1.bmp"
"JogMode", 130, 95, 35, 25, 2, 0, ".01", "", "", -1, -1, "p01.bmp"
"JogMode", 170, 95, 35, 25, 8, 0, "10", "", "", -1, -1, "10.bmp"

Frames
Syntax: "Frame", Left, Top, Width, Height, FrameType, Unused, Unused, Unused, Unused, ScanOn, ScanOff, Unused, Group #   Using group frames allows the user to position buttons, labels, and indicators inside of a frame using the upper left corner of the frame as the (0, 0) location. When the frame is moved all buttons in the same group move with it. Groups can be assigned by adding a group number to the end of a "Frame" line, and then adding the group "AuxIn" line. Allows user to draw boxes around screen elements to organize them. The appearance of the frame is determined by FrameType: 1 = Etched Frame.   Following are some examples of how to use the group frames.   "Frame", 210, 0, 115, 125, 1, 0, "", "", "", -1, -1, "", 1 [Group 1 Frame]   "UserBtn", 5, 5, 50, 25, 0, 0, "G90", "G90", "", -1, -1, "", 1 [Group 1]   "UserBtn", 5, 35, 50, 25, 1, 0, "G91", "G91", "", -1, -1, "", 1 [Group 1]   "UserBtn", 5, 65, 50, 25, 2, 0, "Home", "G28", "", -1, -1, "", 1 [Group 1]   "Frame", 325, 0, 45, 170, 1, 0, "", "", "", -1, -1, "", 2 [Group 2 Frame]   "AuxIn", 0, 5, 5, 20, 18, 1, 1, "C1", 2 [Group 2]   "AuxIn", 0, 5, 25, 20, 18, 2, 1, "C2", 2 [Group 2]

AuxIn Indicators
Syntax: "AuxIn", Type(0), Left, Top, Height, Width, Channel(1-32), Color(1-6), Label   Auxin Indicators represent auxiliary inputs for which you want a status display. Channel corresponds to channel number in WINCNC.INI.   The following are the color schemes for the color section of the AuxIn screen line: 1 - On=Green, Off=Red   2 - On=Green, Off=Gray   3 - On=Red, Off=Gray   4 - On=Red, Off=Green   5 - On=Gray, Off=Green   6 - On=Gray, Off=Red   "AuxIn", 0, 340, 60, 18, 20, 1, 1, "Aux-In 1"

WinCNC Controller

Virtual AuxIn Indicators

Syntax: "Virtualin", Type(0), Left, Top, Height, Width, Channel(1-32), Color(1-6), "Label",""

Virtual inputs work in similar fashion to standard aux-inputs, except that virtual input states (on/off) can only be changed by command. Virtual input states are stored and re-loaded when WinCNC closes and re-opens. There are 64 virtual input channels available for use. The following explains configuring and using virtual inputs.

L60 C# - Turn specific virtual input off, # can be 1-64
L61 C# - Turn specific virtual input on, # can be 1-64
M17.3 C# - Run next line if the specified input is on, # can be 1-64

AuxOut Buttons - buttons used to control auxiliary outputs.

Syntax: "AuxOut", Left, Top, Width, Height, Channel, (*see below), Label, AuxOn, AuxOff, ScanOn, ScanOff, Bitmap

Channel should be set to the auxiliary channel you want to control. Label determines the text that is displayed on the button. AuxOn is a G-Code command that turns on your auxiliary output and may utilize macros. AuxOff is the G-Code command that turns off the auxiliary output.

Note: The 0 channel is reserved for laser enable control only! Standard auxout do not use 0.

"AuxOut", 280, 35, 50, 25, 0, 0, "Laser", "L251", "L250", -1, -1, "Laser.bmp"
"AuxOut", 280, 65, 50, 25, 1, 0, "AuxOut 1", "M11C1", "M12C1", -1, -1, "Auxout.bmp"

* - This is normally set to 0. By changing it to 1, it will allow use of the AuxOut when a file is running.

Menu Buttons

Syntax: "MenuBtn", Left, Top, Width, Height, Function#, Unused, Label, Unused, Unused, ScanOn, ScanOff, Bitmap, Group

The menu button can be used to add buttons to the main screen and serve as shortcuts to menu options. The Function# can only be set to 1 for the Settings->Local Coordinates menu option. Function numbers for other menu options will be added in future WinCNC versions.

"MenuBtn", 5, 95, 50, 25, 1, 0, "Set Local", "", "", -1, -1, "", 1

Job Command Buttons

"JobCmd", Left, Top, Width, Height, Command, Unused, Label, Unused, Unused, ScanOn, ScanOff

JobCmd buttons represent non G-Code commands you want to send to WinCNC. The Command parameter can be one of the following:

1 = Abort motion	7 = Manual Digitizing Rapid
2 = Run current command	8 = Manual Digitizing Feed
3 = Pause motion	9 = Manual Digitizing Arc
4 = Refresh keypad file list	10 = Digitizing Close shape
5 = Restart	11 = Digitizing Undo
6 = Single Step	

"JobCmd", 0, 0, 0, 0, 1, 0, "Abort", "", "", 17,0, ""
"JobCmd", 0, 0, 0, 0, 3, 0, "Pause", "", "", 13, 0, ""

Shell Command Buttons

Syntax: "ShellCmdBtn", Left, Top, Width, Height, ButtonNum, Unused, Label, Command, Directory, ScanOn, ScanOff, Bitmap

Shell command buttons allow you to create buttons inside the WinCNC window, which execute outside commands or programs. Any command that can be executed from the Windows START->RUN line can also be run from a shell command button. The following example shows how to create a shell command button for the calculator application that is included in all versions of Windows.

"ShellCmdBtn", 215, 95, 50, 25, 0, 0, "Calc.", "calc.exe", "", -1, -1, ""

User Buttons

Syntax: "Userbtn", Left, Top, Width, Height, ButtonNum, Unused, Label, Command, Unused, ScanOn, ScanOff, Bitmap

Userbtn is used to create customized push buttons that send G-Code whenever they are pressed. The ButtonNum is used internally to identify the button. It is recommended that you use sequential integers starting with 0. The Label parameter determines the text that is actually printed on the button. The Command parameter is G-Code that is executed when the button is pressed.

Userbtn", 225, 5, 50, 25, 0, 0, "G90", "G90", "", -1, -1, "g90.bmp"

Repeat Command Buttons

Syntax: "RptCmd", Left, Top, Width, Height, Command, Unused, Label, Unused, Unused, ScanOn, ScanOff

RptCmd buttons represent non G-Code commands you want to send to WinCNC, using the serial keypad. These buttons are not shown on the main window. RptCmd commands should be used for JobCmds that need repeating key message. The Command parameter can be one of the following:

1 = Increase override
2 = Decrease override
3 = Increase laser power
4 = Decrease laser power

"RptCmd", 0, 0, 0, 0, 1, 0, "Inc Override", "", "", 7, 0, ""
"RptCmd", 0, 0, 0, 0, 2, 0, "Dec Override", "", "", 8, 0, ""
"RptCmd", 0, 0, 0, 0, 3, 0, "Inc Laserp", "", "", 6, 0, ""
"RptCmd", 0, 0, 0, 0, 4, 0, "Dec Laserp", "", "", 5, 0, ""

Key Pad Out Buttons

Syntax: "KeyPadOut", Left, Top, Width, Height, Command, Unused, Label, Unused, Unused, ScanOn, ScanOff

KeyPadOut buttons represent commands you want to send to the keypad. The command parameter can be one of the following:

1 = Scroll keypad file list up
2 = Scroll keypad file list down
3 = Run current keypad file
4 = Refresh keypad file list
5 = Repeats last command
6 = Shifts the keypad axis display up
7 = Shifts the keypad axis display down

"KeyPadOut", 0, 0, 0, 0, 1, 0, "Up", "", "", 2, 0, ""
"KeyPadOut", 0, 0, 0, 0, 2, 0, "Down", "", "", 3, 0, ""

State Buttons

Syntax: "StateBtn", Left, Top, Width, Height, Type, Unused, Label, Command, Unused, ScanOn, ScanOff

State buttons represent the state of a feature on or off. The buttons will stay down when clicked until clicked again to disable the feature. The type sets the class of the feature. The example below displays type 8 which is for the scanner feature. If an L800 or L801 command is typed into the command line, the button will change states to represent the command. The commands are represented left to right, with the left command, "L801" in the example to represent what happens when the feature is disabled, the state button is out in a normal state and can be clicked to turn on the feature. The second command "L800" is the command that is issued when the button is up and is clicked to the down state.

"StateBtn", 335, 25, 50, 20, 8, 0, "Scan", "L801", "L800", -1, -1, ""
State button types are as follows:
0 - Reserved
1 - Reserved
2 - Reserved
3 - Reserved
4 - Velocity mode for pump On/Off
5 - Reverse mode for pump indicator. Indicates if in reverse mode.
6 - Purge mode for pump
7 - Tangential Knife mode On/Off
8 - 3D Probe Scanning On/Off
9 - Smart Bitmap engraving
10 - AutoAuxOut

Axis Controls

Syntax: "AxisControl", Left, Top, Width, Height, Axis, Direction, Unused, Unused, Unused, ScanOn, ScanOff, Bitmap

Axis numbers are zero-based. The letter displayed is determined by the axis given. You can change these letters in WINCNC.INI.

"AxisLabel", 50, 37, 35, 13, 0, 0, "", "", "", -1, -1, "x.bmp"
"AxisLabel", 90, 37, 35, 13, 1, 0, "", "", "", -1, -1, "y.bmp"

WinCNC Controller   **Command Reference**

Command Reference

Parameters in [Brackets] are optional.

XYZWIJ . . .	Axis Specification X# Y# Z#
Axis values are specified with a decimal point. A value with no decimal is read as an integer value. No value is equivalent to specifying 0. Example: XYZ is equivalent to X0Y0Z0	

F	Feed Velocity - F#
Velocity is stored separately for Linear XY, Linear ZW. Arc's Rapid and Feed Velocity is stored separately for Linear Moves. Independent velocities are stored based on the XYZW specified in the line containing the F# command. Velocity is specified in units of measure per minute. Example: F60 sets all axes in the feed rate override command group to a Feed Velocity of 60 (see "vgroup="). G1 X F60 sets the X axis and all other axes grouped with X to a Feed Velocity of 60. F100 X Y sets the X and Y axes to a Feed Velocity of 100. It also sets any other axes that are grouped with either axis to a Feed Velocity of 100. For information about axis grouping, see "vgroup=". Feed Velocity changes affect all axes in that vgroup.	

P	Stored Positions - P#
Move to the selected stored position.	

H	Stored Home Positions - H#
Move to the selected stored Home position	

S	Spindle Speed - S#
Sets the spindle speed to the given value.	

[ ]	Comment
Used to add comments to programs. A closing bracket is optional.	

{ }	Braces
Used to substitute axis or parameter values in a line of G-Code. Internally defined variables as well as constant numbers can be used within the braces. Following is a list of internally defined variables and what they represent.	

Internally Defined Variables

TP1	Tool number, tool changer 1.
TP2	Tool number, tool changer 2.
TN1	Tool number 1 + 10.
TN2	Tool number 2 + 20.
TS1	Tool number 1 + tool changer 1 offset.
TS2	Tool number 2 + tool changer 2 offset.
TH1	Tool height for tool changer 1.
TH2	Tool height for tool changer 2.
TM1	Height of a tool length measure switch. Set using M37.1.
TM2	Height of a second tool length measure switch.
TMX	X position of a tool length measure switch.
TMI	X position of a second tool length measure switch. (In rare cases, all heads cannot reach the same tool measure switch.)
TMY	Y position of a tool length measure switch.
TMJ	Y position of a second tool length measure switch.
TMD	Rapid move distance for auto tool measure.
TMF	Feed rate for auto tool measure.
LZ0	Current G92 value for axis 0.
LZ1	Current G92 value for axis 1.
LZ2	Current G92 value for axis 2.
LZ3	Current G92 value for axis 3.
LZ4	Current G92 value for axis 4.
LZ5	Current G92 value for axis 5.

Internally Defined Variables continued

THM	1 if THC Manual Voltage Mode is enabled, 0 if disabled.
THA	1 if THC Auto Voltage Mode is enabled, 0 if disabled.
THO	The specified output channel used to turn the torch on/off for THC.
THI	The specified input channel used to monitor the "arc good" signal for THC.
THP	1 if pierce height for THC is enabled, 0 if disabled.
THC	1 if cut height for THC is enabled, 0 if disabled.
THZ	The specified pierce height for THC.
THR	The specified cut height for THC.
THS	The specified pierce delay for THC.
THE	The specified end delay for THC.
THD	The specified switch compensation amount.
THL	The specified torch lift amount.
THF	The feed rate specified for THC.

Additional variables can be defined in the CNC.MAC file. Defined variables and constants can be used together with simple math operators. (*) multiplication, (/) division, (+) addition, (-) subtraction.

Example: G0Z{TH1+2} moves Z to tool height 1, plus 2 units.

NOTE: ANY use of braces requires a value or axis spec and a number. The second parameter inside the braces must be a number.

G Codes

G0	Rapid Move - G0 X# Y# Z# W#

Moves to the position specified at Rapid velocity. G0 is modal. After a G0 is executed lines with no G-Code command are executed as a G0.
Example: X1Y1 is equivalent to G0 X1Y1 if mode is G0.

G0.1	Rapid Move with Vertical Lift - G0.1 [X#] [Y#]

First lifts all vertical heads then moves the position specified at Rapid velocity then drops the vertical heads back to their previous positions.

G1	Feed Move - G1 L# X# Y# Z# W#

Moves to the position specified at Feed velocity. G1 is modal. After a G1 is executed lines with no G-Code command are executed as a G1.
Example: X1Y1 is equivalent to G1 X1Y1 if mode is G1
L# is used to allow setting laser power in vector cutting (1-100)

G2	Clockwise Arc - G2 L# X# Y# I# J# Z# W# K#

Moves to the position specified at Feed velocity. I is the X distance to the center point. J is the Y distance to the center point. If no XY move is specified a full circle is cut. If no I or J is specified previous I J values are kept. Any axis spec that is not part of the arc itself is a simulated move with the arc.
L# is used to allow setting laser power in vector cutting (1-100)

G3	Counter Clockwise Arc - G3 L# X# Y# I# J# Z# W# K#

Moves to the position specified at Feed velocity. I is the X distance to the center point. J is the Y distance to the center point. If no XY move is specified a full circle is cut. If no I or J is specified previous I J values are kept. Any axis spec that is not part of the arc itself is a simulated move with the arc.
L# is used to allow setting laser power in vector cutting (1-100)

G4	Dwell - G4 X#

Stops movement for the time specified by the X value in seconds. There is no limit to delay time. If no time is specified then the machine will be stopped until the operator pushes ENTER. Place a comment after the dwell to prompt the operator.
Example: G4 [Ready To Start Section 2]
WARNING: Never use Dwell to stop the machine while changing parts! Instead program a single part and use the TAB key at the Program prompt. This will repeat the last part cut.

G9	Smoothing - G9 S# A#

Used to set the smoothing factor. When the XY direction of motion changes, this setting can reduce the "slowdown" to improve the smoothness of motion. Increasing the S# decreases the slowdown of the machine when turning corners. Too high a factor can result in missed steps or motor stalls. Smoothing can also be set in the WINCNC.INI file using G09=S#.
S# - smoothing factor to be used. (not a specific unit of measure)
A# - used to change the angle used in the velocity matching algorithm. The A# value represents an angle. If a small angle, such as say 20, is specified, the G9=# will not apply to any angle smaller than 20.

G20	Converts unit of measure to inches.
G21	Converts unit of measure to centimeters.
G22	Converts unit of measure to millimeters.
G23	Returns to the last used unit of measure.

G28	Return to machine zero - G28 XYZW

Moves specified axes to Lo Limit for XY - Hi Limit for ZW (these are default, they can be reversed). Moves specified axes to WINCNC.INI specs from limits. Sets all axis positions to 0. All axes are moved if none are specified.

G31	Stop file parsing

Must be used with the M28 command to measure machine position after touching a limit switch.
Example: L91 G1 Z-10 F20 M28 G31 M37 Z1

G40	Cutter Diameter Compensation Off

Turns off cutter diameter compensation.

G41	Cutter Diameter Compensation Left - G41 O#

Compensates for differences in nominal and actual tool diameters. Starts to the left on the first move.
O# is the diameter compensation amount on machines without a tool changer.

G42	Cutter Diameter Compensation Right - G42 O#

Compensates for differences in nominal and actual tool diameters. Starts to the right on the first move.
O# is the diameter compensation amount on machines without a tool changer.

G43	Tool Length Offset On - G43 ZWUV

Sets tool length offsets to the values set by the last M37 command. G43 alone turns all offsets on.

G49	Tool Length Offset Off - G49 ZWUV

Cancels tool length offsets for axis specified. G49 alone turns all offsets off.

G50	Scaling Mode Off

Cancels the scaling mode.

G51	Scaling Mode On - G51 X# Y# Z# I# J# K# C# W#

Enables the scaling factor. Scale each axis type based on the values specified by X# Y# Z#. The object will be centered based on the I# (X center) J# (Y) K# (Z). You can center all axes using the C# line.

G53	Rapid Move - G53 X# Y# Z# W#

Moves to the position specified at rapid velocity, ignoring tool measures and local coordinates. G53 alone will lift all vertical or type 3 axes.

G54	G54 Workspace - Rotational: G54 X# Y#

Select the Z head. If G54 is commanded from G55, G56, or G57 mode, the Z head will be moved to the current XY position.

G55	G55 Workspace - Rotational: G55 X# Y#

Select the W head. If G55 is commanded from G54, G56, or G57 mode, the W head will be moved to the current XY position.

G56	G56 Workspace - Rotational: G56 X# Y#

Select the U head. If G56 is commanded from G54, G55, or G57 mode, the U head will be moved to the current XY position.

G57	G57 Workspace

Select the V head. If G57 is commanded from G54, G55, or G56 mode, the V head will be moved to the current XY position.

G73	Chip Break Cycle - G73 X# Y# Z# R# Q# F#

Moves to XY specified at Rapid velocity. On the first peck, moves to R retract height at Rapid velocity, moves to Q peck depth at F feed velocity, lifts 0.05" at Rapid velocity to break the chip, moves to next peck depth at Feed velocity. The cycle repeats until Z depth is reached. Then moves back to retract height R at Rapid velocity. Without this cycle, some materials can produce a long "stringer" which can become a safety issue to the operator.

G80	End Drill Cycle

Used to disable the G81 Drill Cycle mode. Note: Switching to G0 or G1 will also break out of the drill cycle mode.

G81	Drill Cycle - G81 X# Y# Z# R# F#

Moves to XY specified at Rapid velocity. Moves to R (clearing height) at Rapid velocity. Moves to Z specified at Feed velocity. Moves back to R at Rapid velocity. F allows you to set the feed rate.

G82	Dwell Cycle - G82 X# Y# Z# R# P# F#

Moves to XY specified at Rapid velocity. Moves to R (clearing height) specified at Rapid velocity. Moves to Z specified at Feed velocity. If P is specified, bit pauses that amount of time at the bottom of the hole, then moves back to R at Rapid velocity. P is measured in milliseconds (thousandths of one second). F allows you to set the feed rate.

G83	Peck Drill Cycle - G83 X# Y# Z# R# Q# F#

Moves to the XY specified at Rapid velocity. On the first peck, moves to R retract height at Rapid velocity, moves to peck depth Q at Feed velocity, then moves back to retract height at Rapid velocity. On each subsequent peck, moves at Rapid velocity to within 0.05" of previous peck depth, then moves at Feed velocity to next peck depth. Cycle repeats until Z depth is reached. Then moves back at Rapid velocity to retract height R. The F allows user to set the feed rate.

G90	Absolute Mode

Can be specified with other G-Codes on any line. XYZW values from the current line forward are read as absolute coordinates. IJ values are always relative to the current XY position, not absolute positions regardless of G90/G91 mode.

G91	Relative Mode

Can be specified with other G-Codes on any line. XYZW values from the current line forward are read as relative movements from the current position.

G92	Set Local Coordinates - G92 X# Y# Z# W#

Used to specify a new coordinate system for running absolute mode programs. Use G92 alone to restore the Machine Coordinates. G92 X0 Y0 Z0 W0 sets the current position to zero. G92 then restores the Machine Coordinates values.

**If running a .tap, .mac, .nc etc file, and there is a G92 in the file, and the user wishes to maintain that G92 after the file is done and closed, then a L92 must be added at the end of the file.

G92.1	Shift MZ Coordinates - G92.1 X# Y# Z# W#

Similar to G92 except that the coordinate system produced shifts Machine Zero by the amount specified. This is useful since a given local coordinate system can be set without positioning the head to a certain position first. G92 X10 Y10 sets the absolute position X10 Y10 to X0 Y0 in local coordinates. G92.1 may also be used to restore a single axis to absolute coordinates. G92.1 X0 leaves YZW local coordinates but sets X back to absolute coordinates.

G92.2	Shift LZ Coordinates - G92.2 X# Y# Z# W#

Similar to G92.1 except that the coordinate system shift is added to the current local coordinates instead of replacing them. This is useful for 'jogging' a local position. If the Z head has been set for running a G90 mode file but then needs to set to cut .010" deeper, G92.2 Z-.01 will accomplish this with a single command. Can be useful as a macro.

G92.3	Stores and Restores G92 values

Allows the user to switch back and forth between local and absolute coordinates. Use the G92.3 command without any parameters or axis characters. If G92 positions are set on any axis then they will be stored and cleared. The next G92.3 call will restore the previous G92 values even if they have been cleared by using G92.

G93	Inverse Time Feed Mode

In inverse time feed rate mode, an F word means the move should be completed in [one divided by F number] minutes. For example, if the F number is 2.0, the move should be completed in half a minute. When the inverse time feed rate mode is active, an F word must appear on every line which has a G1 motion, and an F word on a line that does not have G1 is ignored. Being in inverse time feed rate mode does not affect G0 motions. It is an error if: inverse time feed rate mode is active and a line with G1 does not have an F word.

G94	Cancels the Inverse Time Feed Mode

Turns off the Inverse Time Feed Mode

L Codes

L1	Spindle Speed Control using the SPIN.MAC macro
Calls the SPIN.MAC macro for spindle speed control. SPIN.MAC must be a file in the WinCNC directory.	

L3	Set Home Position - L3 XYZW
Stores current position of each axis specified. Values specified are ignored.	

L4	Return to Home
Moves each axis specified to the last L3 position stored. Values specified are ignored.	

L5	Set Contouring Velocity - L5 XYZ
Sets the contouring velocity in units per minute.	

L6	Set Acceleration - L6 XYZ
Sets the acceleration in units per minute per second. L6 alone restores the original values.	

L7	Set Acceleration Percentage L7 P#
Sets the acceleration to a percentage of the current acceleration. P# is the percentage (1-100) to set. L7 alone restores the original.	

L8	Turns Backlash Compensation On, Restore INI values for all axes.

Examples:
     L8XY - Turn Backlash Compensation On, Restore INI values
          for specified axes.
     L8X.001 - Turns Backlash Compensation On, Set new value
          for specified axis.	

L9	Turns Backlash Compensation Off

L10	Cut Array - L10 R# C# X# Y# E#

Sets up array cutting. Repeats all code following until the end of the file or another L10 is reached. Program pointer is moved back to initial L10. The head is moved to the next column or row specified by XY at rapid velocity. Cycle repeats until all array points have been cut. L10 specified without RCXY values can be used to end an array cut. Lines that follow will not be repeated. If an 'E' is found on the L10 command line then WinCNC will skip the final move back to the starting position at the end of the array. If no 'E' is found then WinCNC will move back to the starting position at the end of the array.

**A G92XY is required after the L10 Line for the cut array to work
X# is the units to move the x axis before starting the next cut
Y# is the units to move the x axis before starting the next cut
R# is the number of rows
C# is the number of columns

Example: L10 r5 c5 x4 y4 e1
       G92xy
       G90
       G01 x3
       G01 y3
       G01 x0
       G01 y0

L11	Set Axis Mapping - L11 XYZW

Allows re-mapping of axis inputs. Specify the axes that should receive the input from XYZW in that order. Specify L11 alone to return to normal parsing.
Example: L11 XYWZ swaps the W and Z axes, L11 XYZZ moves Z and W together.

L12	Set Axis Mapping - L12 ZW

Allows re-mapping of axis inputs. Specify the axes that should receive the input from ZW in that order. This command works like L11 except that the XY inputs are not used. Specify L12 alone to return to normal parsing.
Example: L12 WZ swaps the W and Z axis, L12 ZZ moves Z and W together.

L13	Set Axis Mapping - L13 XYZW

Allows re-mapping of axis on a pulse basis. This means when the controlling axis takes a step, the slave will follow in that direction
Specify L13 alone to return to normal mapping.
Examples:
L13 ZW (Z is controlling axis, W follows Z)
L13 WZ (W is controlling axis, Z follows W)
L13 XY (X is controlling axis, Y follows X)

L20	Enable Soft Limits - L20 [X1, X2]

Enable Soft Limit and Boundary Checking: L20 X1 enables Boundaries only. L20 X2 enables Soft Limits only. L20 alone enables both.

L21	Disable Soft Limits

Disable Soft Limit and Boundary Checking Soft Limits and Boundaries are used to define a cutting area which is checked during parsing of a file or command. This effectively keeps the machine from moving out of a defined area. L21 disables Soft Limit and Boundary checking.

L22	Set Lo Boundaries - L22 X# Y# Z# W#

Set Lo Boundaries to values specified. If no values are given then all Lo Boundaries are set to WINCNC.INI defaults.

L22.1	Restore Lo Boundaries - L22.1 XYZW

Restores default Lo Boundaries to the specified axes.

L23	Set Hi Boundaries - L23 X# Y# Z# W#

Set Hi Boundaries to values specified. If no values are given then all Hi Boundaries are set to WINCNC.INI defaults.

L23.1	Restore Hi Boundaries - L23.1 XYZW

Restores default Hi boundaries to the specified axes.

L28	Set Position - L28 X# Y# Z# W#

Sets Machine Coordinates. Machine Coordinates specify the fixed machine zero point for your machine. L28 is normally used only to write a custom homing program. L28 does not move the machine, but sets the current position to the values specified. Use G92 to set a local coordinate system for running absolute mode programs from any table position.

L29	Clear Home Flags - L29 XYZ

Clears all require home flags. Axes can be used with the command to clear require home flags only on the specified axes.

L29.1	Ignore Home Flags

Ignores require home flags for all axes during execution of the currently running file.

L30	Knife Mode On

Enables support for the Knife Mode.

L31	Knife Mode Off

Disables support for the Knife Mode.

L32	Saw Mode On

Enables support for the Saw Mode.

L33	Saw Mode Off

Disables support for the Saw Mode.

L34	AutoAuxOut Enable - L34 D#

Enables an auxiliary output that will turn on during G1/G2/G3 moves, and will turn off for all other moves. L34 D# sets the dwell time in seconds used when the channel is set to on and L35 D# sets the dwell time in seconds when the channel is set to off. When the 'D' parameter is used with the L34 command the "AutoAuxOut" mode does not change. L34 parameters are set in the WINCNC.INI file by the "autoaux=" line.

L35	AutoAuxOut Disable - L35 D#

Disables the auxiliary output that is turned on by the L34 command. L35 D# sets the dwell time in seconds when the channel is set to off. When the 'D' parameter is used with the L35 command the "AutoAuxOut" mode does not change.

L36	Torch Height Control On
L36.1	Auto Voltage Set Mode On
L36.2	Reset Auto Voltage Set Status
L37	Torch Height Control Off
L37.1	Auto Voltage Set Mode Off
L38	Torch On
L39	Torch Off

L40	Feed Rate Override - L40 P#

Allows user to set the feed rate override by command. P must be within the bounds specified in the INI file. If not, the command is ignored. The L40 command without a P# resets the feed rate to the default.

L41	Shell Command - L41 C:\Path\Program.exe

Allows the user to run an executable program from within WinCNC. NOTE: There is no error reporting for this command. If the path or filename are incorrect the command will fail without aborting.

L51	Set D/A - L51 V# P# S#

Sets the output voltage for DA. The V parameter is used to set the DA by voltage, the P to set the DA by a Percentage, and the S is used to set the DA by spindle speed. ONLY 1 PARAMETER CAN BE USED.

L53	Set Vertical Head Height

Raises all vertical heads as high as possible while keeping the bits at equal height.

L55	Pump Velocity Mode On
Enables velocity matched pump control.	

L56	Pump Purge Mode On
Sets pump to purge mode.	

L57	Pump Off
Stops the pump.	

L58	Pump Off with Auto Reverse
Stops the pump, using auto reverse.	

L59	Set Up Pump Max - L59 M#
Sets the pump max to the specified M parameter.	

L60	Turn Virtual Input Off - L60 C#
Turn specified virtual input off, # can be 1-64.	

L61	Turn Virtual Input On - L61 C#
Turn specified virtual input on, # can be 1-64.	

L70	Re-Enter Last Command
When L70 is used in a file, the last used command will automatically be re-entered on the command line after file completion.	

L82	Measures the Surface - L82 Z#
L82 Z# measures the surface with the probe and sets the measured point to the Z value given. L82 alone will set the measured point to Z0.	

L82.1	Sets the Surface for Probe Digitization
Sets a G92 for vertical axis based on the L82.3 calibration.	

L82.2	Moves the Surface for Probe Digitization
Adjusts G92 positions set by the last L82.1 based on a new measurement.	

L82.3	Calibrate Digitizing Probe

Calibrates, or stores, the distance from the probe contact point to the table surface.

L83	Measures Z Depth - L83 X# Y#

L83 command will do a one-shot Z probe at the specified location, and record the XYZ position values in a comma delimited text file named SCAN.CSV. Where X# and Y# specify the (X,Y) coordinates to move to before doing the depth probe.

L90	One Time Absolute

L90 is used in a command to specify G90 coordinates on one line without changing program mode to G90.

L91	One Time Relative

L91 is used in a command to specify G91 coordinates on one line without changing program to G91.

L92	Cancel G92 Restore

The current G92 Local Coordinate system is saved every time a program is started or a subprogram is called using M98. When the program is finished the original G92 system is restored. This is not desirable if the program is being used to preset a system configuration for other programs. L92 causes the normal G92 coordinate restore to be skipped after the program containing the L92 is finished.

L110	Tool Change - L110 T#

Sets current tool number for machines with an automatic tool changer. The tool number is available as {TP1} in subsequent commands, or {TN1} and {TN2} for dual tool changer machines. The stored tool measure is reloaded for the new tool.
(See M37 command for storing tool measures)

L111	Test Tool Number - L111 T#

Run next line if tool number is equal to T#.

L111.1	Test Tool Number with Dual Tool Changers - L111.1 T#

Run next line if both tool numbers are equal to T#.

L112	Test Tool Number - L112 T#.
Skip next line if tool number is equal to T#.	

L113	Spindle Speed Conditions - L113 L# H# V#
Skip the next line if the V# is not greater than the L# and less than the H#.	

L120	Tool Change - L120 T#
Sets current tool number for machines with an automatic tool changer. The tool number is available as {TP2} in subsequent commands.	

L201	Set 201 Signal Command - L201 FileName T# C# S#
Sets the command used with the 201 signal type.	

L202	Set 202 Signal Command - L202 [Command]
Sets the command used with the 202 signal type.	

L204	Activate Signals
L204 C# activates signals and enables the specified channel. L204 activates all signals and enables	

L205	Deactivate signals
L205 C# deactivates signals and enables the specified channel. L205 deactivates all signals and enables.	

L210	Select Alternate Low Limits - L210 XYZW
Selects axis to use alternate low limits.	

L210.1	Select Alternate Low Limit Pin - L210.1 XYC#
Selects alternate low limit pin from the specified C# channel. If no channel is specified, the original settings are restored.	

L211	Select Alternate High Limits - L211 XYZW
Selects axis to use alternate high limits.	

L211.1	Select Alternate High Limit Pin - L211.1 XYC#
Selects alternate high limit pin from the specified C# channel. If no channel is specified, the original settings are restored.	

L212	Select Primary Limits
Selects the primary limits for all axes.	

L213	Dual Axis, Select Main - L213 XYZW
Selects the main motor on a dual motor axis.	

L214	Dual Axis, Select Secondary - L214 XYZW
Selects the secondary motor on a dual motor axis.	

L215	Dual Axis, Select Both Axes - L215 XYZW
Selects both motors on a dual motor axis.	

L249	Laser Power Setting - L249 S# P# D# O# M# I#
Using the L249 command allows you to specify the laser power. Example: L249 P50 (sets laser power to 50) S# - 0 or 1 sets smart engrave off/on. P# - sets the laser pulse width in percentage (1-100). D# - sets the laser pulse duty cycle (1-100). O# - sets the bitmap engraving offset. M# - sets the bitmap engraving margin. I# - reverses color polarity.	

L250	Vector Laser Off
Disables vector laser cutting.	

L251	Vector Laser On - L251 P# D# L#

Enables vector laser cutting.
P# - laser power (1-100).
D# - sets the distance for the distance based vector laser firing.
L# - sets the distance for the off signal when perforating.

L251.1	Laser Dwell - L251.1 D#

D# - time in seconds for dwell.

L252	Bitmap Engraving

L252.1 Stores rotation for future use.

L253	Bitmap Engraving rotated 90 degrees clockwise.

L253.1 Stores rotation for future use.

L254	Bitmap Engraving rotated 180 degrees clockwise.

L254.1 Stores rotation for future use.

L255	Bitmap Engraving rotated 270 degrees clockwise.

L255.1 Stores rotation for future use.

L256	Offset for bitmap engraving row alignment

Offset on positive motions to compensate for laser firing delays to line up bitmap rows during dual motion engraving.
R# sets the radius used for rotary engraving in the current unit of measure.

L300	Lock Axis - L300 XYZW

Locks specified axis motion. Any motion commanded to a locked axis will be ignored.

L301	Unlock Axis - L301 XYZW

Unlocks specified axis motion.

L400	Create Home File

Creates a home file at the machine's current position.

L401	Recall Positions

Moves all axes to the positions where they were at previous to the last file ran.

L500	Store Absolute Positions - L500 [X, Y, Z...] [P H T] L# C# N"Name"

[X, Y, Z...] - axes to set positions for if none specified then all axes positions are stored.
[P H T] - type of position (P)osition, (H)ome Position, (T)ool Position. Can only use one value.
L# - 0 no vertical head lift, 1 include head lift.
C# - Clearance for vertical heads if L1.
N"Name" - Name for specified position. < 16 characters long.

L500.2	- L500.2 T# N# A# C# O# H# M#

T# specifies the tool number.
N# specifies the nominal diameter for the tool.
A# specifies the actual diameter for the tool.
C# specifies the diameter compensation for the tool.
O# specifies the height offset for the tool.
H# specifies the tool height.
M# specifies the maximum current allowed for the tool.
This command sets up a tool library instead of using the dialog boxes.

L501	Store Local Positions - L501 [X, Y, Z...] [P H T] L# C# N"Name"

[X, Y, Z...] - axes to set positions for if none specified then all axes positions are stored.
[P H T] - type of position (P)osition, (H)ome Position, (T)ool Position. Can only use one value.
L# - 0 no vertical head lift, 1 include head lift. Default L1.
C# - Clearance for vertical heads if L1.
N"Name" - Name for specified position. < 16 characters long.

L600	Disable Mapping Adjustments

Disable mapping adjustment for the specified axis. If no axis is specified mapping adjustments for all axes are disabled.

L601	Enable Mapping Adjustments

Enable mapping adjustment for the specified axis. If no axis is specified mapping adjustments for all axes are enabled.

L602	Start Mapping Setup

Starts mapping setup for specified axis. First axis specifies the axis to map and the second axis specifies the axis to adjust. The axis map will automatically move to the start map position. No axis spec stops the mapping setup.

L603	Store Current Adjustment and Move - L603 {NPSE}

Stores the current adjustment and moves the map axis to the specified map increment. N is next, P is previous, S is start, and E is end. The default move is to the next map increment position.

L604	Move Map Axis By Increment - L604 {NPSE}

Moves the map axis to the specified map increment. N is next, P is previous, S is start, and E is end. The default move is to the next map increment position.

L605	Store Current Adjustment

Stores the current adjustment.

L800	Disable Probe

Disables the probe.

L801	Enable Probe

Enables the probe.

When probe is enabled (L801) it acts similarly to a limit switch. When it comes in contact with an object, it will stop motion. If it is disabled (L800) it no longer functions.

L802	Scan Given Rectangle - L802 X# Y# I#

Scans a given rectangle XY with an increment of I in and up/down, and side-to-side style scan. (Use automatic scan)

L803	Scan Given Rectangle - L803 X# Y# I#

Scans a given rectangle XY with an increment of I in an up/down style scan.

L810	Scan Slice (Drop-off Detection) - L810 FILENAME. EXT X# Y# I# K#

Digitizes a slice (line), either in X direction, or Y direction and saves the points to a text file. If a drop-off is encountered, or distance stated in X or Y is reached, the slice is saved and the scan is ended. **Only one axis can be scanned at a time: X or Y.** Uses up/down, and side-to-side probe type.
FILENAME.EXT - replace with your filename. **Do not surround with quotation marks.**
X# - the actual X axis distance that you want the probe to travel, looking for an edge. If it finds one, the file is saved and the scan ends.
Y# - works like X#, except on the Y axis.
I# - the maximum increment for the horizontal axis.
K# - the vertical increment (how high the probe raises).

L820	Disable axisadj

Disables automatic axis position adjustment.

L821	Enable axisadj

Enables automatic axis position adjustment.

L910	Enable G-Code Line Display

Enabling G-Code display when running large G-Code files can increase CPU and memory usage resulting in worse overall computer system performance.

L911	Disable G-Code Line Display

Disabling G-Code display when running large G-Code files can reduce CPU and memory usage resulting in better overall computer system performance.

M Codes

M11	Auxiliary Output On - M11 C#

Controls auxiliary outputs. Channel specified by # is turned on.
Port and bit addresses are set up in WINCNC.INI
Note: For use with a Drill Bank
A d# can be added to the M11 command for use with a drill bank.
The d# is the binary number of channels up to the n# specified in the drill bank line to turn on.
Example: M11D7 would turn on the first 3 channels of a ten channel drill bank.

M12	Auxiliary Output Off - M12 C#

Controls auxiliary outputs. Channel specified by # is turned off.
Port and bit addresses are set up in WINCNC.INI.

M13	Auxiliary Output On - M13 C#

Same as M11, except the output is turned on without pausing motion.

M14	Auxiliary Output Off - M14 C#

Same as M12, except the output is turned off without pausing motion.

M17	Pause Until Input On - M17 C#

Wait for channel specified by # to come on. Port and bit addresses are set up in WINCNC.INI.

M17.1	Run Next if Input On - M17.1 D# C#

Run next command only if channel specified by # is on. Port and bit addresses are set up in WINCNC.INI.
D# can be 0 or 1. D0 sets the simulate logic to false. D1 sets the simulate logic to true.

M17.2	Run Next if AuxOut is On - M17.2 D# C#

Run the next command only if channel specified by the # is on. Port and bit addresses are set up in WINCNC.INI.
D# can be 0 or 1. D0 sets the simulate logic to false. D1 sets the simulate logic to true.

M17.3	Run Next if Virtual Input On - M17.3 C#

C# is a "virtual input" channel, will run the next line if the specified "virtual input" channel is on and skip the next line if the channel is off. # can be 1-64.

M17.4	Run Next if C# is 1, Otherwise Skip. M17.4 C#

C# can be 0 or 1. 1 will run the next line, 0 will skip the next line.

M18	Pause Until Input Off - M18 C#

Wait for channel specified by # to go off. Port and bit addresses are set up in WINCNC.INI.

M18.1	Run Next if Input Off - M18.1 D# C#

Run next command only if channel specified by # is off. Port and bit addresses are set up in WINCNC.INI
D# can be 0 or 1. D0 sets the simulate logic to false. D1 sets the simulate logic to true.

M18.2	Run Next if AuxOut is off - M18.2 C#

C# is an "auxiliary output" channel, will run the next line if the specified output channel is off and skip the next line if the channel is on. # can be 1-64

M18.3	Run Next if Virtual Input is Off - M18.3 C#

C# is a "virtual input" channel, will run the next line if the specified "virtual input" channel is off and skip the next line if the channel is on. # can be 1-64.

M18.4	Run Next Line If C# is 0, Otherwise Skip - M18.4 C#

C# can be 0 or 1. 0 will run the next line, 1 will skip the next line.

M28	Disable Limit Abort

M28 is used to write your own home program instead of using G28 to home the machine. WinCNC normally aborts a program when a limit switch is encountered. Adding M28 to a G0 or G1 command disables the program abort and instead stops the limited axis while allowing other axes to continue. Limit Cushions are disabled while using M28 so any move done with M28 should be done slowly to avoid machine damage. M28 must be specified on each line separately.

M29	Disable Limit Abort

M29 is used to write your own home program instead of using G28 to home the machine. Adding M29 to a G0 or G1 command disables the program abort and instead stops the current G-Code line but continues on the following line. M29 must be specified on each line separately.

M37	Measure Tool - M37 O# Z# W# H# T#

M37 is used to set Tool Length Offsets. The end of the tool is first set to a known position. M37 is then specified to set the tool length offset.
O# manually specifies the tool height offset.
H# is the tool number of the currently loaded tool. Use this with mode 0 only, M0 on the ATC line in the WinCNC.ini file.
T# is the tool number of the currently loaded tool. UUse this with mode 1 only, M1 on the ATC line in the WinCNC.ini file.
Example: Jog the Z head to the spoil board. Enter M37Z0 to the Z work coordinate to 0. Once tool length offsets are set, Boundaries may be used to insure that the table is not routed by accident. Normally zero values are set to the top of the spoil board. Use G92 to set material thickness if programs reference zero from the top of material. This eliminates re-measuring tool for different thickness materials. G43 and G49 are used to enable and disable offsets.

M37.1	Measure Tool for Calibrating G37 - M37.1 Z# W#

M37.1 is used to store a tool measure in the TM1 variable. This command is used primarily with calibrating G37 for automatic tool measuring.

M37.2	Measure Tool for Calibrating G37 - M37.2 Z# W#

M37.2 is used to store a tool measure in the TM2 variable. This command is used primarily with calibrating G37 for automatic tool measuring.

M37.3	Shift Workspace Offsets M37.3 S# H# T# M# X# Y# Z# ...

M37.3 is used to shift workspace offsets for tool measures, tool measure switch heights, or individual axis.

S# is the shift amount to be used for tool measures and tool measure switch positions.

H# is the H register number to be shifted by the shift amount. H0 will shift all H registers.

T# is the tool number height to be shifted by the shift amount. T0 will shift all tool numbers

M# is the measure switch height to be shifted by the shift amount. M0 will shift both switches.

X# Y# Z# ... shifts each axis workspace offsets by the specified value.

M98	Subprogram Call - M98 <filename>

Run a subprogram. A subprogram may be any G-Code file. Control is passed back to the calling program when finished. To run a G90 mode subprogram from any position on the table use G92X#Y#Z#W# to set the local coordinates at the beginning of the file.

M98.1	Subprogram Call - M98.1 <filename>

Same as M98, except line numbers in the subprogram are not counted. This is most useful for macros that call subprograms.

M99	End Subprogram

Ends a subprogram.

M99.1	Abort

Initiates a commanded program abort.

Macro File Configuration

The CNC.MAC file allows the user to create his or her own com-
mands already built into WinCNC. If macros are not used, a blank
text file should be created in the installation directory, and named
CNC.MAC.

If the user wanted to create shortcut commands to turn auxiliary
output channel 2, which controls the spindle power, on and off the
entries to the CNC.MAC file might be as follows:

M1 = M11C2	[SPINDLE ON]
M2=M12C2	[SPINDLE OFF]

Macro commands defined in the CNC.MAC file can only contain a
single command. If multiple commands are needed within a macro,
you can define a macro that calls a subroutine. Using a subroutine,
you can call multiple commands. The following example shows how
to set a macro to call a subroutine and an example of what that
subroutine would contain. This example demonstrates a macro that
calls a subroutine that will turn on a spindle and wait for it to get up to
speed before moving.

Macro Define:
M3=M98 SPIN.MAC [SPINDLE ON]

Subroutine:
M11C1 [SPINDLE ON]
M17C10 [WAIT FOR SPEED UP]

Another useful feature of the CNC.MAC file is to disable unwanted
M-Codes output by some CAD/CAM packages. WinCNC will ignore
disabled codes. Following is an example of how to disable an un-
wanted M2 code.
M2=[

Also the CNC.MAC file allows the user to override command names
which have already been built into WinCNC. Once you override a
built in command you will have to use a tilde (~) any time you want
to use the built in function. One example of this would be if a user
wanted to use the command G28 to return to 0 on axis XYZ in that
order. The CNC.MAC file entry to accomplish this would be:

Macro Define:

G28=M98 HOME.MAC

This sets the WinCNC G28 command to call the macro instead of its built in function.

In the HOME.MAC file the tilde (~) tells the controller that the built in command has been renamed, and must override the renamed assignment and run its built in function. Keep in mind that by declaring this macro the built in G28 command will be disabled and cannot use its built in function without calling it with a tilde (~). The HOME.MAC file would look similar to this.

Subroutine:

~G28X [Return to 0 on X axis]

~G28Y [Return to 0 on Y axis]

~G28Z [Return to 0 on Z axis]

Note: The .MAC extension of the subroutine file names is NOT required, accept for the CNC.MAC file. That extension can be used if desired, but was used here only as an example.

Messages and Error Codes

Error - Line Too Long
Input line more than 256 characters.

Error - Unsupported G Code
G Code in input line which is not supported.

Error - Unsupported L Code
L Code in input line that is not supported or ignored.

Error - Unsupported M Code
M Code in input line which is not supported.

Error - Multiple Commands
Input line contains more than one command.

Error - Syntax - Command or File Not Found
Non-supported code or invalid file name entered.

Error - Arc Radius
Distance from start point to center is not equal to distance from endpoint to center. Arc Radius errors can occur from running a G2 or G3 intended for G91 mode in G90 mode and vice versa.

Error - Soft Limit Exceeded
Move specified would result in the head being moved outside the soft limits.

Error - Boundary Exceeded
Move specified would result in the head being moved outside the workspace boundaries.

Error - Acceleration Out of Bounds
Acceleration rate too high for resolution. Maximum (acc*res) in steps is 65535.

Error - Illegal Value

The value specified is invalid.

Error - Unknown Position Specified

The position specified is invalid.

Error - Subprogram Nesting Too Deep

Too many nested subprogram calls.

Error - L10 in Subprogram

L10 command found in subprogram.

Error - File Not Found

The specified file name cannot be found.

Error - Parameter Not Specified

A required parameter is not specified.

Error - Unknown

An unknown error has occurred.

Error - Memory Error

Insufficient or bad memory.

Error - User Level 5 Required

A Level 5 user license is required.

Error - Invalid L10 Command

L10 command has invalid parameters.

Error - Invalid P1 Command

P1 command has invalid parameters.

Error - Command Not Implemented

This command has not yet been fully implemented.

Error - Tool Changer Not Enabled

Tool changer is not set up in the WINCNC.INI file.

Error - Parameter Out of Range

Specified parameter is out of range.

Error - D/A Not Enabled

D/A channel is not set up in the WINCNC.INI file.

Error - Arc Too Small

Specified arc move is too small.

Error - Device Not enabled

The requested device is not enabled.

Error - Could Not Open File

Requested file could be opened. May be in use by another program.

Error - Pump Control Not Enabled

The pump control feature has not been enabled.

Error - Soft Limits Not Enabled

Softlimits are set as disabled.

Error - Tangential Knife Not Enabled

The tangential knife feature is not enabled.

Error - Forced Limit Exceeded

Move specified would result in the head being moved outside the forced limits.

Error - Digitizer Probe Not Enabled

The digitizer probe feature has not been enabled.

Error - Arcs Disabled When X or Y scaled

When scaling is enabled, arcs are not supported.

Error - Rotating Saw Not Enabled

The rotating saw feature is not enabled.

Error - Macro Expansion Error

Occurs when macro commands exceed 256 characters.

Aborted - Limit Switch

Limit switch encountered while running.

Aborted - Aborted By User

Escape button pushed while running.

Key Not Found

The software key cannot be found.

Wrong Key Found

The software key found is incorrect.

Key Error #

A software key read error has occurred.

Errors Reading WINCNC.INI

The WINCNC.INI file contains errors or is not found.

Errors Reading CNCSCRN.INI

The CNCSCRN.INI file contains errors or is not found.

CNC.MAC Not Found
The CNC.MAC file is not found.

Upgrade needed for Key Version 5, or 6 Axis card
And upgrade of the daugtherboard is needed.

Pausing.....
Motion is pausing.

Continuing.....
Motion is continuing.

Finished Running "Filename"
The file "Filename" is complete or has been aborted.

Runtime: #d Minute(s) and #f Second(s)
Run time for the last file in minutes and seconds.

File Not Found
A specified file cannot be found.

Error - Demo Count Exceeded
Demo mode only. 25 commands have been completed.

Error - Helical Interpolation Not Supported
No Z or W moves during an arc.

Escape Button Pushed While Running
User has aborted motion with the ESC key.

Error - Not a BMP File
Tried to open a non-bmp file for bitmap engraving.

Error - BMP File Invalid Header

Tried to open an incompatible bmp file.

Error - 2-Bit Bitmap Engrave Not Enabled

Tried to open a 2-Bit bitmap without the feature being enabled in the software.

Error - 8-Bit Bitmap Engrave Not Enabled

Tried to open an 8-bit bitmap without the feature being enabled in the software.

Error - Vector Laser Not Enabled

Vector laser feature not enabled in the software.

Error - Auto-AuxOut Not Enabled

User has tried to use an Auto-AuxOut without the "autoaux=" line being set in the WINCNC.INI file.

Error - Depth Must Be Lower Than Retract

Tried to retract a tool to a height that was below current depth.

Error - SST Drive Control Not Enabled

User tried to use the SST Drive and the drive had not been initialized. See "sst=" and "sstnode=".

Error - SST Drive Communications Failed

Drives resolution (feed or rapid) could not be set.

Error - Unknown

An error has occurred that is undefined in WinCNC.

Keypad

WinCNC Controller Level 4 and above includes a keypad to allow transit and jog of the machine from up to 20 feet from the computer. Plug the keypad into the COM2: port on your computer or change settings in WINCNC.INI to select another available COM port.

There are three types of keypads that can be used with the WinCNC Controller. Type one is an input only keypad which is a serial number pad, just like the one found on the far right of a standard keyboard. Each key can be configured individually in the CNCSCRN.INI file, using the ScanOn and ScanOff parameters. The type two keypad is an input/output keypad with a 20x4 LCD screen display. When this type of keypad is defined X, Y, and Z axis information is displayed on the keypad screen. This keypad can also be configured to browse and select files from the directory last used by the WinCNC Controller. Configuration for this keypad is done in the CNCSCRN.INI file, just as for keypad type one. To use the file browsing capabilities of keypad type two you will need to setup three "KeyPadOut" buttons, as shown in the previous CNCSCRN. INI Settings section. The type 3 is the 2Tech brand keypad with LCD display and optional emergency stop. This keypad would be configured the same way as the type 2 keypads.

NOTE: When defining keypad buttons it is important to set the ScanOff code to zero, only for buttons that you wish to use with the keypad. If this is not done properly a button can be pressed, but will not be released. When configuring the keypad for use with jog or transit keys, this is particularly important because the motion will not stop when the key is released. It is recommended that the keypad setup is done after you have verified that all limit switches are working properly.

Below is a list of scan codes for both types of compatible keypads.

Keypad Type 1: without LCD
WINCNC.INI Setting: keypad=c[com port #] b1200 p1

Key	OnCode	OffCode
1	96	64
2	97	65
3	98	66
4	99	67
5	100	68
6	101	69
7	102	70
8	103	71
9	104	72
ESC	105	73
/	106	74
*	107	75
-	108	76
+	109	77
.	110	78
0	111	79
ENTER	116	84
NUMLOCK	121	89

Keypad Type 2: with LCD

WINCNC.INI Setting: keypad=c[com port #] b9600 p2

Key	OnCode	Shift+Key OnCode
T1	1	51
T2	2	52
T3	3	53
T4	4	54
Left Arrow	5	55
Right Arrow	6	56
Up Arrow	7	57
Down Arrow	8	58
ESC	9	59
1	10	60
2	11	61
3	12	62
SPACE	13	63
4	14	64
5	15	65
6	16	66
.	17	67
7	18	68
8	19	69
9	20	70
DEL	21	71
0	22	72
ENTER	23	73

Note: OffCodes on Keypad Type 2 with LCD are all 0. The "Shift+Key OnCode" refers to holding the Shift Key on the keypad down while pressing the corresponding key.

Keypad Type 3: with LCD
WINCNC.INI setting: keypad=c[com port #] b9600 p3

Key	OnCode	Key	OnCode
CTRL 1	0	CTRL	29
CTRL A	1	CTRL =	30
CTRL B	2	CTRL -	31
CTRL C	3	SPACE	32
CTRL D	4	!	33
CTRL E	5	"	34
CTRL F	6	#	35
CTRL G	7	$	36
CTRL H	8	%	37
CTRL I	9	&	38
CTRL J	10	'	39
CTRL K	11	(	40
CTRL L	12	)	41
CTRL M	13	*	42
CTRL N	14	+	43
CTRL O	15	,	44
CTRL P	16	-	45
CTRL Q	17	.	46
CTRL R	18	/	47
CTRL S	19	0	48
CTRL T	20	1	49
CTRL U	21	2	50
CTRL V	22	3	51
CTRL W	23	4	52
CTRL X	24	5	53
CTRL Y	25	6	54
CTRL Z	26	7	55
ESC	27	8	56
CTRL <	28	9	57

Keypad Type 3: with LCD continued

Key	OnCode
:	58
;	59
<	60
=	61
>	62
?	63
@	64
A	65
B	66
C	67
D	68
E	69
F	70
G	71
H	72
I	73
J	74
K	75
L	76
M	77
N	78
O	79
P	80
Q	81
R	82
S	83
T	84
U	85
V	86
W	87

Key	OnCode
X	88
Y	89
Z	90
[	91
\	92
]	93
^	94
_	95
`	96
a	97
b	98
c	99
d	100
e	101
f	102
g	103
h	104
i	105
j	106
k	107
l	108
m	109
n	110
o	111
p	112
q	113
r	114
s	115
t	116
u	117

Keypad Type 3: with LCD continued

Key	OnCode
v	118
w	119
x	120
y	121
z	122
{	123
|	124
}	125
~	126
DEL	127

PCI 7200 37 Pin Connector CN1:

Pin	Signal		Pin	Signal
Pin #1	P0 B0		Pin #20	P4 B0
Pin #2	P0 B1		Pin #21	P4 B1
Pin #3	P0 B2		Pin #22	P4 B2
Pin #4	P0 B3		Pin #23	P4 B3
Pin #5	P0 B4		Pin #24	P4 B4
Pin #6	P0 B5		Pin #25	P4 B5
Pin #7	P0 B6		Pin #26	P4 B6
Pin #8	P0 B7		Pin #27	P4 B7
Pin #9	P1 B0		Pin #28	P5 B0
Pin #10	P1 B1		Pin #29	P5 B1
Pin #11	P1 B2		Pin #30	P5 B2
Pin #12	P1 B3		Pin #31	P5 B3
Pin #13	P1 B4		Pin #32	P5 B4
Pin #14	P1 B5		Pin #33	P5 B5
Pin #15	P1 B6		Pin #34	P5 B6
Pin #16	P1 B7		Pin #35	P5 B7
Pin #17	+5VDC		Pin #36	Ground
Pin #18	N/C		Pin #37	N/C
Pin #19	N/C			

Aux. Inputs (Pins #1–#16)

Aux. Outputs (Pins #20–#35)

6-Axis 37-Pin Daughter Board Connector CN2:

Step/Direction
Analog Spindle Speed Only
Aux. Inputs

Pin #1	P2 B6
Pin #2	P2 B7
Pin #3	P3 B0
Pin #4	P3 B1
Pin #5	P3 B2
Pin #6	P3 B3
Pin #7	P3 B4
Pin #8	P3 B5
Pin #9	P3 B6
Pin #10	P3 B7
Pin #11	Pump / THC / Freq. Output
Pin #12	D/A Ground
Pin #13	D/A Output
Pin #14	D/A 5V+ / 10V+ DC IN
Pin #15	Pump Reverse
Pin #16	P0 D5	Axis 6 Dir
Pin #17	P0 S5	Axis 6 Step/Laser
Pin #18	P0 D4	Axis 5 Dir
Pin #19	P0 S4	Axis 5 Step

Pin #20	+5VDC Out
Pin #21	Ground
Pin #22	P7 B7
Pin #23	P7 B6
Pin #24	P7 B5
Pin #25	P7 B4
Pin #26	P2 B5
Pin #27	P2 B4
Pin #28	P2 B3
Pin #29	P2 B2
Pin #30	P0 D3	Axis 4 Dir
Pin #31	P0 S3	Axis 4 Step
Pin #32	P0 D2	Axis 3 Dir
Pin #33	P0 S2	Axis 3 Step
Pin #34	P0 D1	Axis 2 Dir
Pin #35	P0 S1	Axis 2 Step
Pin #36	P0 D0	Axis 1 Dir
Pin #37	P0 S0	Axis 1 Step

Aux. Outputs
Aux. Inputs
Step/Direction

6-Axis 25-Pin Daughter Board Connector CN2:

	Pin #	Signal
Aux. Outputs	Pin #1	P7 B6
	Pin #2	P7 B5
	Pin #3	
	Pin #4	
Inputs	Pin #5	P3 B5
	Pin #6	P2 B7
	Pin #7	P3 B0
	Pin #8	P2 B5
	Pin #9	Dir P0 D3
Step/Direction	Pin #10	Step P0 S0
	Pin #11	Step P0 S1
	Pin #12	Step P0 S2
	Pin #13	Step P0 S3

Pin #	Signal	
Pin #14	P7 B7	Aux. Output
Pin #15	P3 B1	
Pin #16	P3 B2	
Pin #17	P3 B3	
Pin #18	P3 B4	Inputs
Pin #19	Dir P0 D0	
Pin #20	Dir P0 D1	
Pin #21	Dir P0 D2	Step/Direction
Pin #22	+5VDC	+5VDC
Pin #23	+5VDC	
Pin #24	Ground	Ground
Pin #25	Ground	

PCI 24H:

24/CTR3	24 & 24H			37	PORT A 0
CLK 0	GND	19		36	PORT A 1
GATE 0	+5V	18		35	PORT A 2
OUT 0	GND	17		34	PORT A 3
CLK 1	+12V	16		33	PORT A 4
GATE 1	GND	15		32	PORT A 5
OUT 1	-12V	14		31	PORT A 6
CLK 2	GND	13		30	PORT A 7
GATE 2	-5V	12		29	PORT C 0
OUT 2	GND	11		28	PORT C 1
	PORT B 0	10		27	PORT C 2
	PORT B 1	9		26	PORT C 3
	PORT B 2	8		25	PORT C 4
	PORT B 3	7		24	PORT C 5
	PORT B 4	6		23	PORT C 6
	PORT B 5	5		22	PORT C 7
	PORT B 6	4		21	GND
	PORT B 7	3		20	+5V
	IR ENABLE	2			
	IR INPUT	1			

PCI 48H:

BASE + 0, 8, 16, 24				
GND	50	● ●	49	+5V
C0	48	● ●	47	C1
C2	46	● ●	45	C3
C4	44	● ●	43	C5
C6	42	● ●	41	C7
B0	40	● ●	39	B1
B2	38	● ●	37	B3
B4	36	● ●	35	B5
B6	34	● ●	33	B7
A0	32	● ●	31	A1
A2	30	● ●	29	A3
A4	28	● ●	27	A5
A6	26	● ●	25	A7
C0	24	● ●	23	C1
C2	22	● ●	21	C3
C4	20	● ●	19	C5
C6	18	● ●	17	C7
B0	16	● ●	15	B1
B2	14	● ●	13	B3
B4	12	● ●	11	B5
B6	10	● ●	9	B7
A0	8	● ●	7	A1
A2	6	● ●	5	A3
A4	4	● ●	3	A5
A6	2	● ●	1	A7

BASE + 4, 12, 20, 28

WinCNC Controller

This page is intentionally left blank.

Symbols

A

B

C

D

E

F